SolidSteel parametric for SOLIDWORKS

New version 4.1 - now available!

With the last update to version 4.0 at the end of March, the functionality of SolidSteel parametric for SOLIDWORKS was significantly extended (read here).
The new update to version 4.1 focuses this time not on new functions, but on technical improvements and revisions of the dialogs (PMP) to make the input of necessary parameters more intuitive.

The improvements in SolidSteel parametric 4.1 concern, among others, various connections, profiles, sections, bolted joints and the drawing derivation. As a new feature, the evaluation tool for bolted connections has been implemented.

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SOLIDWORKS Add-Ins for structural steel design,
DSTV NC Export, Aluminum Profile Design

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Clip angle connections

Previously, angle connections could only be placed or deleted. With the new edit function, the previous deletion of an already placed connection is no longer necessary and the parameters can simply be changed as desired.

With the new „ Switch sides" function, you have a direct influence to which profile the long or the short arm of the angle is placed.

The new option „without holes", ensures that you can now use clip angles without the otherwise automatically generated holes.

As is already the case with other functions in SolidSteel parametric, the last used inputs are now also taken over for the angle connections, as long as the dialog has not been closed by „pinning". This increases the comfort and the working speed.

The input dialog (PMP) has been revised so that the input of parameters is more intuitive.

Sheet angular connection

The new option „without holes", ensures that you can now use sheet angular connections without the otherwise automatically generated holes.

As is already the case with other functions in SolidSteel parametric, the last used inputs are now also taken over for the sheet angular connections, as long as the dialog has not been closed by „pinning". This increases the comfort and the working speed.

The input dialog (PMP) has been revised so that the input of parameters is more intuitive.

PMP clip angle connections
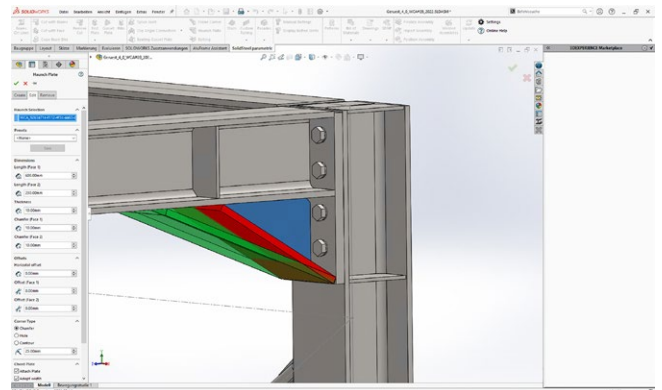

PMP sheet angular connections

Haunches

With the new Presets feature, you can now save frequently used haunches as templates and reuse them at any time.

Previously, haunches could only be placed or deleted. With the new edit function, the previous deletion of an already placed haunch is no longer necessary and the parameters can simply be changed as desired.

As is already the case with other functions in SolidSteel parametric, the last used entries are now also taken over for haunches as long as the dialog has not been closed by „pinning". This increases the comfort and the working speed.

The input dialog (PMP) has been revised so that the input of parameters is more intuitive.
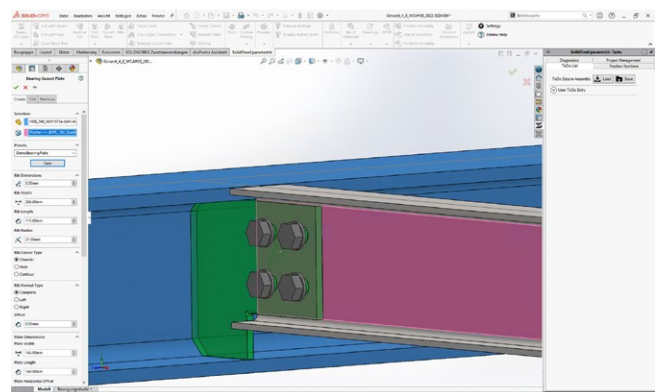

Bearing Gusset Plates

With the new Presets feature, you can now save frequently used bearing gusset plates as templates and reuse them at any time.

Previously, bearing gusset plates could only be placed or deleted. With the new edit function, the previous deletion of an already placed bearing gusset plate is no longer necessary and the parameters can simply be changed as desired.

As is already the case with other functions in SolidSteel parametric, the last used entries are now also taken over for bearing gusset plates as long as the dialog has not been closed by „pinning". This increases the comfort and the working speed.

The input dialog (PMP) has been revised so that the input of parameters is more intuitive.

# New features <small>in depth</small>

Splice Joints

With the new Presets feature, you can now save frequently used splice joints as templates and reuse them at any time.

Previously, splice joints could only be placed or deleted. With the new edit function, the previous deletion of an already placed splice joint is no longer necessary and the parameters can simply be changed as desired.

As is already the case with other functions in SolidSteel parametric, the last used entries are now also taken over for splice joints as long as the dialog has not been closed by „pinning". This increases the comfort and the working speed.

The input dialog (PMP) has been revised so that the input of parameters is more intuitive.

Bug fixing for the preview of bolted joints.
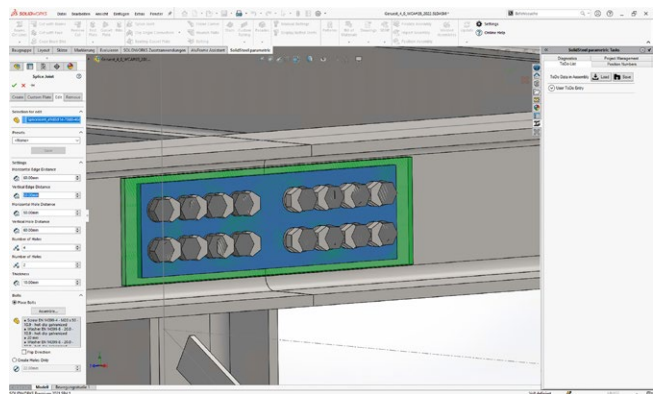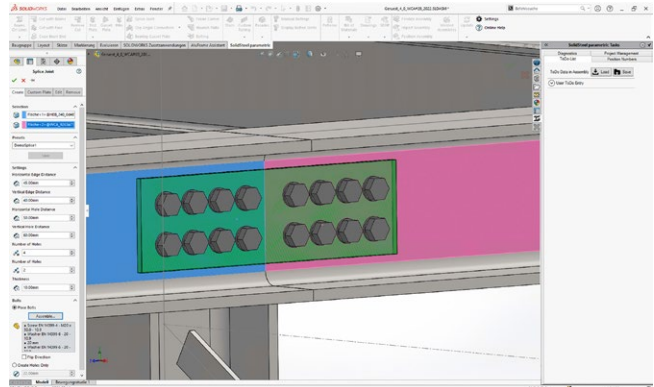
End plates

End plates are often ground to size for constructions with increased precision requirements. Therefore it is necessary to provide the end plates with an overlap to the beam in the design. In the new version it is now possible to enter a negative offset for the gap dimension.
If a negative offset exists, the information about it is written into the user-defined properties of the plates.

The presets management has been completely revised. Presets for end plates now include all settings in the dialog. The new presets management gives you full control and allows you to directly delete, import, export and also exchange presets over the network. This eliminates the previous exchange of presets via Excel tables.

The deletion of faulty bolted connections has been improved.

The input dialog (PMP) has been revised so that the input of parameters is more intuitive.

# SolidSteel parametric
for SOLIDWORKS
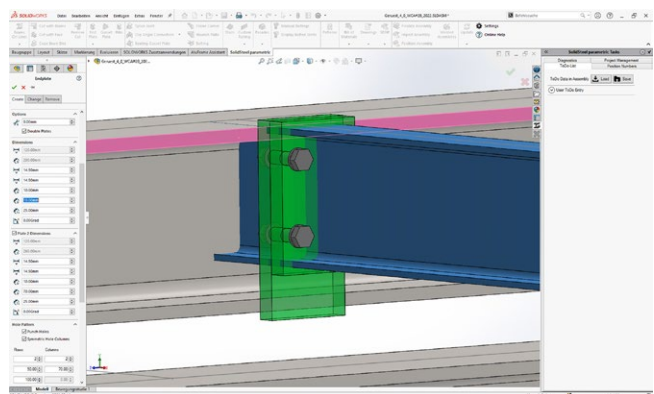
Ribs

With the new Presets feature, you can now save frequently used ribs as templates and reuse them at any time.

Previously, ribs could only be placed or deleted. With the new edit function, the previous deletion of an already placed rib is no longer necessary and the parameters can simply be changed as desired.

Ribs can now also be aligned to sketch lines, instead of previously only to body edges.

The input dialog (PMP) has been revised so that the input of parameters is more intuitive.
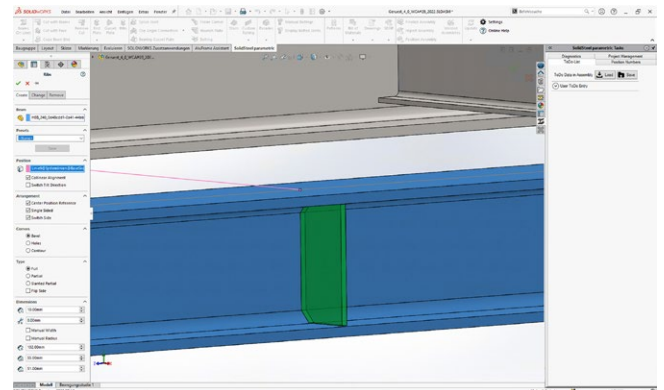
Rib aligned to sketchline
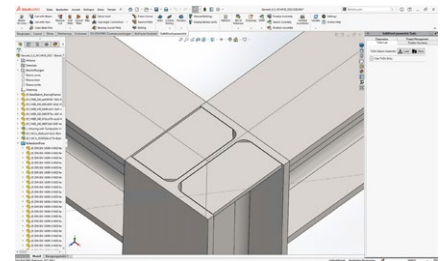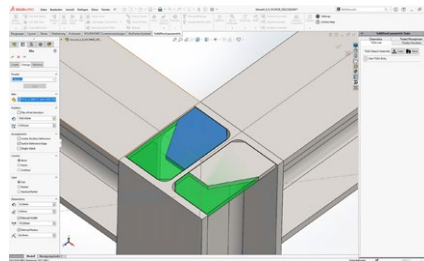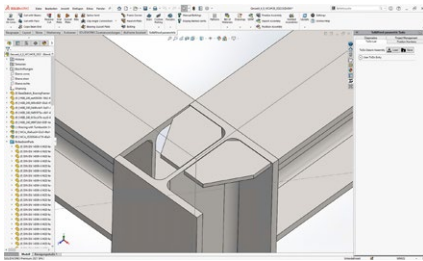
Profiles

Profile database:

• Improved messages about incorrect template tables
• Empty lines in text files of profile templates no longer trigger errorss
• Stability update when using unknown profiles
• South African profiles have been added to the database

Assembly update:

• Profiles without database entry no longer trigger an error message, but are skipped and appear on the ToDo list
• Profiles with missing sketch line also end up on the ToDo list

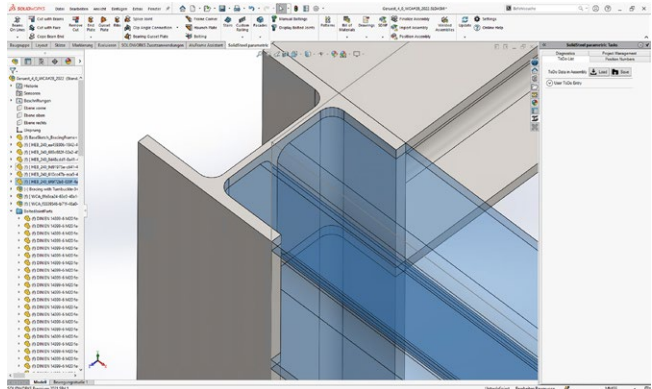Bugfix with face cuts on curved beams.

New features in depth

Cuts

The contour cuts function has been revised and now runs more stable.

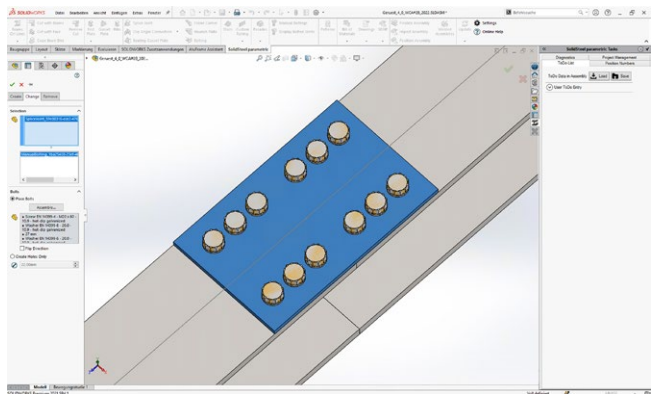The preview error for offset round profiles has been fixed.

Error with face cuts on curved profiles has been fixed.

Bolting

Compatibility in combinations of connections with manual bolting has been improved.

Stability and speed have been improved.

Drawing export

An extended dialog for naming .slddrw files to be exported has been introduced for exporting drawings.

The assignment of the designations is done by „drag & drop" and thus offers the possibility to define the naming scheme for drawings in a uniform way.

Naming Scheme for exported .slddrw files

Position Number	Component Name	Plain Text	Assembly Name	Order Number	Custom Property

Order Number	Plain Text	Position Number	Plain Text	Component Name
	-		-	

Evaluation tool

The new function „Scan bolted joints" can be found in the Diagnostics tab of the SolidSteel parametric Task pane.

It scans the assembly for incorrect bolting data (metadata or attributes). This can take several minutes depending on the size of the assembly.
In concrete terms, this means that the data of all bolted joints is compared in the background with the data of the connections and the existing components.
If faulty data is found, this is displayed after the scan is complete and a dialog asks whether the faulty data should be deleted.
By confirming with „yes" this meta data will be removed, if in the dialog instead „no" is chosen, the detected data will be written into the ToDo Manager (the ToDo Manager is also located in the SolidSteel Task pane).
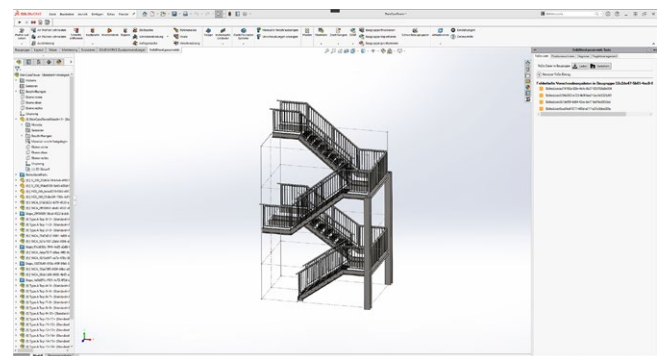
Optionally the permanent update for bolting data can be activated in the SolidSteel settings. However, since this can take time as described above, this item is disabled by default.
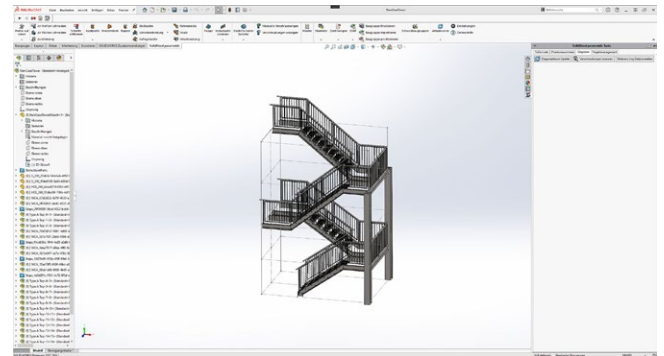
Function „Scan bolted joints"
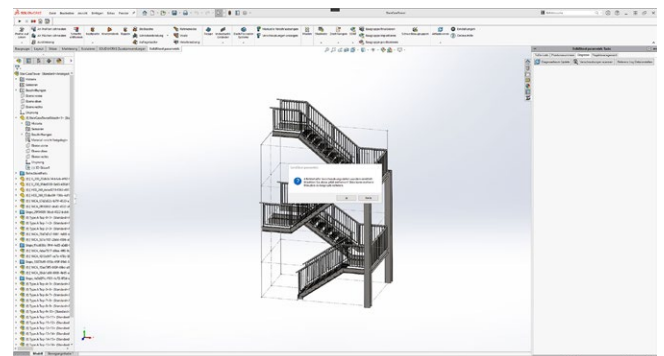
Dialog after Scan

Listing in ToDo Manager

New features in depth

More Updates

DSTV Assistant v2.1

The DSTV Assistant for SOLIDWORKS is now also available for you in the new version 2.1.

More information:
klietsch.com/news/swx/dstv_210

AluFrame Assistant v2.0

The AluFrame Assistant for SOLIDWORKS is now also available in the new version 2.0.

More information:
klietsch.com/news/swx/afa200

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Apps for SOLIDWORKS by Klietsch on LinkedIn:

Apps for SOLIDWORKS by Klietsch on YouTube:

How to get our software

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Currently there are resellers in the countrys marked darker in the world map above, which are:
Argentina, Australia, Austria, Bosnia & Herzegovina, Canada, China, Colombia, Croatia, France, Germany, Greece, Hong Kong, Hungary, Italy, Poland, Portugal, Slovakia, Slovenia, Switzerland, Turkey, USA, Vietnam...

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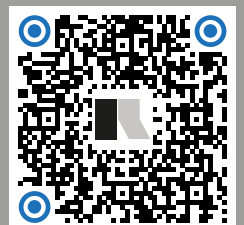
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