

This is new in Smap3D Plant Design 2021 for Solid Edge

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

Contents:

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





Table of Contents

GENERAL IMPROVEMENTS	4
Supported CAD versions	4
LICENSING IN VERSION 2021 REQUIRED	5
Product licenses from version 2020 still valid	5
INSTALLATION OF THE APPLICATIONS TOR C:\PROGRAM FILES	6
Save button only active after changes	6
New option, not delete redundant pipe files	7
New graphical layout of Smap3D online Help	
REMINDER: AUTOMATIC LOG FILES DURING INSTALLATION	8
NEW IN SMAP3D SCANTOCAD	9
SELECTION AND CAD EXPORT FROM THE SPHERICAL VIEWS	9
NEW IN SMAP3D P&ID	10
FORMULA EDITOR SUPPORTS MATHEMATICAL OPERATIONS	
SUPPORT OF MICROSOFT ACCESS MDB FILES	
New option Component ID	
USE OF DATA SYMBOLS FOR EQUIPMENT TABLES	
OBJECT LISTER – INDIVIDUAL DATA FIELDS DISPLAY	
New NON-standard conform aspect delimiter	
COMPARE DIAMETER SUPPORTS FRACTIONAL NUMBERS	
New option to allow writing to symbol data fields from DB	
New function "Move to other layer"	
NEW OPTION WHEN SWITCHING LAYER WITH ALT KEY	
LINE LINK REFERENCE TO UNIT PAGES POSSIBLE	
New special function Check for Unique P&ID ID's	
SETTINGS FOR SELECT PAGES ARE NOW STORED IN PDFSET	
NAME LIST IN PARTS LISTS SUPPORTS ASPECTS	
Text translator supports Pretexts	
DESCRIPTION IN ALL LIST SETTINGS	
New language Italian	
New language Chinese Traditional/Taiwanese	
NEW IN COORDINATE SYSTEM WIZARD	33
New INPUT FIELD FOR OUTSIDE DIAMETER	
New end treatment Welding nipple	
New end treatment Custom Gasket	
NEW IN SMAP3D PIPING	38
New QuickPlace method Custom Gasket	
NEW QUICK PLACE METHOD WELDING NIPPLE	
New Function Define Cut piece allowance	
WORKING WITH ANGLED BRANCH DUMMY PARTS	
REPLACE PART FUNCTION SUPPORTS COMPONENT PACKAGES	
NEW OPTION TO AUTOMATICALLY SPLIT A SKETCH LINE	
MATERIAL FROM TEMPLATE IS USED FOR PIPELINE PATH	
FILE PROPERTIES AND MATERIAL FROM TEMPLATE IS USED FOR BASIC ENGINEERING TUBES	54
NEW OPTION TO HIDE THE TOOLBAR OF P&ID TO-DO LIST	
PIPING DESIGN CHECK ALLOWS CHECKING FOR PARAMETER OD	
NEW OPTION TO SET OCCURRENCE PROPERTY HIGHER LEVEL TO NO	57



NEW IN PIPE SPECIFICATION EDITOR	58
New key name Custom Gasket New key name Welding nipple	58 60
NEW IN THE P&ID TO-DO LIST	61
CLEAR FEEDBACK IF PRO FILE IS ALREADY OPENED AND IN USE	61
NEW IN SMAP3D ISOMETRIC	62
Use of the Dynamic file naming of Isogen [©]	62
Using the Solid Edge occurrence properties	65
SUPPORT OF ANGLED BRANCH PLACEHOLDER PARTS	
Take over weld seam attributes from adjacent fittings	67
SUPPORT OF ADDITIONAL OLET BRANCHES ON TEE-PIECES	68
Further standard definitions for Output mappings	69
IMPRINT	70

General improvements



General improvements

Supported CAD versions

Solid Edge

The release version of Smap3D Plant Design 2021 supports:

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



Licensing in version 2021 required

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

For existing license server installations of CAD-Partner Licensing 2020 (when using floating and node-locked licenses), must be updated/upgraded to version 2021 (for details see installation guide chapter > Upgrade).

This upgrades an existing installation of License Server Manager 2020 to version 2021. The updated version number of License Server Manager 2021 is then **6.21.100.5**. The corresponding FlexNet Publisher version is then **11.17.1.0**.

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

Product licenses from version 2020 still valid

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



Installation of the applications tor C:\Program files

All Smap3D Plant Design applications from version 2021 onwards are now installed by default in the path **C:\Program files\CAD-Partner**, which is intended for 64-bit programs.

Up to version 2020, the default installation path was always C:\Program Files (x86), which was intended for 32-bit programs.

Save button only active after changes

After opening the administrator programs **Plant Design Administrator** and/or **Parts Management Administrator**, the **Save** button was always active immediately.

Users could not always clearly see whether they had made unintentional changes to the settings or not.

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





New option, not delete redundant pipe files

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

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

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

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

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

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



New graphical layout of Smap3D online Help

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



Reminder: Automatic log files during installation

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

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



New in Smap3D ScanToCAD

Selection and CAD export from the spherical views

In previous versions of ScanToCAD, point and geometry selection for CAD export was only possible in the **planar view** (unfolded planar 360-degree view) of a scan.

The navigating and working in planar views took some users some getting used to or was even problematic.

To simplify this, the basic handling has now been considerably extended and all existing selection functions can now also be used in the **spherical views**.

This is intended to help experienced 3D CAD users to avoid having to deal with the planar view of Scan.

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





New in Smap3D P&ID

Formula editor supports mathematical operations

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

Symbol defaults O Primary header	Symbol data fields + = 🛱 ▲
O Secondary header	
◯ Line link symbols	Custom 1
O Line link names	Details for selected data field
Symbol data fields	
O Line data fields	

After selecting an existing data field, this button opens the Formula editor dialog.

With the previous functions it was mainly possible to create a string of single data field values, to a single text result and display it visibly in the diagram.

With the new functions you can now also perform **mathematical operations** on values from data fields and then display the calculation results visibly.

For this purpose, the dialog of the formula editor has been extended with the additional areas **Functions** and **Test Data**.

The new mathematical operations can be used in formulas, once the **Edit formula manually** option is enabled.

Only then the corresponding Append button for the **Functions** area becomes active.

Supported are arithmetic operators, boolean operators, comparison operators, equality operators and string operators.

All available functions are available in the list field and can be selected from there.

Formula editor for data field: Cu	ustom 1			X
Formula				
				^
				~
Edit formula manually				
Symbol data	Symbol data fields	Static text	Functions	
Name	\sim Identification letter \sim		ABS()	\sim
Append	Append	Append	LOGO POW(,) ROUNDO SINO SQRTO TANO	^
			Test ormula with test data	Ľ



Depending on the mathematical operation to be performed with a formula, it must contain all needed components to be a valid formula.

With the button **Validate formula** the string defined in the **Formula** field can be checked for its correct syntax.

For a formula to function correctly later, no error message may appear during validation. If you receive error messages, they must be corrected before leaving the formula editor, or the whole formula is deleted again with the **Clear formula** button.

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

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

Formula editor fo	r data field: Custo	m 1							×
Formula									
FORMAT(VAL <mark>(DAT</mark>	FAFIELD("Design tem	nperature (°C)')) - VAL(<mark>DATAFIEL[</mark>)('Op	erating temperature (°C)'))))				< >
🗹 Edit formula mai	nually								
Symbol data		Symbol data fields		Static text		Functi	ons		
Name	~	Operating temperature (°C)	\sim			VAL()			\sim
App	pend	Append		Append			App	pend	
115				Valida	Test ate formula	formula v	vith test da	ata ear formula	
Test data									
Symbol data			Syr	mbol data fields					
Name	Name		Da	itafields	Value				
Туре	Туре		De	sign temperature (°C)	200				
Article	Article		Op	erating temperature	85				
Spec	Spec								
						(Ok	Canc	el

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

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



Since the application possibilities, but also the rules behind these new mathematical operations are very extensive, we have created a separate documentation for them.

This documentation is included in every P&ID 2021 installation and can be accessed via the **Help menu**.



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



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

Support of Microsoft Access MDB files

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

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





New option Component ID

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

Use Component ID instead of NAME as ID for export and re-import

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

t contents	Edit visibi	lity Edit definitio	ns Symbol lit	orary editor	Edit sequence	Settings				
Options Scope O Curren D Entire (21 items)	nt page project		Get data from Project Excel Access						Get data	
First col	lumn locked	l Artide no	Model	Medium 🔻	Operating	Design ter	Operating	Design pre	Volume	^
B2001 (1	133)	22000V	Behälter	ot water 🗸	90	100	6	10	2500	
B2002 (1	134)	22000V	Behälter	Hot water	75	100	10	10	1500	
P2002 (3	39)	96539052		Hot water	95	100	6	10		
W2001 (147)	PK400-35		Cold water	40	50	6	10	1	
P2003 <mark>(</mark> 1	129)	1510		Cold water						
X6 <mark>(140)</mark>)	125014								
V2008 (1	136)	1200600805								
2002 <mark>(1</mark> 5	5 <mark>2)</mark>	50H80								¥
1									>	
`			Export to Excel Cancel Apply							

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

- 2	А	В
1	Name	Article no
2	2004 (146)	
3	P2003 (129)	1510
4	X7 (142)	125014
5	X6 (140)	125014
6	X5 (141)	125014
7	B2002 (134)	22000V
8	B2001 (133)	22000V
0	COLLAD VACAN	501140



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

- If this option is **not activated** (default), only the name value in the **Name** column is used as unique ID for the assignment of data field contents to the diagram symbols during re-import. Here the values for Name (TAG) in the external file **must not be changed**, otherwise the necessary assignment between external file and the symbols in the diagram no longer exists.
- If this option is activated, it is also possible to change the name values/TAG in the Name column (not the value in brackets) in the external file (Excel or Access).
 The component ID (value in brackets) is then used for the re-import and thereby for the unique assignment between external file and diagram symbol.



Use of data symbols for equipment tables

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

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

	Symbol settings	×
JAG: +	General Advanced Reference Titles	Ok
∰ype: _Model		Class
Jemp.: Operating temperature (°C)	Title: Equipmenttable Vessel	Close
Pres.: Operating pressure (bar)		
Vol.: Volume (m3)	Symbol type	
Dia.: Outer diameter (mm)	Data Whole symbol	
Height: Height (mm)	O Reference point	

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

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

- 1. Instead of manually entering the **Name** of the desired component symbol (whose information is to be displayed in this table), this name must be selected using the **Used** function.
- 2. After clicking on the button Σ , the corresponding **Select name** dialog opens.
- 3. From the list of used (existing) names, you should select the desired name of the component and press **OK**.
- 4. Close the **Component Data** dialog with **OK**.



Component data		×
■ a × ⊗ -		Ok
Name:		Cancel
<u></u> <u></u> ype:		Unit
Artide no:		Database
Spec:	+=Σ □□	content
Description:		
General Ref.design. Symbol data fields Reference Acc	📕 Select name 🛛 —	
Symbol type: Data Scale: 1,0 Angle: 0,0 Symbol: Equipment_table_vessel Physical Conduction Name prote	Used names 50H10 0000 1002 1003 1004 2000 2001 2002 2003 2004 2005 2009 4101 B	
	B2001 B2002 Ok Canc	el

The result is an automatically filled out table symbol with the information from the component symbol.

The associativity created in this way, also updates information changes in the component symbol, automatically in the table symbol.





Object lister – individual data fields display

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



-				1		ne (C)			
Object	Lister				KATEGORIE				
Symbols	<u>L</u> ine link	Tex	ts Reference frames P	à	OPERATING PRESSURE (BAR)			
€2 🛧		Ì			BETRIEBSTEMPERATUR DESIGNTEMPERATUR			740	MEDTIN
Name	- 1	Pag	je No.		INSULATION THICKNESS	(MM)		TAG	MEDIOM
IG_1		Visi	ble		SERVICE		L	IG_1	
PL 70	4	Par	ie tyne		TYPE OF INSULATION			PL 70 - 40 - CO - Spec10CA	со
PL 50	5	0	ie type		TAC			PL 50 - 50 - CW - Spec10CA	CW
PL 50	5	Qu	antity						CW
PL 50	5 ~	Nar	ne		DESIGN TEMPERATURE (°C)			PL 50 - 50 - CW - Spec10CA	CW
PL 51	5 ~	lyp	e		ISOLIERUNGSDICKE			PL 51 - 50 - CW - Spec10CA	CW
PL 51	5 ~	Arti	icle	\sim	MEDIUM			PL 51 - 50 - CW - Spec10CA	CW
PL 51	5 ~	Spe	c		MODUL			PL 51 - 50 - CW - Spec10CA	CW
CON 1	0 ~	Lin	e kind		BETRIEBSDRUCK			CON 1 - 01	
CON 1	0	Cha	apter		DESIGNDRUCK			CON 1 - 01	
CON 1	0	Dat	afields >		TYP DER ISOLIERUNG			CON 1 - 01	
PL 60	40		333355		Spec10CA	Straight lines		PL 60 - 40 - HW - Spec10CA	HW
PL 61	65		046666		Spec10CA	Straight lines		PL 61 - 65 - HS - Spec10CA	HS
DI 61	65		046666		Sport0CA	Straight lines			uc



New NON-standard conform aspect delimiter

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

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

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

Their use must be activated in the Standard tab.

Reference designations - Settings	×
Format Standard Import General	Ok
Nonconformity to ISO/IEC/EN81346	Cancel
Do not display sign for full reference designation (>)	
Combined aspects	
Allow line break between aspects	
Do not show trailing prefix on reference designation on ref.frames	
Separated aspects	
Don't display slash between separate aspects	
Allow unauthorised line break between individual aspects	
Allow non-standard prefixes	
Note: All the options above are nonconformities to the ISO/IEC/EN81346 standard. In order to conform to the standard these deviations should be described in the documentation.	

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

Delimiters between aspects	
Standard (/)	\sim
Standard (/)	
:	



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

Component data [PT-6751-A1]	×
🖻 🖷 🗙 🚳 🗝	Ok
Visible Name: A1	All
<u>I</u> ype: ΣΣ	Cancel
<u>A</u> rtide no: 50H80 Σ Σ	Unit
Spec: $+ = \Sigma$	Database
Description: Flowmeter Proline Promag 50H, DN80, PN16	content State:0
General Ref.design. Symbol data fields Reference Conn: Image: Co	·





Compare diameter supports fractional numbers

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

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

Componen	t data [V2009	n				×
		L. L				
<u>∎</u> ⊡ X	M -					Ok
Name:	V2009			E +	Visible 	All
Type:	Valve					Cancel
Type:						Unit
<u>A</u> rticle no:	1200600805	j				Database
Spec:					$+=\Sigma$	Database
Description:	Globe valve S	TOBU 12006 D	N80,PN16, c	ontrol co	one	content
General Ref.	lesign. Symb	ol data fields	Reference	Conns	Accessory	
Name:	Functio	n:	Size	Lin	e Size	
1	Spec 10	CA	1 1/2"	1	1/2"	
2						
						\$
						-
Designched	k Settings:					×
						Ok
Design check						Cancel
Symbols -					· · ·	
Connection	S				^	
Missing	Name					
Used m	Used more than once					
Not Connected						
Overlapping connection points						
Nonconducting line connected						
Compar	e diameter					
Net/Potent	als				~	
Lines/wires					~	
All design error	ors (recomme	nded)	All warning	s		
						_



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

PL1 - 1 1/4 - Oil - ASME flanged

Pesign Check	×
🗖 🏝 🐼 🔅	
0 warnings found !	
0 warnings found !	.::



New option to allow writing to symbol data fields from DB

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

Component data [B1]		×
°ia X ⊗ -		Ok
All=Change all symbols for the component Name: B2000		All
Type: Vessel	Σ Σ/ 🗆	Cancel
Article no: 25006V	Σ Σ/ 🗆 🗆	Unit
Spec:	+ = Σ	Database
Description: Vessel 5000 liter		content
General Ref.design. Symbol data fields R	Reference Conns. Accessory	
Datafield	Value 🗘 ^	
Custom 1		
Custom 2		
Custom 3		
Custom 4		7
Custom 5		
DB[MANUFACTURER]	MaBe GmbH	10 Ca.
Add Edit Remove		

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

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

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



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

- component data [be]		×
🖻 🔓 🗙 🕅 👻		Ok
Name: B2	Visible	All
<u>T</u> ype:	ΣΣ/ □□	Cancel
Article no:	ΣΣ/ 🗆 🗆	Unit
Spec:	+=Σ □□	Database
Description:		content
General Ref.design. Symbol data fields	Reference Conns. Accessory	
Datafield	Value 💡 🔨	
Datafield Custom 1	Value 💡 ^	
Datafield Custom 1 Custom 2	Value ^	
Datafield Custom 1 Custom 2 Custom 3	Value ^	
Datafield Custom 1 Custom 2 Custom 3 Custom 4	Value	
Datafield Custom 1 Custom 2 Custom 3 Custom 4 Custom 5	Value Image:	
Datafield Custom 1 Custom 2 Custom 3 Custom 4 Custom 5 DB[MANUFACTURER]	Value Image: Constraint of the symptotic symptot	

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



New function "Move to other layer"

This new function now offers the user a much faster and easier way to move selected P&ID objects between layers.

Previously, such tasks always had to be performed using copy/cut and paste via the Windows clipboard.

As before, the desired objects must be selected before moving.

Basically, these can be symbols, lines or a combination of both, selected using the Area or Object Groups function.

PL 110 - 100 - CW - Spec10CA X 410° ♣ Move Move to other layer ž Cut Ctrl+X ħ Ctrl+C Copy X Delete Del Invisible

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

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

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



New option when switching layer with ALT key

Since many versions it is already possible to switch between layers (if P&ID objects are placed on different layers) with the shortcut **ALT + click** on object.

Until now, each time you did this, a dialog was displayed in which you had to select the desired action (Yes, No or Cancel) and click.



With the new option **Don't show this dialog again during this session**, you can prevent this dialog from being shown until the next restart of Smap3D P&ID.

Each time the keyboard shortcut is used, the selected action will then be performed automatically.



Line link reference to UNIT pages possible

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



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





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

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

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

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

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

Piping ×
There are lines with the same PIDID in the project. It can cause problems. (PL1) Use CtrI+C to copy message box to clipboard.
ОК

To solve this problem easily in the affected PRO file, we have developed the special function **Check** for unique PidID's.

This can be found in the menu Functions > Special Functions > Check for unique PidID's

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

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

Smap3D P&ID	Ş	×
Highest ID = 3		
		ОК

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



Settings for Select pages are now stored in PDFSET

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

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

PDF Export	×
Select pages Setup Popup information	Ok
Selected pages: 4	
Cover: Cover	Cancel
TOC ToC: Table of Content	
Diagrams: Diagrams	
DIA 1: Main process	
DIA 2: Cooling water	
Lists: Lists	
PAR BoM 1: Main process - BoM	
COM Comp 1: Main process - Comp.	
SVM Legend: Legend	
STM Legend, Legend	
O Current page O All pages	
O Actual chapter	
Last used page active	Update All Lists
Quick setup	
with pages setting.pdfset 🗸 🖓 💾	
· · · · · · · · · · · · · · · · · · ·	



Name list in Parts lists supports aspects

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

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

	Patafield -			
Name list	Data type Name list			Ok
-	🔿 System Data	User name	\sim	Cancel
Name list	O Project Data	Projekt Nr.	\sim	
	O Page Data	Zusätzlicher Titel	~	
Name list	O Symbol Data Field	Identification letter	~	
	◯ Line Data Field	TAG	\sim	
Name liet	 Table of Contents 	Zusätzlicher Titel	\sim	
Inditie list	Parts/Components List	Name list	~	
Name list	O Connections list	Signal	~	
Name list		Function aspect Location aspect Product aspect		
Name list				
Name list	Activate next Wrap	text		
Name list	Pre-text:	Width	18	
Name list				



Text translator supports Pretexts

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

Options
Dictionaries
Texts to translate > FreeText > Symbol Name Type Article Y Function Symbol datafield Fieldvalue Y Ref. text
 Symbol definition (includes page headers) Page Project
Innore leading/trailing numbers and special characters in texts
Enabled symbolnames (if enabled, export/import-features will be
Close



Description in all list settings

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

New language Italian

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

New language Chinese Traditional/Taiwanese

With P&ID 2021, the software can also be operated with Chinese Traditional/Taiwanese language for the user interface.

In the **Select language** setting, the entry **TW** stands for **Taiwanese** equivalent to **Chinese** traditional.



New in Coordinate System Wizard

New input field for outside diameter

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

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

Coordinate System Wizard	- ×
🍫 Appearance	e 👻 🥐 Help 🎁 About
Instructions Set the desired CSInfo parameters of the edited [O coordinate system. Use the OK button to accept d	UTPUT_POINT_1] hanges.
Name and index:	
OUTPUT_POINT_XX *	1 🔺
✓ Create CSInfo CSInfo	
03: Butt Weld	•
End condition (ISO_CONNECTION):	
Not Specified	•
Sequence number:	1
Input/Output:	Undefined 🝷
Gender	Undefined -
Size:	DN80
Wall thickness:	3,20 mm
Outer diameter:	88,90 mm
Custom parameters: Name	Value
Remove row	Add row
☑ Edit CSInfo only	
Rotation X [90°] Y [90°]	2] Z [90°]
ок	Close



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

Parameter **OD** = Outer Diameter

Parameter WT = Wall Thickness

OUTPUT_POINT_1_CSInfo	Text 🗸	IO=0 GR=0 SN=1 ET=5 SZ=DN80 <mark>OD=88,90 mm</mark> <mark>WT=3,20 mm</mark>
		1

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

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



New end treatment Welding nipple

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

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

-CSInfo	
End treatment:	
Not Specified	•
Not Specified	
01: Threaded	
02: Socket Weld	
03: Butt Weld	
04: Flanged	
05: Welded Neck	
06: Blind	
07: Slip On	
U8: Lap Joint	
19: Socket	
10: Butt weld Branch	
11: Spigot 12: Dush Eit Plastic Socket	
13: Solvent Welding Plastic Socket	
14: Plastic Spinot	
15: Cone Socket	
16: Cone Spigot	
17: Hose	
18: Tri Clamp Coupling	
19: Grooved Coupling	
1001: Flanged Without Gasket	
1002: Counter Flanged	
1003: Counter Flanged Without Gasket	
1004: Shape Gasket	
1005: Custom Gasket Flanged	
1006: Custom Gasket Counter Flanged	
1007: Welding Nipple	



New end treatment Custom Gasket

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

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

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

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

1001: Flanged Without Gasket 1002: Counter Flanged 1003: Counter Flanged Without Gasket 1004: Shape Gasket 1005: Custom Gasket Flanged 1006: Custom Gasket Counter Flanged 1007: Welding Nipple



For each of these two end treatments, the definition of the <u>key name from the pipe specification</u> to be used by the QuickPlace method must also be made.

This is done in the **Custom parameters** area with the entry **C_GASKET** in the **Name** column. The entry **C_GASKET** must be written exactly as shown below!

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

🤞 G	oordinate System Wizard	- >
	🏹 Appearan	ce 👻 🥐 Help 🎁 Abo
Instructions		
Set the desired CSInfo coordinate system. Us	parameters of the edited [e the OK button to accept (OUTPUT_POINT_1] changes.
Name and index:		
OUTPUT_POINT_xx		1
✓ Create CSInfo		
-CSInfo		
End treatment:		
1005: Custom Gasket I	Flanged	-
End condition (ISO_CO	DNNECTION):	
Not Specified		•
Sequence number:		1 📥
Input/Output:		Undefined 🔹
Gender:		Undefined 👻
Size:		
Wall thickness:		0,00 mm
Outer diameter:		0,00 mm
Custom parameters:		
Name		Value
C_GASKET	Spezial1	•
L		
	Remove row:	Addrow
	Kenioverow	Addrow
J Edit CSInfo only		
Rotation		2 [000]
	X [90°] Y [90	[-] Z [a0-]

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

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

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

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



New in Smap3D Piping

New QuickPlace method Custom Gasket

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

Until now, only **one gasket** per diameter could be defined for piping in a pipe specification for simultaneous automatic use (favorite).

In practice, however, there are situations where special components (valves, instruments, etc.) within a pipe specification and diameter require a different gasket for assembly and correct functioning than that required for other fittings of the same pipe spec and same diameter.

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

This new method is additional and should make it possible to automate extended customer requirements relating to gaskets in pipeline planning.

However, since we do not know all the practical variations on gaskets for all customers around the globe and for all available fitting standards and industries, the concept and implementation of this QuickPlace Method **Custom Gasket** has been implemented as **flexible** and **adaptable** as possible.

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

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

or,

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

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

- In a pipeline to be generated/processed, 3D components must be present, which contain the <u>definitions for the requesting of a custom gasket</u> at their connection points (CSInfo property).
- In the pipe specification and diameter used, gasket components must be defined under the <u>specific key names specified</u> in the respective <u>CSInfo property of a requesting component</u>.



Example

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

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

Special_1	Custom gasket	Flange	4,50 82	J	
Gasket_3	Custom gasket	 Flange 	4,50 83	v	

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

	tandards - Metric design\Gasket\Gasket forflange\Spiral wound gasket\G ASME_B16.20_2017_mm_A_Gasket_4_2500_v16) [a] [
Gasket_3 🔹 80 💽 - 🔢 Lense /2x116 DIN 2696 DIN Gasket Gasket Gasket Cense /DIN 2696-19/2 Lense /2x116 DIN 2696	Gasket\Gasket forflange\Stainless steel gasket\Lense\DIN 2696:1972 DIN_2696_1972_Lense_PN250_72x116_v14.00,p	-	

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	🥐 – 🗖 ×	Pipe specification row detail	🥐 – 🗖 ×
Basic Part Virtual characteristics Additional items		Basic Part Virtual characteristics Additional items	
Name	Value	Name	Value
ADDITIONAL_ITEM_1_BOLT_ITEM_CODE	123	ADDITIONAL_ITEM_1_BOLT_DIA	15
ADDITIONAL_ITEM_1_BOLT_QUANTITY	2	ADDITIONAL_ITEM_1_BOLT_QUANTITY	4
ADDITIONAL_ITEM_1_BOLT_DIA	12	ADDITIONAL_ITEM_1_BOLT_ITEM_CODE	987
ADDITIONAL_ITEM_1_BOLT_ITEM_DESCRIPTION	Special1 Bolt	ADDITIONAL_ITEM_1_BOLT_ITEM_DESCRIPTION	Gasket_3 - Bolts
щ,		F.	
ок	Cancel	ок	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 and are processed and displayed accordingly during the generation of drawings (depending on the used style).

			AL LIST - FAE	
F4 G6/7	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	
\times	7	15	987	Gasket_3 - Bolts
	8	12	123	Special1 Bolt
	9	80	Butterfly Valve DN 8	



New quick place method Welding nipple

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

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

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

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

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





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





New function Define Cut piece allowance

Basically, **CUT_PIECE_ALLOWANCE** is a parameter for Isogen[©] to define a value for an additional length to a pipe end of a pipeline.

This value is then displayed accordingly on a production isometric.

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



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

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

The new Smap3D Piping function **Define Cut piece allowance** has now been developed to make this definition for a specific pipe end easier and more user-friendly.

The end selection on a 3D pipe is graphically displayed, and the writing of the correct parameter with its length value to the selected pipe end and its end number is automated.



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

⊡… 🏂 path 002 Asm1.par:1	Hide pipeline p <u>a</u> th(s)
	Update all custom occurrence properties
🔿 DIN_EN_1514_1_1997 🗠	Place part
🔇 Flange_DIN_EN_1092 🙀	Repla <u>c</u> e part
P DIN_EN_10253_2_200	Place one concentric reducer
🛱 🥓 Segment1	Place concentric reducer twice
Segment2	Place component pattern
Insulations	Split pipe
	Place split point for Isometric
	Define 'Cut piece allowance'
Ľ	Show part(s)
	Hide part(s)

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.

Distance	: (?	> - >	×		
Write 'Cut piece allowar (higlighted) pipe end p	ce' value for the int	eselected			
		150 mm	n		
Swap	endpoint				
Write	Can	cel		6	



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

The stored value for this characteristic can be edited either by calling the **Define Cut Piece Allowance** function again for the same pipe,

or by using the **VC Writer** if the appropriate characteristic names CUT_PIECE_ALLOWANCE1 and CUT_PIECE_ALLOWANCE2 are defined in its configuration file.

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





Working with angled branch dummy parts

We received support requests from more and more customers asking us to recommend a procedure for the best possible realization of angled pipe branches (pipe on pipe without branch fittings) with Smap3D Piping.

The special challenge here was that every branching 3D pipe (regardless of branch angle and diameter) should be shortened on the circumference of the main pipe in order to obtain realistic cut lengths for prefabrication.



Since in such pipe-on-pipe branch situations no regular fittings can be used for dimensional control, the only way to give the piping software the necessary information for correct pipe length generation is to use placeholders (dummies).

For most customer requests, the branch situations should also be very variable with respect to the branch angle, so that it is not possible to react only to a few fixed standard angles.

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

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

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

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





The most important contents for the Smap3D Piping software are the contained coordinate systems and their positions.

For the main pipe there is a pair of coordinate systems with information about the correct handling of a branch pipe (end treatment is ET=10), and for the branch pipe there is a pair of coordinate systems with information about where the branching pipe must end.

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

Smap3D Piping does **NOT offer any automation** to automatically add such a placeholder/dummy part to an angled branch situation.

A placeholder/dummy part adapted to a geometric situation must be placed **manually by the user on the pipeline paths using native CAD constrains**.

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





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

Ф I	Part Finder	- 🗆 ×
Home		🛛 🏹 Options 👻 🅐 Help 🌓 About
Part Editor Manage DB Refresh DB CAD Refresh	v Add Delete bookmark bookmarks	operties in Properties in Other lists detail Other Other Search Layout
Main	Bookmarks	Options • •
Part selection [C:\Smap3D_PlantDesign\Database\SWD	efaultDB.mdb]	Selected part
Visual Categ. Specif. Cat. Bk Queries	🛉 💻 🥒 🕑 Priority 10 🔄 🔝	Preview
⊕ DIN ⊖ Fabrication standard ↓ Angled branch ⊕ Branch ⊕ Branch ⊕ Brow universal └ Slope angle universal ⊕ US Standards - Inch design ⊕ Victaulic@ - inch design ⊕ Victaulic@ - metric design	Characteristic Value Unit	Angled branch CS Dummy Properties Name Value Type CS Dummy Angled 55 5
C:\Smap3D_PlantDesign\WorkingFolder\Angled_Branch_CS	Dummy_template_SWX#2.SLDPRT	Outside diameter 1 114,3 mm Outside diameter 2 42,3 mm

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

Variable	s 🥜 – 🗙	
Characteristic	Value	Unit
Outside diameter 2	42,3	mm
Outside diameter 1	114,3	mm
Angle	66	0
		ок

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



Replace part function supports component packages

With the introduction of the automation for tube fittings in version 2020, there are currently three key names for which several components (component packages) can be defined in a pipe specification.

These are automatically placed on a pipeline path by the corresponding QuickPlace methods when a request is made.

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

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



The **Replace Part** function could not detect such component packages until now and could not replace them as packages.

This limitation has now been removed in version 2021.



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



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



New option to automatically split a sketch line

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



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



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

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



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

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

	Options		😗 — 🗙
✓ Use default settings from Plant Design ad	ministrator		
Generate		-Placing of parts	
Use subfolder for the settings below		Create an axial relationship on flanges	and gaskets
Use assembly name as subfolder nam	e		
OUse custom subfolder name:		- Appearance	
		Toolbar position O Left	Right
		Different icons for automatically and r	manually placed fittings
Folder for generated pipes:		Different icons for invisible parts	
Assembly path		Highlight occurrences in CAD	
Custom folder:			hroute
		Show pipe preview	
2: \E Wetzlaufwerk PID \2021_beta \M	antis 37288 (Mantis_3728	Show sketch functions to olbar	
Copy parts to working folder		User settings	
		✓ Automatic synchronization	
Assembly path Custom folder		Do not display minimum flatlength ale	ert
Ocustoni folda:		Create mates between shape pipes	
		Group Piping components in CAD ass	embly tree
Copy parts indexed		Automatic split lines with a branch line	n
		V Automatic spire intes with a branch inte	
		Set the occurrence properties status of pi	neline naths to:
		Yes No	penne paulo con
Default pipe values		Document name formula:	
4in. lengthtolerance:	10 mm		Change
-		Detection of Piping objects:	
Default preview		based on relations (accelerates Piping with many imported parts)	when working on large assemblie
None 2D	ЗD	based on position of coordinate system	s
			OK Cancel
			UK Calicel

If this option is active, it works as follows:

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

so, when closing / exiting the 3D sketch environment, all lines in the active 3D sketch feature are checked to ensure that **they are only connected at line end points**.

If this is not the case and there are line elements also connected somewhere along a continuous line (relation endpoint to line), the baseline is automatically split at this point (as with **split line**), and then only endpoint to endpoint connections are made between the individual physical lines.

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

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



Material from template is used for pipeline path

For the generation of pipeline path files, a template file can be set in the Plant Design Administrator.

Previously, this template was only used as a supplier of **specific file properties** for the generated pipeline path files.

Starting with version 2021 it is now also possible to **define a material** in the template file (using the Solid Edge material library).

When generating new pipeline path files, not only the prepared file properties are then transferred to the individual pipeline path, but also the existing material definitions (density, face coloring, etc.).

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



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

Since the introduction of the Basic Engineering editing mode (version 2019), more and more customers have been using this mode in the design and pre-planning of their piping systems, thus taking advantage of the corresponding performance benefits.

Only at a later point in time (days or weeks) are these pipelines then switched over to detailed pipeline generation.

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

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



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



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

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

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

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

Display P&ID To-Do list toolbar in CAD

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



Piping Design check allows checking for parameter OD

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

CS	ilnfo check setti	ngs	? -	- 🗆 🗙
Check type:				
Check con	nections			
◯ Check fitti	ngs against valu	es of first fittir	g	
Check for:				
Fittings or	ıly			
 Fittings & 	Pipes			
Compare	all existing size a	nd custom fiel	ds	
Size (SZ)				
Wall thic	kness (WT)			
 Outside 	diameter (OD)			
Custom fields				
Use	Name			
		ОК	0	lancel



New option to set occurrence property Higher Level to No

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

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

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

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

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

Higher Level	
No	



New in Pipe specification editor

New key name Custom Gasket

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

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

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

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

	Key name editor	🥐 – 🗙
Key name:		
Custom gasket	EN 👻	
Add by edit		
Order:		
	4,50 🜲	
Reduction		
QP method:		
Custom gasket	 Image: Constraint of the second second	
Non-unique index		
80		
Conditions:		
Flange	+/-	
Is pipe		
	Basic 🝷 OK	Cancel



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 <u>QuickPlace method example</u> described, the new key name definition should look like this:

	Key name editor	🥐 – 🗙
Key name:		
Special_1	EN 🔻	
🗸 Add by edit		
Order:		
	4,50 🜲	
Reduction		
QP method:		
Custom gasket	× 🧖	
Non-unique indec		
82		
Conditions:		
Flange	+/-	
Is pipe		
	Basic 🔹 OK	Cancel

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

If Smap3D Piping is to use several different key names at the same time for the same automation, the **non-unique index** must be **different** for each key name.

If this were not the case (different key names would have the same non-unique index), the pipe specification editor would ask for a favorite when saving. This would negate the advantage of this new QuickPlace method.



New key name Welding nipple

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

With version 2021, the new key name **Welding nipple** with corresponding QuickPlace method **Welding nipple** is available in the pipe specification editor.

This new key name is automatically added to existing pipe specification configuration
 (Common settings - PipeSpecificationCommon.pssx) during an upgrade (from PD 2020 to PD 2021
 - Installation option Existing data).

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

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

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





New in the P&ID To-Do list

Clear feedback if PRO file is already opened and in use

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

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

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





New in Smap3D Isometric

Use of the Dynamic file naming of Isogen[©]

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

1	Use file name generator		
2	Drawing style in filename None Prefix Subdirectory	Assembly in filename None Prefix	Pipeline None Suffix

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

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

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

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

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



When upgrading from previous versions, existing styles **must** also **be changed** to this new definition method before generating drawings with Isometric 2021.

Detailed instructions can be found in the 2021 installation guide included in the software package in the chapters Upgrade and FAQ.

				Options	
PCF Templates	Advanced	Feature overrides	Output mappings	Custom symbols	
Drawing filename Use filename ger Output path Use Plot File Path	nerator				
C:\Smap3D_Plant	tDesign\Isomet write to Isog	tric\ProjectList.xml			🛃
Smap3D					~
Drawing style:					
Final-Basic	ОК	\$STYLE\$\Draw	ings\Final-Basic-\$PIPEL	INE-REFERENCE\$-???	
Final-Cut-List	ок	\$STYLE\$\Draw	ings\Final-Cut-List-\$PIPE	ELINE-REFERENCE\$-???	
Final-Weld-Box	ОК	\$STYLE\$\Draw	ings\Final-Weld-Box-\$PI	PELINE-REFERENCE\$-??	??
Spool	ОК	\$STYLE\$\Draw	ings\Spool-\$PIPELINE-I	REFERENCE\$-???	

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

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

Isometric output file name editor			-	×
Paths ¢STVI E¢\Drawings\				
STILLS (Drawings) Fina	al-Basic-SPIPELINE-REFERENCES-???			
Macro	Description			^
?	Sheet number (e.g. ??? = 001)			
\$PIPELINE-REFERENCE\$	Name of the pipeline			
\$PIPING-SPEC\$	Primary Piping specification			
\$SPOOL-NAME\$	The spool name			
\$INSULATION-SPEC\$	Primary insulation specification			
\$MISC-SPEC1\$	Miscellaneous specification 1			
\$MISC-SPEC2\$	Miscellaneous specification 2			
\$MISC-SPEC3\$	Miscellaneous specification 3			
\$MISC-SPEC4\$	Miscellaneous specification 4			
\$MISC-SPEC5\$	Miscellaneous specification 5			
\$REVISION\$	Revision identifier			
\$AREA\$	Plant area			
\$BATCH\$	Plant batch (same as area)			
\$DATE-DMY\$	Date			
\$ATTRIBUTE0\$ to \$ATTRIBUTE199\$	Any of the 200 user defined attributes			4
	0	к	Cano	el



This **Isometric output file name editor** contains all available macros (variables) with which you can define a syntax for future drawing file names based on the new Isogen[©] Dynamic File Naming method.

Each macro listed here can be combined individually or together with others to a desired file name syntax (in the **Path:** field). It is also possible to use static texts (strings) or only single letters or characters.

If the status of each available style in Smap3D Isometric is set to **OK**, then the file name definitions performed in Isometric will also be correctly written to the **Plot File Path** setting of the corresponding Isogen styles (visible in I-Configure).

These definitions, made in Smap3D Isometric, are then used for the file names of the drawings that will be generated with this style in the future.

I-Configure for Plant Design		- 🗆 X
Datei Bearbeiten Werkzeuge H	ilfe	
	I 🗙 🛍 🖪 🥑	
🖃 🙀 C:\Smap3D_PlantDesign\Isome	ric Property	Value
🖃 🖓 Smap3D	Name	Final-Basic
Final-Basic	Path	C:\Smap3D_PlantDesign\Isometric\Smap3D\Final-Basic
🛅 Spool	ExportBehaviour	Always Export
🗈 Final-Cut-List	StyleState	Export Pending
🔚 Final-Weld-Box	StyleType	User
	ProjectName	Smap3D
	CanWrite	True
	CanExport	True
	Output Format	Smart DWG (DWG)
	Drawing Size	A2
	Drawing Frame	\$STYLE\$\A2-BASIC.dwg
	Plot File Path	\$STYLE\$\Drawings\Final-Basic-\$PIPELINE-REFERENCE\$-???
	Isometric Type	Combined
	North Direction	Top Right
	Units	Metric / MM Bores
	Material List	True
	Weld List	False
1	Cut List	False
	Reports	MATERIAL-CONTROL
	Edit	
C:\Smap3D_PlantE Smap3D	Final-Basic Empty	

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

ECP. ISOGEN CONFIGURATION	Intergraph 🍇 ? 🗆 🗙
Final-Basic + Drawing Manager + Reports	Search P
 Reports 	Material control X
Material control	Depart Leveling SETVIES Departs SUBELINE DEEERENCES Material by
Bending Report 4	Report tocation 3111123 (Reports 3111121112-RCI CREmeteration)
Bulk Material List	Include Title Block
Centreline / Insulation Length	Z Include Column Headers
Centreline length	Mindude Column readers
Cut list summary	PIPELINE ITEM SIZE
Drawing cross-reference	NAME ISS. CODE (INS.) QTY
Drawing report	
Erection information	
Fabrication information	Report Layout
Material / sheet identification	Pipeline Reference 🗙 🗈 Revision 🗙 🗈 Item Code 🗙 🗈 N.S. 🗙 🖻 QTY 🗙
Material control	Width Start Width Start Width Start Width Start

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



Using the Solid Edge occurrence properties

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

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

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

	Cccurrence Properties			- 0
	Translate/rotate about: Model Space V	2 🔏 🖻 🛍	12	
	Placement Name	BOM-ID	Maintenance	Notes
	🖃 🥵 Strang_02.asm			
	🕡 path_001_Strang_02.par:1			
	DIN_EN_10253_2_2008_A_90E_3D_DN50_60.3x2.9.p			
	- 🗇 DIN_EN_1514_1_1997_Gasket_IBC_DN50_PN16_T1			
	- 🗇 Flange_DIN_EN_1092_1_2008_11B1_DN50_PN16.pa			
	- 🕡 pipe_001_Strang_02.par:1			
	- 🕡 pipe_002_Strang_02.par:1			
	🕡 path_002_Strang_02.par:1			
	Flange_DIN_EN_1092_1_2008_11B1_DN50_PN16.pa			
	DIN_EN_10253_2_2008_A_TS_DN50_60.3x2.9.par:1	Tee 1	Weekly	Information entered by the user
	(I) nine 002 Streen 02 new1			

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

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

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





Support of angled branch placeholder parts

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



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

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

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





Take over weld seam attributes from adjacent fittings

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

This previous standard behavior led to limitations in welding seams that were required directly between fittings (when there is no pipe in between).

In these cases, the parameter values (outside diameter and wall thickness) for the weld could not be read and displayed.

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

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



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

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



Support of additional olet branches on tee-pieces

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



From the point of view of Smap3D Isometric and Isogen, the difference so far has been whether such a branch is placed on a pipe or a tee-piece fitting.

Olet branches on pipes have long been supported in Isometric and Isogen and output in drawings.

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

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





Further standard definitions for Output mappings

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

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

			Options			? -		×
PCF	Templates	Advanced	Feature overrides	Output mappings	Custom sy	mbols		Ŧ
F	Priority	C	AD feature	PC	F feature		×	
	10	VC_CUT_PIE	CE_ALLOWANCE1	CUT_PIECE_ALLOW/	NCE1		-	
	10	VC_CUT_PIE	CE_ALLOWANCE2	CUT_PIECE_ALLOW/	NCE2		•	
	10	DETAIL_SKE	TCH_IDENTIFIER	DETAIL_SKETCH_IDE	ENTIFIER		•	
	10	DIAL_FACE_[DIRECTION	DIAL_FACE_DIRECTI	ON		•	

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







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