Apps for SOLIDWORKS

by KLIETSCH INGENIEURGEMEINSCHAFT

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

www.klietsch.com

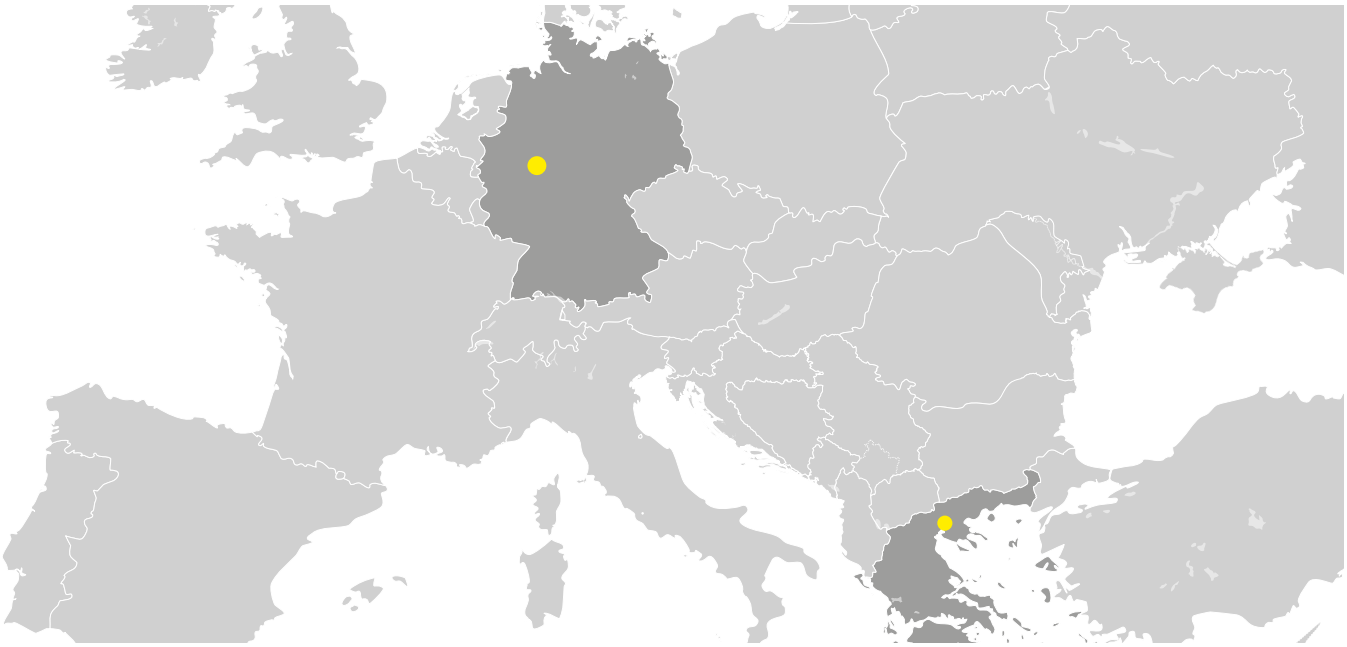
Our company

1973 marks the start of the Ingenieurgemeinschaft Klietsch GmbH as an engineering office for civil, steel and plant engineering.

After the successful sales start of static analysis software solutions, first made only for internal use, the Ingenieurgemeinschaft Klietsch started in the ‘80s with the development of cad solutions for steel, plant and mechanical engineering.

Today the Ingenieurgemeinschaft Klietsch is present on the market as a full service provider for 3D CAD, PLM and PDM solutions from the PTC Creo product family including own solutions and assistants for PTC Creo Elements/ Direct Modeling, SOLIDWORKS and Siemens NX, software for technical calculations in steel construction, and the development of browser-based configurators.

50 years of presence on the market, 3 locations, international sales, more than 1500 customers, thousands of sold licenses… Ingenieurgemeinschaft Klietsch, a strong and reliable partner!



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Be part of the *Gemeinschaft**

**Gemeinschaft* [gəˈmaɪnʃaft] is the German phrase for *community.*
The Klietsch Ingenieur*gemeinschaft* is developing *software for engineers made by engineers* for some decades.
The key to our successful Apps was always the interaction between our developers and the customers.
So the *Gemeinschaft* isn‘t only us – it is a community of users and developers.

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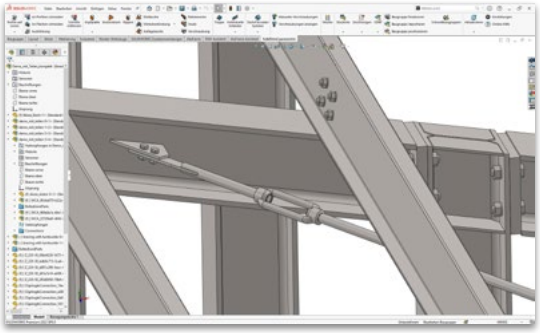

SolidSteel parametric
for SOLIDWORKS

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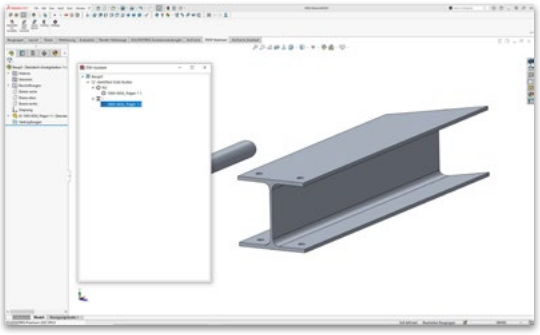

DSTV Assistant
for SOLIDWORKS

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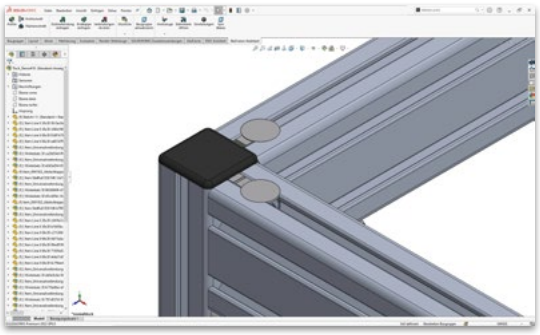

AluFrame Assistant
for SOLIDWORKS

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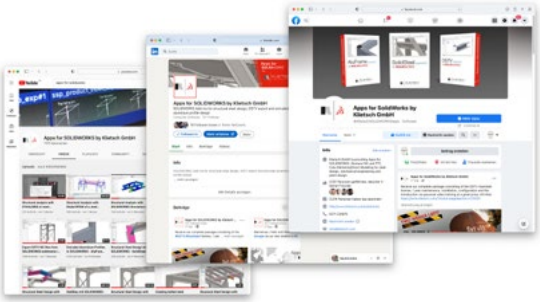

Social media

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Reseller worldwide

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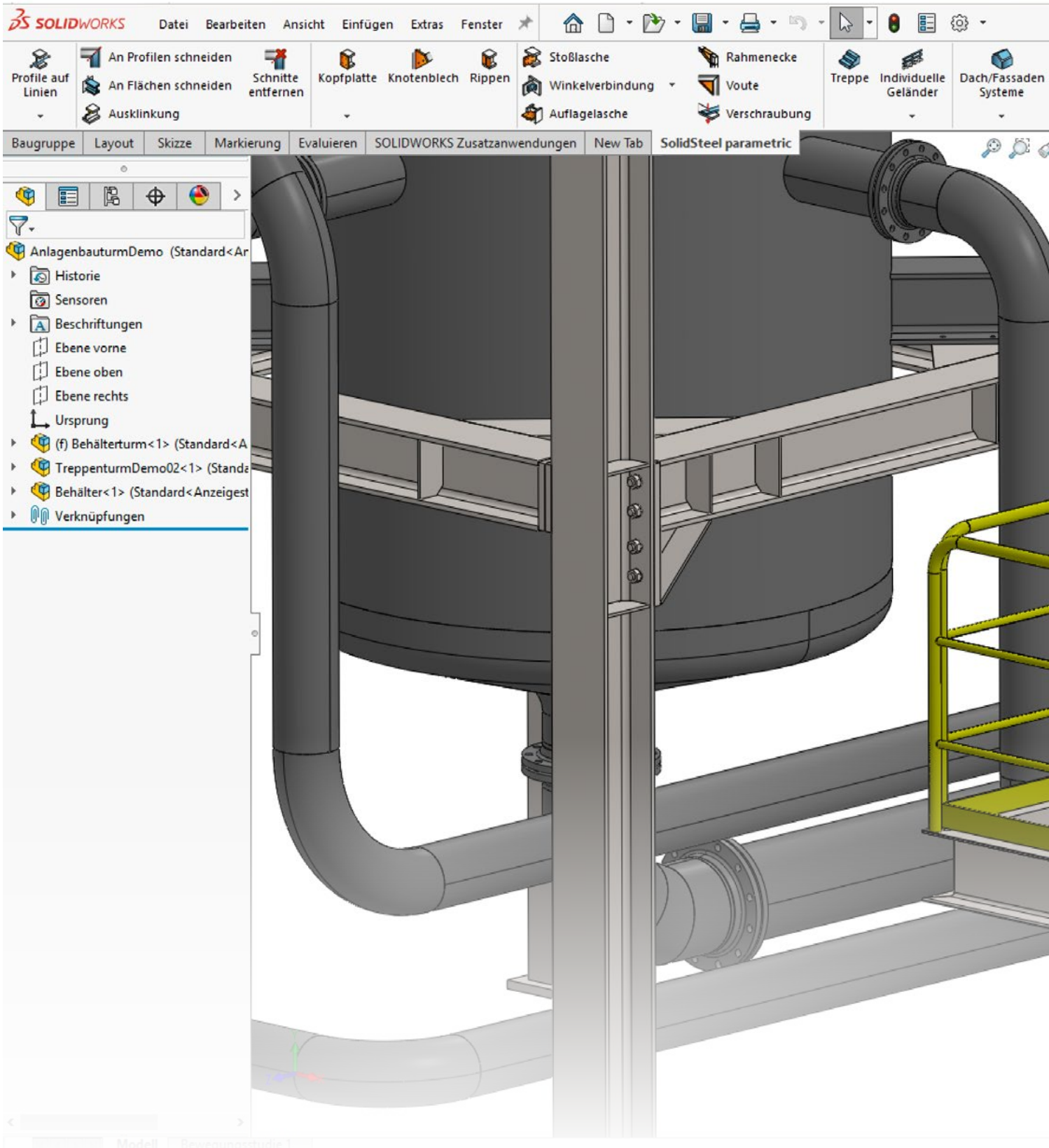
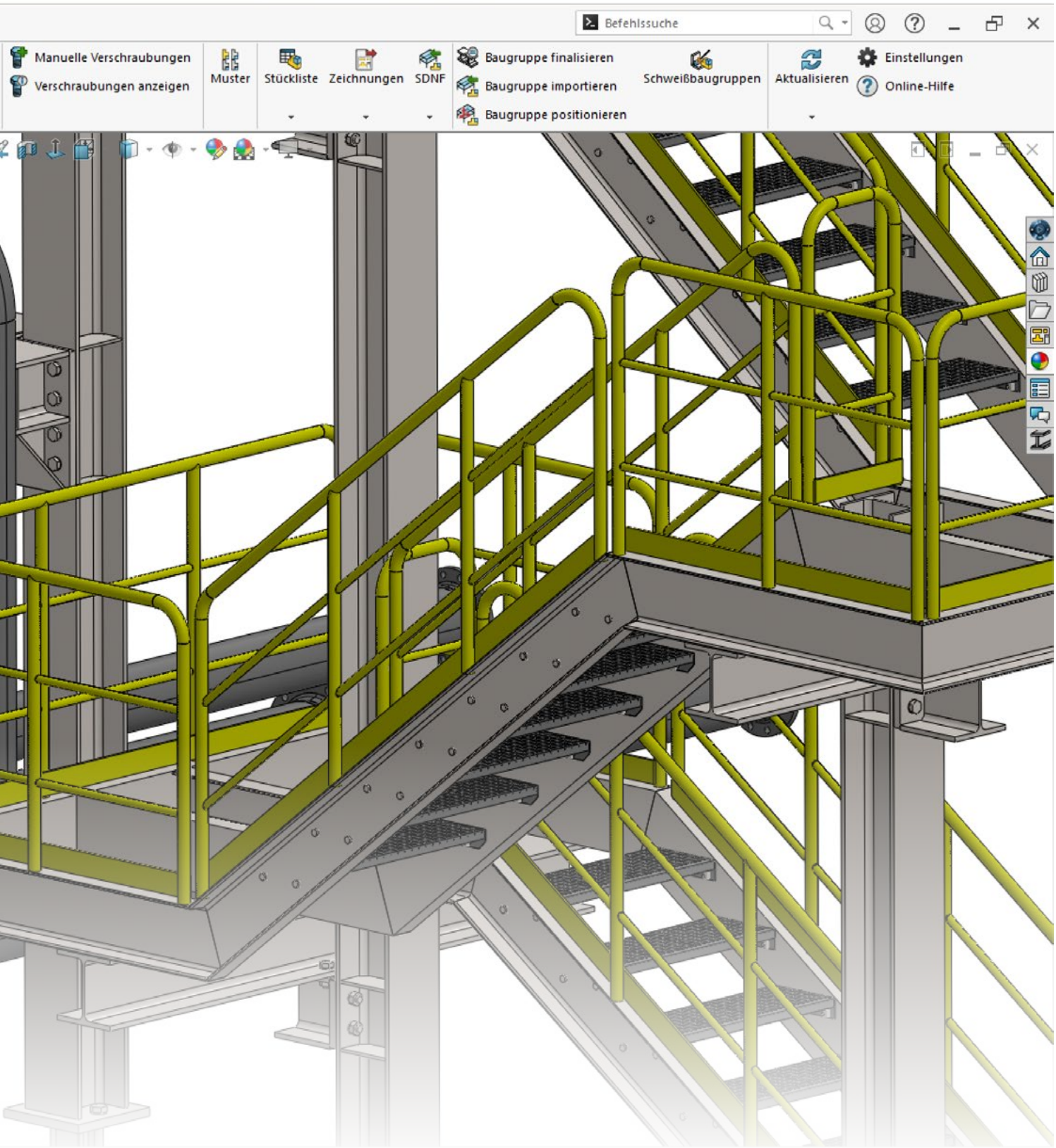


SolidSteel parametric

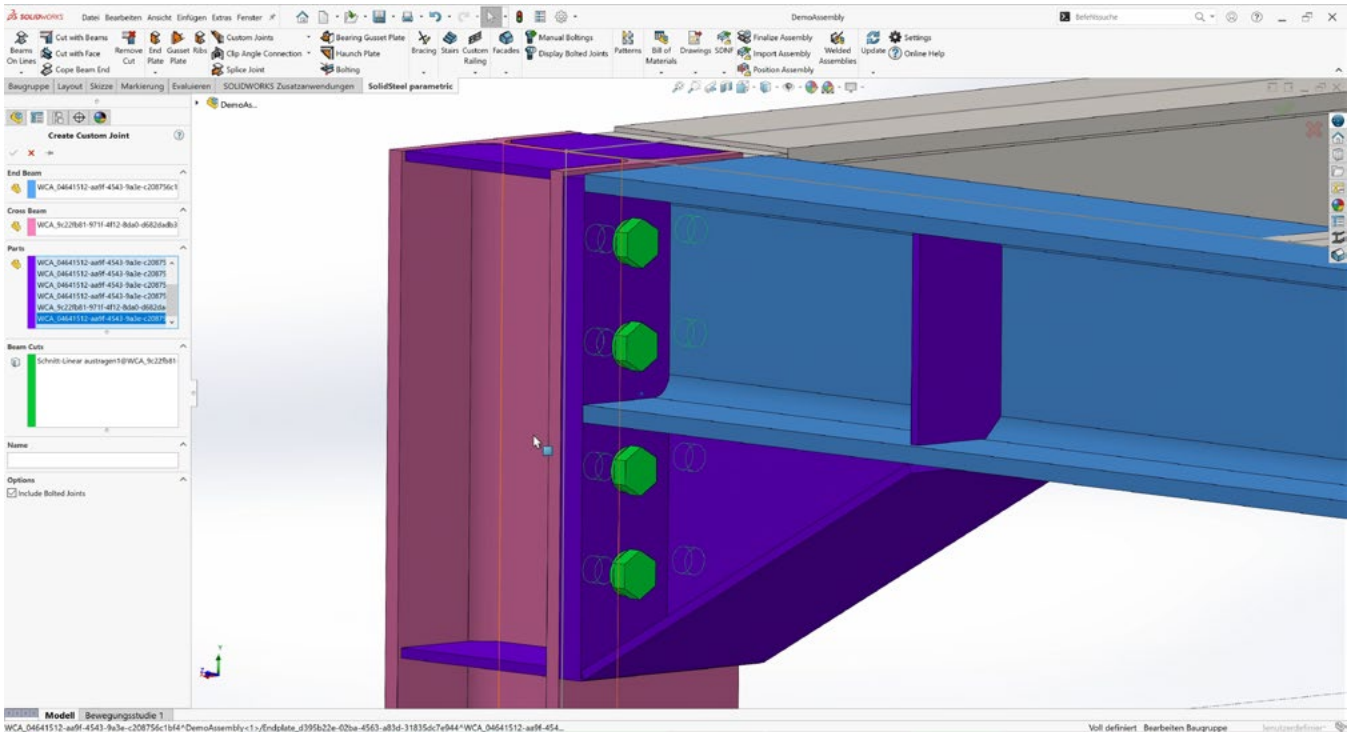
for SOLIDWORKS



Steel design with SOLIDWORKS

SolidSteel parametric is a parametric 3D CAD steelwork solution, integrated into SOLIDWORKS by 100%. Using the powerful and user-friendly functions for steel design, there are no limits to your creativity. The software is not branch-specific and can be used for classic steelwork, as well as for plant and mechanical engineering.

Create complex steel designs fast and efficient, including the typical steel connections. With SolidSteel parametric even complex connection types can be created with a few clicks and all connected beams will be extended, shortened or modified. Of course fully automatic and conforming to standards.



Typical steel connections

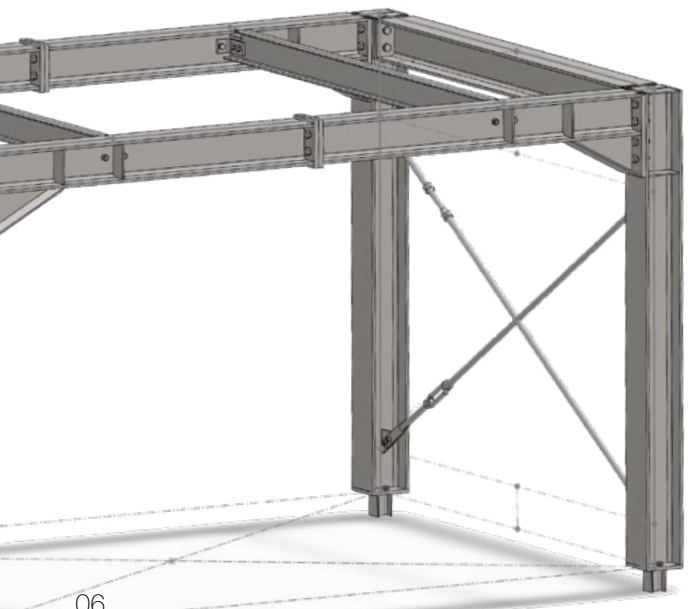
With SolidSteel parametric for SOLIDWORKS, typical steel connections such as end plates, gusset plates or clip angle connections are not only placed in seconds, but the software‘s intelligent selection also directly selects, for example, the end plate size that matches the profile.

Various libraries, such as DAST, are available for this purpose, which can be individually extended and shared with other users.

In total, more than 500 different joints can be created with SolidSteel parametric by combining different functions. SOLIDWORKS standard functions can also be linked to SolidSteel parametric functions.

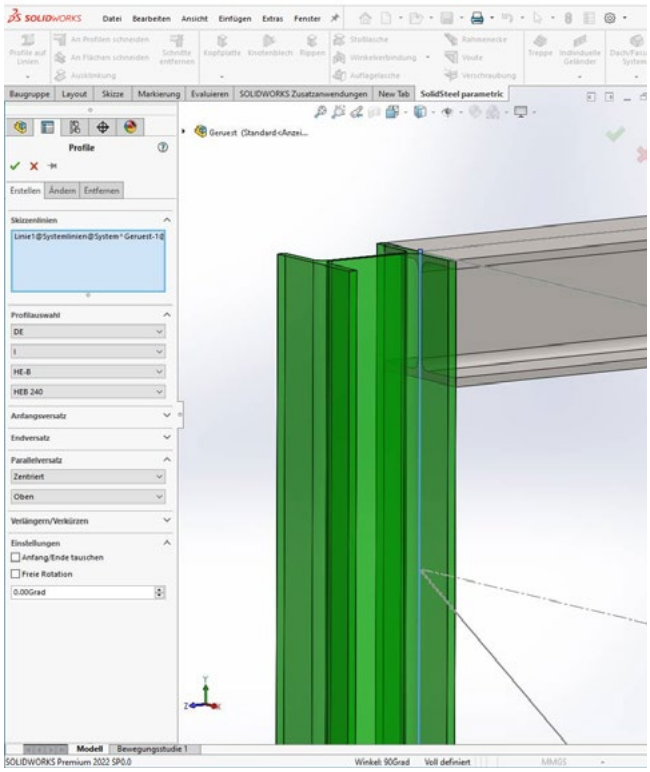
The „Custom Joints“ feature allows you to create even the most complex user-defined joints. The joint is created once, whether using SolidSteel parametric functions or standard SOLIDWORKS functions.
Then, with just a few clicks, the joint can be integrated into the database and reused whenever the geometric installation situation allows.

Have a look at the „Custom Joints“ video for all the details in about 6 minutes.



In addition to connections, steel structures typically include bracings such as cross bracings, K-bracings, and other shapes. SolidSteel parametric simplifies this tedious design task.
The bracing feature automatically generates bracings based on templates that can be customized. The use of standard parts such as turnbuckles or the definition of connection details is flexible.

All typical steel connections, including custom joints and bracings, are already included in the smallest package – the SolidSteel parametric Engineering Package.



Profiles & Libraries

SolidSteel parametric for SOLIDWORKS comes with an extensive profile library.
This includes among others:

- European profiles - DIN EN 10 034 / DIN 1025, T2
- Russian profiles - GOST, JIS
- US profiles - ASTM (mm & inch)
- British profiles - ASB
- Australian profiles - AS/NZS 3679.1 / AS/NZS 3679.2
- Chinese profiles - GB/T 706 / GB/T 11263
- South African profiles - SANS 10162-1:2011

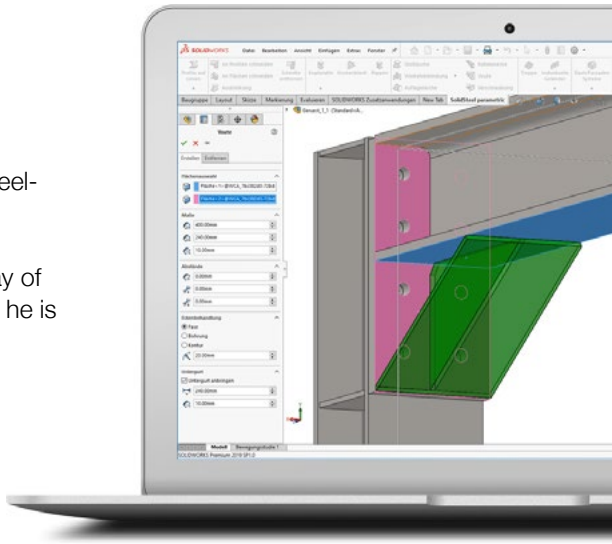
In total, more than 10,000 different profiles are available. Of course, the profile library can be extended with your own profiles and profile series.

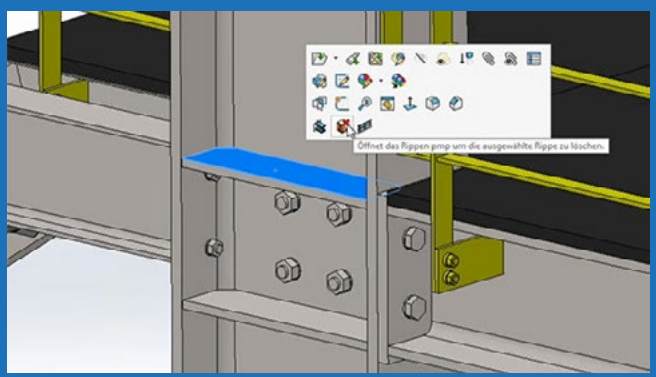
The position of all profiles can be changed independently of the system line. Profiles can be rotated around the longitudinal axis, moved parallel to the system line and the start and end points of the profiles can be swapped.
In addition, it is possible to set individual offsets at the start and end points of the profile.

Integration

SolidSteel parametric for SOLIDWORKS is not just a software solution for steel-work. It is integrated into SOLIDWORKS by 100%.

In the case of the user interface of SolidSteel parametric we adopted the way of the standard SOLIDWORKS user interfaces. So the user will not notice, that he is working with a third-party software product.



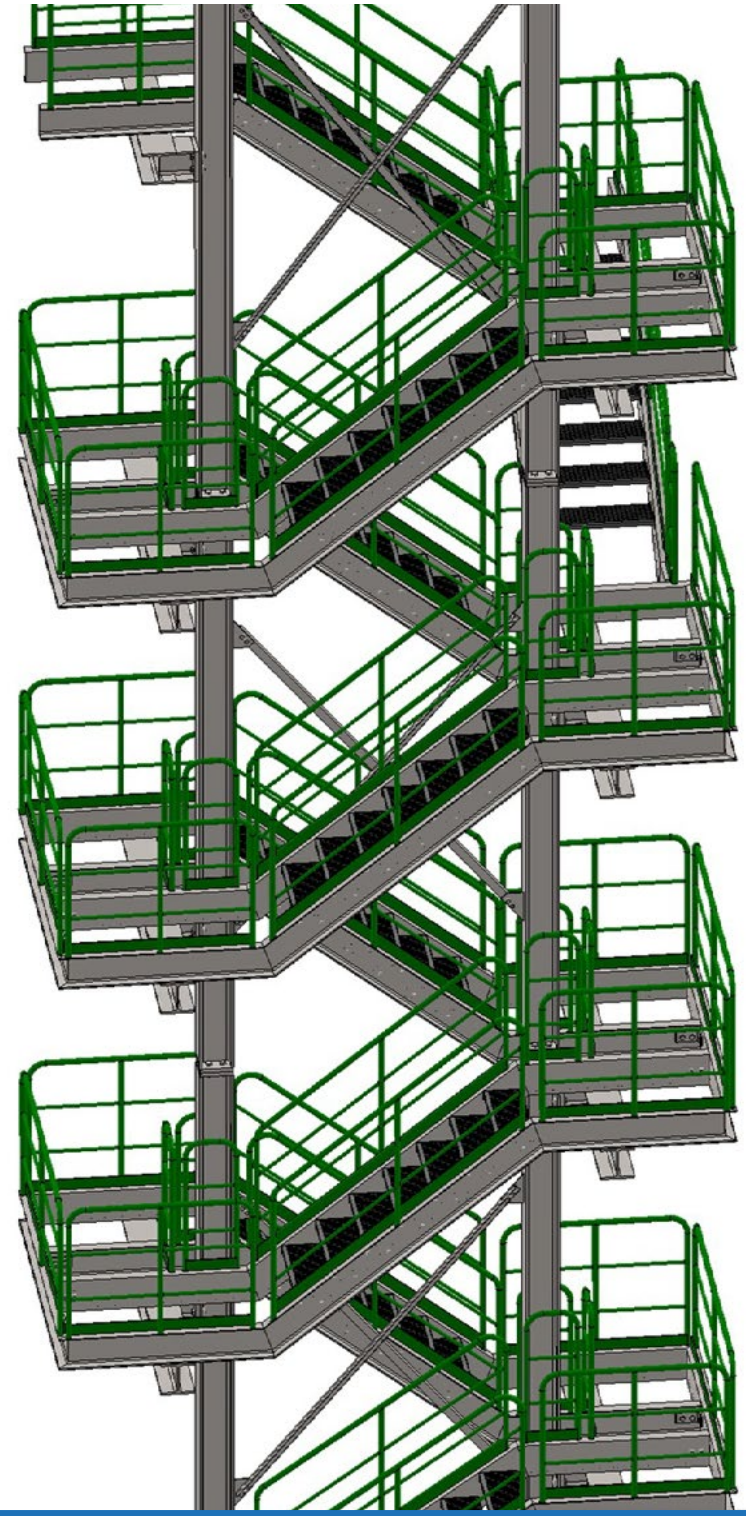


Context menus

The usual SOLIDWORKS context menus also speed up your workflow in SolidSteel parametric.

Depending on the selected reference, relevant functions can be accessed directly in the 3D area of SOLIDWORKS.

Then the corresponding dialog opens, where the selection fields are already filled as far as the context allows.



Stairs

Industrial stairs can be created in a few clicks and within seconds with SolidSteel parametric for SOLIDWORKS.

The start point and the endpoint of the stairs can be selected directly in the 3D model. As known from the other functions of SolidSteel parametric, the flight of stairs can be edited individually. Stair landing platforms can be created, offsets for floor heights can be added or the position of the stairs can be changed individually.
During the creation of the staircase, all profiles of the stair stringers are generated fully automatically, mitered and placed in appropriate welding assemblies.
Also, the stair steps will be placed, the boreholes will be placed into the stringer and the necessary bolts will be created.

Libraries

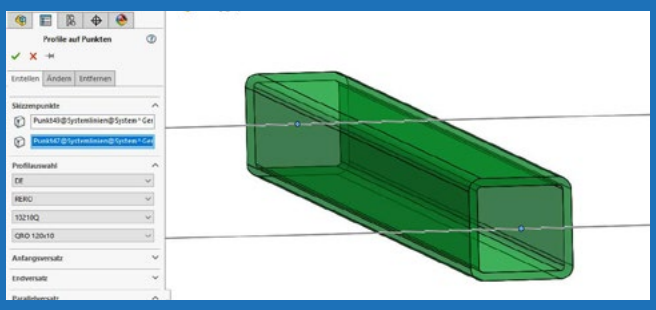
For the creation of the stair stringers the comprehensive profile library of SolidSteel parametric is available.
Exclusively for stairs, another library of stair steps is available that contains more than 100 different standard stair steps. Of course, this library can be extended with user defined steps. There are no limits.

Changes

If user defined limit values are crossed, SolidSteel parametric sends a warning to the user, during the creation of the stairs or a parametric change to their design, that the stairs might not be correct. So, the user always has the control, that the stairs fit into the requirements of the current project.

Good to know

Forgot a system line on the sketch? Or you just have a complex steel design? No problem at all!
With the intelligent functions of SolidSteel parametric for SOLIDWORKS you can easily place profiles from one sketch point to another.
The same functions are available to place the profiles from point to point than for placing them on sketch lines.
Of course, all the typical steel connections and trimming functions are available too.



Handrails

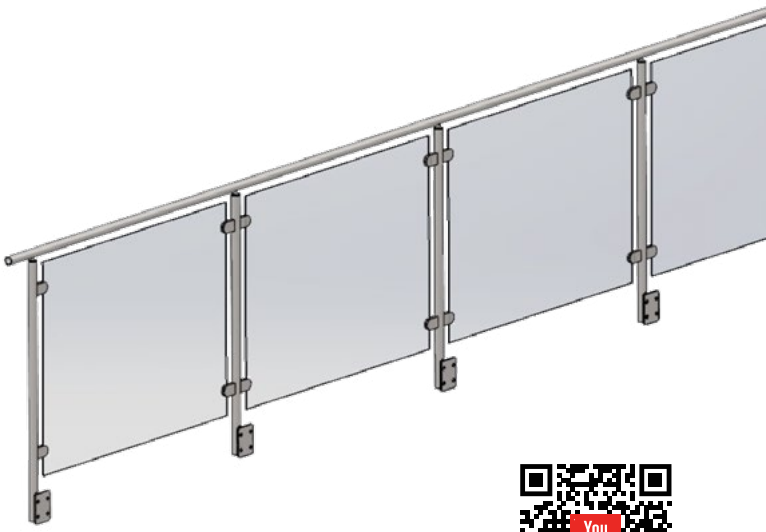
SolidSteel parametric for SOLIDWORKS offers the possibility to create segmented handrails in an automated way.

For this purpose, only a reference edge and a connection surface have to be selected in the 3D model.

Due to the individual railings it is possible to design almost all types of railings as long as they can be built with SOLIDWORKS functions.

It does not matter whether it is a conventional industrial railing with knee rails and baseboard or a complex harp railing or a stainless-steel railing with glass elements.

If the dialog is confirmed, the railing is created according to the user‘s specifications. Here, all the individual profiles (posts, balusters, baseboards, curved corner profiles, etc.) are inserted as parts into the design and, if desired, sorted directly into a subassembly.

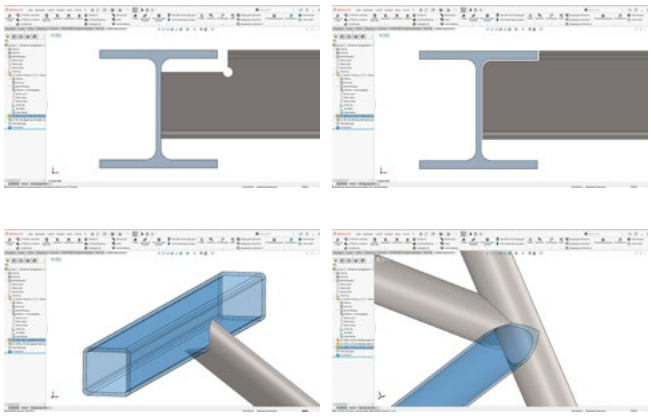


Simple & complex trims

Using the comfortable functions of SolidSteel parametric for SOLIDWORKS simple, as well as complex trim cuts can be done with just a few clicks.

No matter if it is a miter cut, a cope cut or a complex contour cut. After selecting the profiles, the cut will be added automatically. In all cutting functions a gap between the surfaces can be added. With this, tolerances can be changed or a gap for welding can be added.

In all cuts there are no external references or links between the parts. Nevertheless, all the edits are done in part context and not as an assembly feature, and the parametrics are 100% preserved.



And much more…

This brochure gives you just a small overview of the possibilities of SolidSteel parametric for SOLIDWORKS.

Visit our website for more detailed information or simply request your personal, free trial version and dive into the world of SOLIDWORKS structural design.

For more detailed information please visit www.klietsch.com or scan QR Code.

Facades and roofs

With the SolidSteel parametric Facades module, facade and roof elements can be placed from an extensive database with just a few clicks and are just as easily modified, deleted, edited and cut.

Using the Facade and Roof Elements function, facade elements and roof elements can be placed in the 3D model.

The outer contours and the cutouts of the facade are defined in a normal SOLIDWORKS sketch.
However, in many cases it is necessary to align the facade to other components. For this purpose, the SolidSteel parametric facade module provides countless alignment functions. There are almost no limits to your creativity, no matter if the panels of the facade should be aligned to another panel or a cutout, if the facade should have longitudinal joints or if the panels should be cut accordingly.

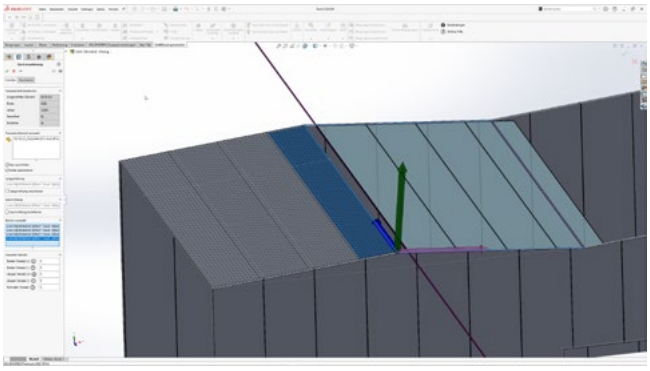
Already during the construction, overlaps of elements can be determined, which will be transferred to subsequent elements.

Of course, the components generated by the facade module are taken into account in the parts list as usual. However, additional facade-typical functions are available here, such as the fact that panels to be cut on site are entered in their full length on the parts list.

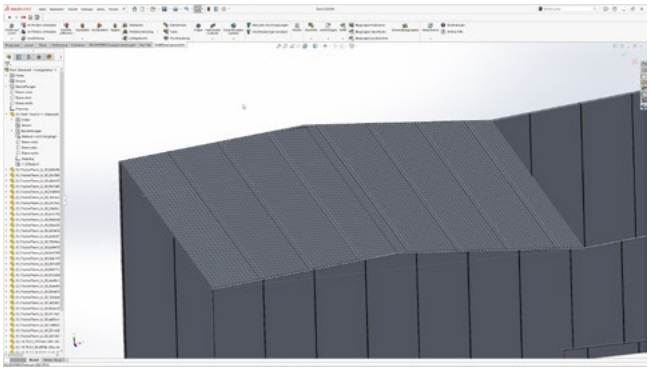
As in all current SolidSteel parametric functions, the facade and roof elements can of course also be edited and deleted via convenient menus.

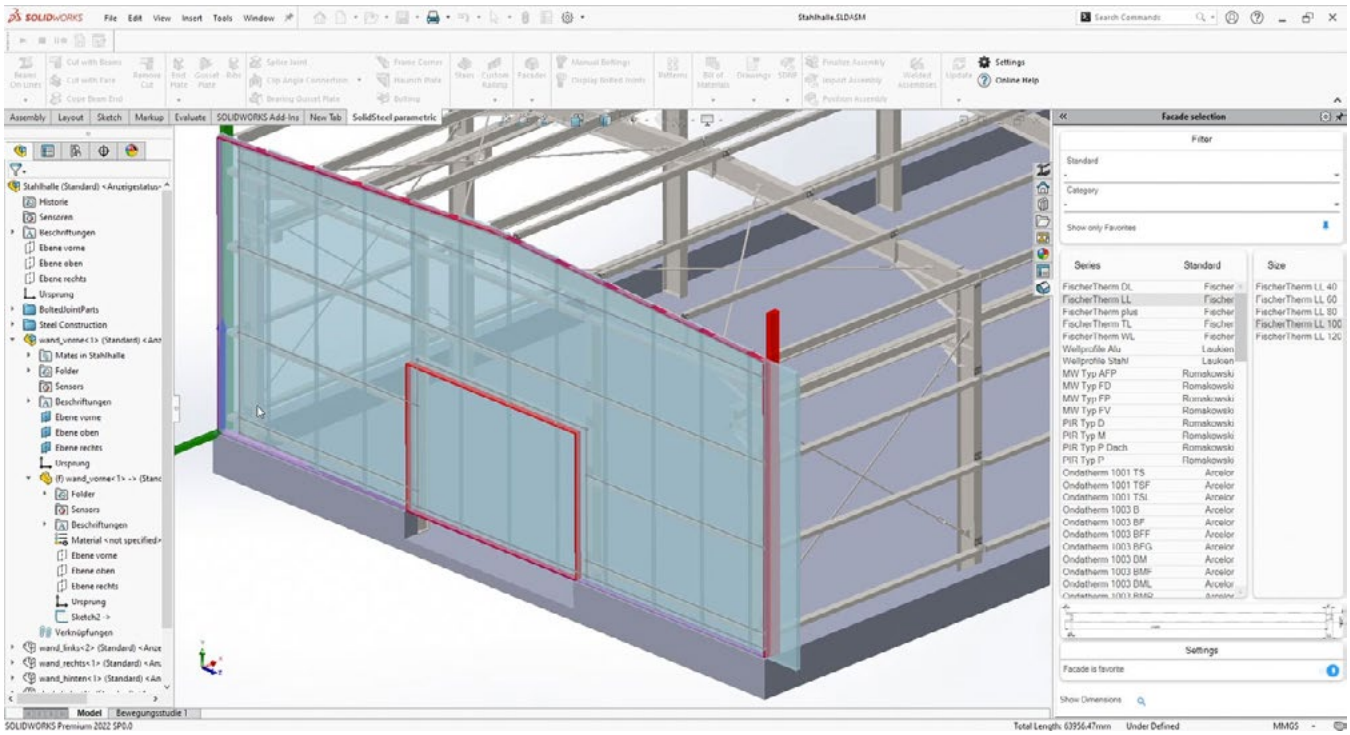
Roof extensions

The roof extension can be used to extend an existing roof area to a second, adjoining roof area.



In this process, all the properties such as profile types and offsets are adopted, and transitions between the two roof surfaces are created automatically.





Edge sheets

Although the function „Edge sheets“ is located in the Facade module, it is available to users (SolidSteel parametric Design Package and higher tiers) for all tasks where canted sheets are used.

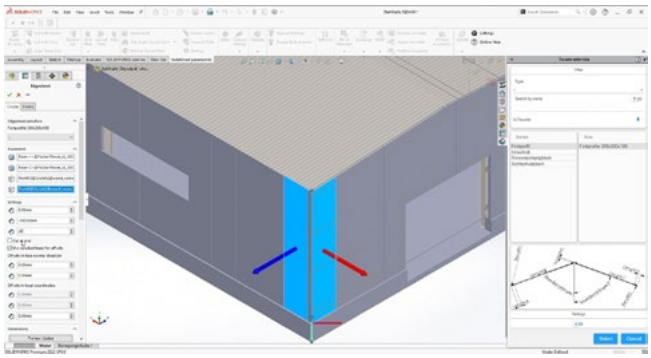
This feature allows profile-like flashings to be placed with an angle as a degree of freedom.

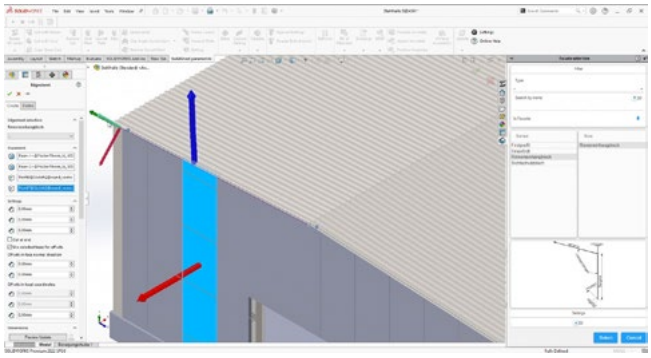
Examples of application would be eave sheets as transition between roof and wall elements or cover sheets at facade corners.

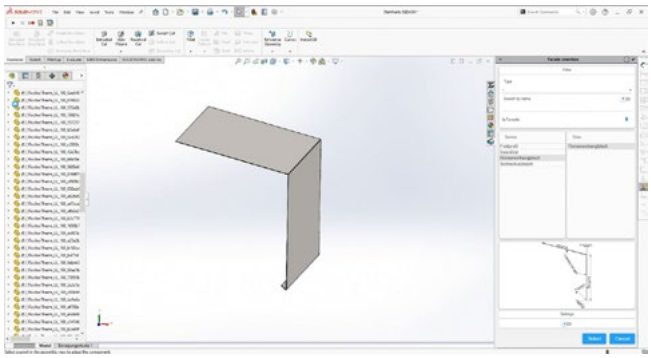
These sheets basically always look the same, but have to be adjusted in angle, e.g. due to the roof pitch.
By selecting both surfaces, this takes over the edge sheet function and enables the placement of sheet metal plates in different installation situations.

The flashings can be individually defined according to your specific ideas and specifications in the database and then be used again and again.

For subsequent processes such as automatic DXF export, the bended parts are processed and exported correspondingly with or without edge lines or automatic labeling.







Good to know

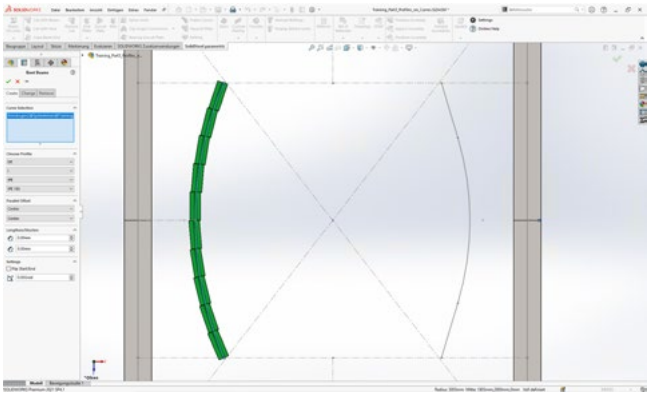
An important subsequent process of steel design is the export of NC data in DSTV Format. These data sets can be send directly to the manufacturing machines. No matter if sawing angles, cope cuts or boreholes. All this necessary information for manufacturing is stored in the NC data files. The DSTV Assistant is available as a separate add-in for SOLIDWORKS and is not only an excellent addition for all SolidSteel parametric users but also for all who work with imported data like STEP, IGES and similar files – **Find all details on page 16 or online via QR code.**

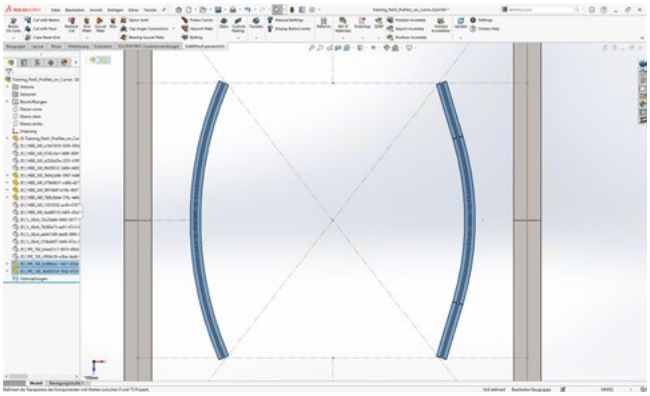
Curved profiles

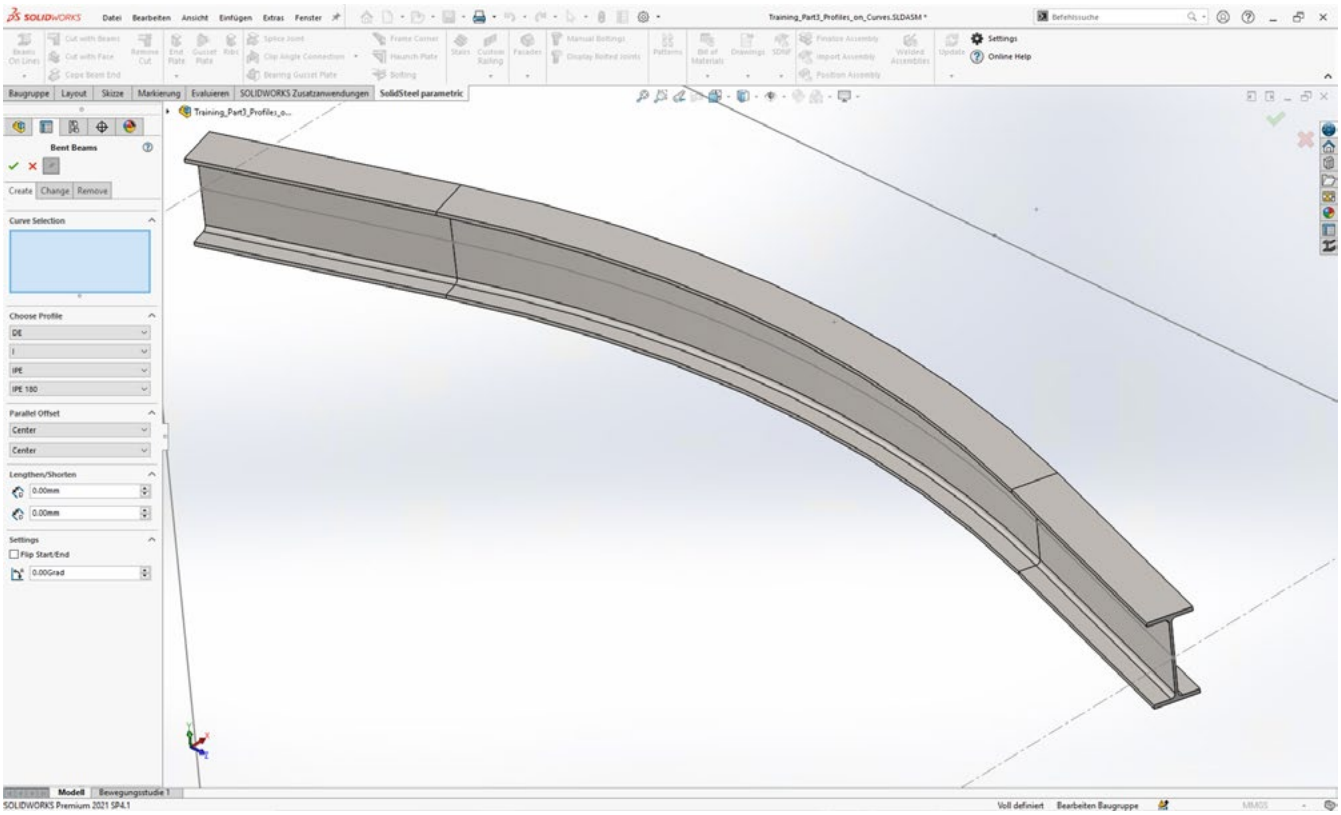
In addition to the familiar placement of profiles on straight lines and from point to point, SolidSteel parametric now also allows profiles to be placed on circular arcs as well as on line segments with circular arcs.

Of course, these profiles can be offset, extended, or shortened and rotated parallel to the line with the usual placement options of SolidSteel parametric.

In addition to surface and miter cutting, end plates, miter plates and ribs are of course also supported in curved profiles. Other joints connected to the curved profiles also will work fully, as long as the joint type is suitable.







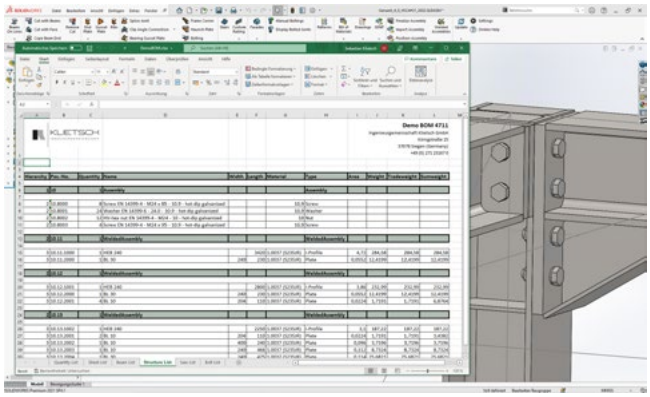
Curved profiles in subsequent processes

In subsequent processes such as the equal part detection, the parts lists (unwound length) as well as the NC data (machining in straight parts), the curved profiles are taken into account accordingly.

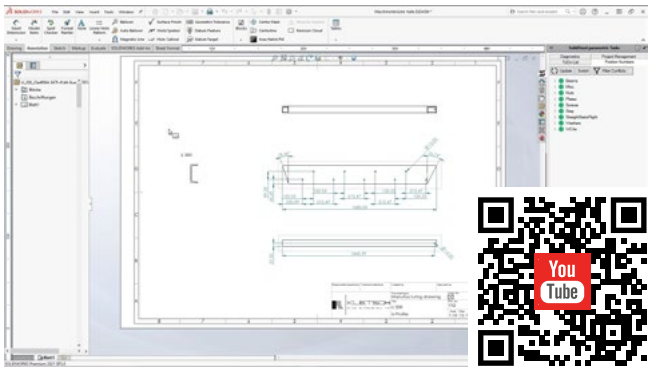
Bills of materials, drawings, labeling & Co.

In addition to the design tools, SolidSteel parametric for SOLIDWORKS offers convenient functions for subsequent processes in steel construction, such as the automatic creation of shop drawings, the export of parts lists of all kinds, export of DXF files or cooperation with external structural analysis software.

Automatic assignment of item numbers provides the basis for most of the subsequent processes. In this process, all parts in the assembly are analyzed. Geometrically identical parts are assigned the same position number.
Individual number ranges can be defined for different groups of parts, such as profiles, sheets, etc.
The functions of SolidSteel parametric always ensure proper order and check for any errors that may exist, such as whether dissimilar parts have the same position number or other relevant information.

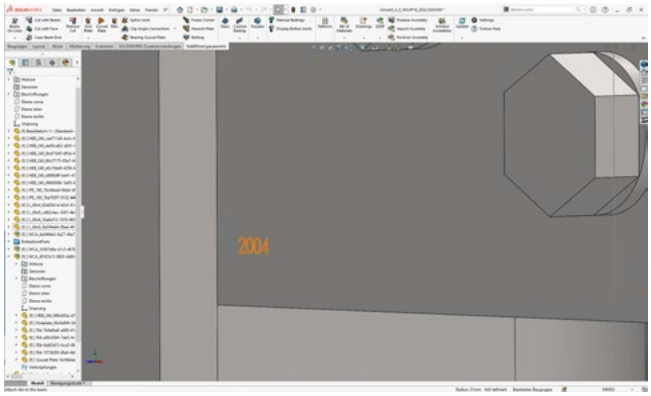


Bills of materials can be created with just a few clicks and in a matter of seconds.
For the BOMs, customizable Excel templates are available, which clearly represent your design in lists of all kinds, such as quantity lists, sheet metal lists, profile lists, structure lists, saw lists or screw lists, including the important parameters such as dimensions, materials, weights, etc.



Shop drawings of assemblies and individual parts can be created as easily as parts lists with just a few clicks.
In the case of individual part drawings, all dimensions required for production are automatically inserted.

The function „Finalize assembly“ combines all geometrically identical SolidSteel parametric parts within the assembly.
The result is that for all identical parts only one reference part (main part) is left in the assembly and all other identical parts are instances of this main part.
This feature is especially interesting for users who work with PDM systems or not with virtual parts.



For the DXF export, SolidSteel parametric for SOLIDWORKS provides automatic, individual labeling of the parts.
Thus, for example, the position number can be labeled fully automatically on all sheet metal parts.

Even more information on the bills of materials?

Here we go:

With SolidSteel parametric for SOLIDWORKS, additional, cross-section-specific, user-defined properties can be added to profiles directly when they are created.
All you have to do is edit the library data and as soon as a profile is created, it is assigned the new user-defined properties immediately upon creation.

Structural analysis

Structural mechanical calculations are an integral part of steel design.
With SolidSteel parametric for SOLIDWORKS there are basically two different ways to perform a structural analysis.

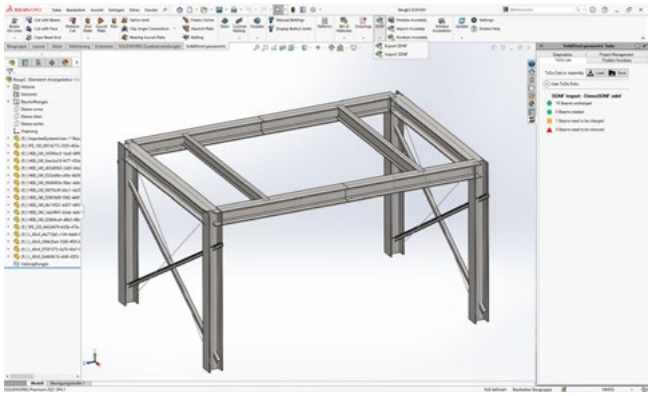
Since SolidSteel parametric for SOLIDWORKS generates native SOLIDWORKS parts and assemblies, a structural analysis can easily be performed with SOLIDWORKS simulation. The results are of course adequately accurate, but are not prepared according to standards and still need to be processed for submission to an inspection authority.

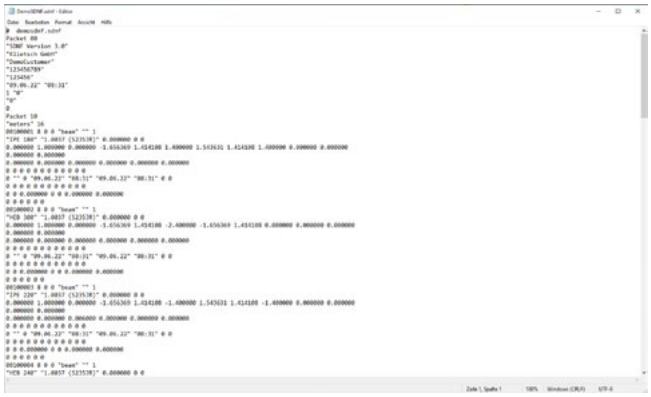
In order to create a standard-compliant structural analysis that can be submitted directly to an inspection authority, it is usually necessary to work with external calculation programs. The SDNF interface is available for this purpose. SDNF stands for „Steel Detailing Neutral Format“ and is a kind of steel construction PDF. This format contains all the information necessary for the structural analysis, i.e. the type and position of the profiles.
With SolidSteel parametric for SOLIDWORKS, SDNF files can be imported and also exported.

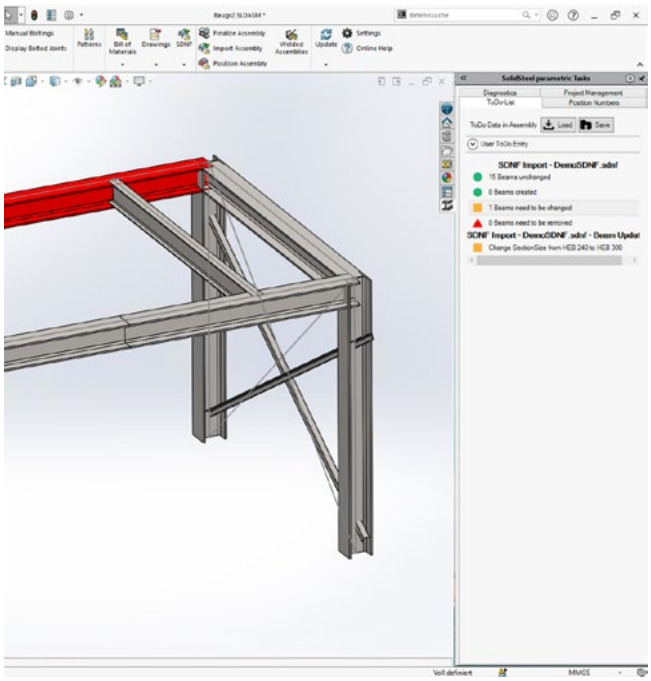
If the static calculation was created prior to the design, it can be transferred directly from the calculation software and does not have to be rebuilt again in CAD. SolidSteel parametric creates all profiles and necessary sketch lines. Afterwards the model can be detailed with the comfortable functions.

If the design was created first, it can be transferred directly to the calculation software with the SDNF export of SolidSteel parametric.

If during the structural analysis it has been determined that profiles have to be changed, removed or added, an SDNF file can be exported from the calculation software and imported into SolidSteel parametric.
During this import, SolidSteel parametric compares the SDNF file with the existing model and creates a convenient to-do list for the user, containing everything that needs to be adjusted in the model.



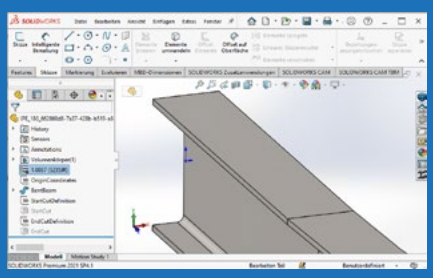




Good to know

SolidSteel parametric for SOLIDWORKS not only supports metric and imperial units, but also allows you to easily work with different materials such as mild steel, aluminum, plastic or stainless steel.

For this purpose, the profile database uses the material library of SOLIDWORKS and you as a user can assign a material to each library part individually.



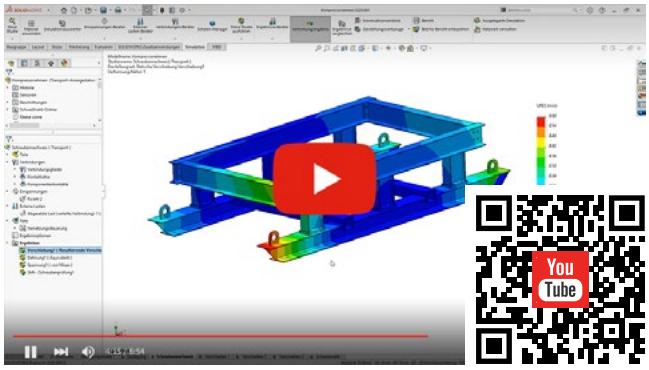
Video series with different methods for structural analysis

With SolidSteel parametric, SOLIDWORKS becomes a professional 3D CAD steel construction solution.
There are many examples and videos that show the design process and demonstrate how to get from the idea to the finished and detailed design quickly and efficiently with SolidSteel parametric.

But the structural analysis is also part of the design!

No matter if it is a simple deformation analysis or a complete static analysis according to standards. SolidSteel parametric offers various possibilities for static calculations.

As in most other cases, many paths lead to the goal. The same applies to structural analysis. We have summarized three common methods in videos for you:



In the first part, we take a closer look at the finite element method with SOLIDWORKS simulation.

An interesting example is a compressor frame, which of course was designed with SolidSteel for SOLIDWORKS.

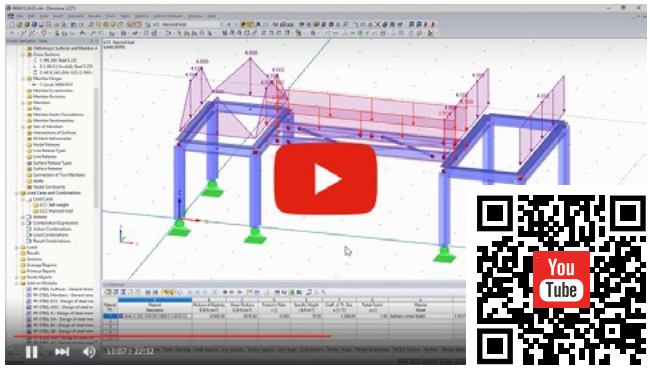
Wolfgang Müller from DPS Software GmbH shows us the calculation in detail.

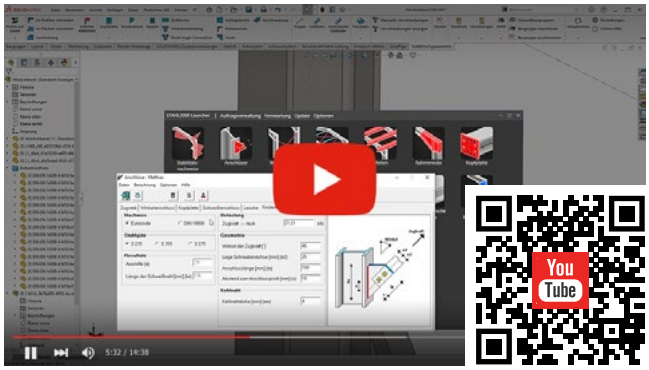
(6:54 min.)

The second part is about the calculations with external 3D framework programs and we use the SDNF export to easily transfer the static system and other important parameters for the structural analysis to third parties.

Daniel Dlubal from Dlubal Software GmbH shows us the structural analysis in detail with their software RFEM 5.

(22:32 min.)

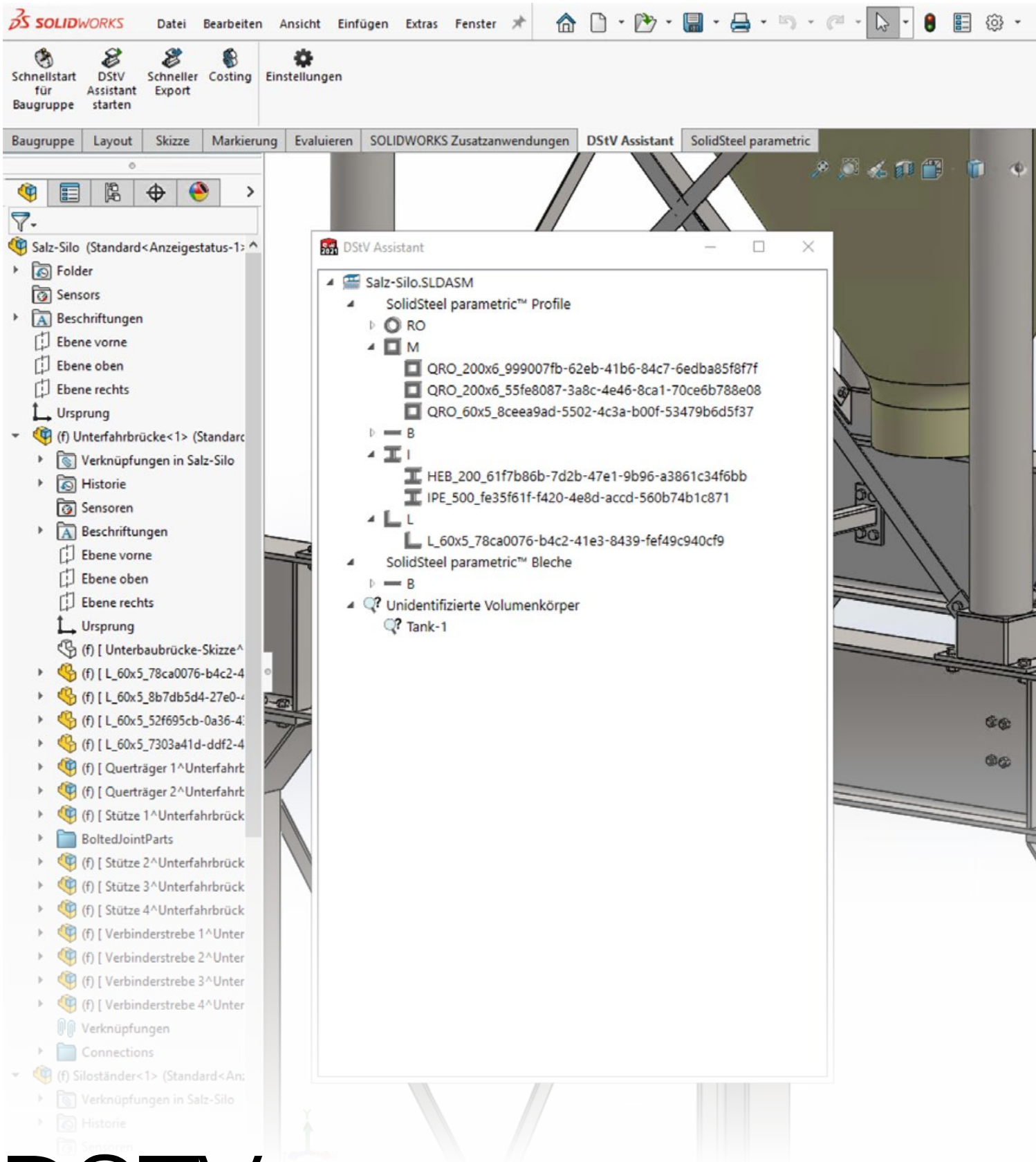




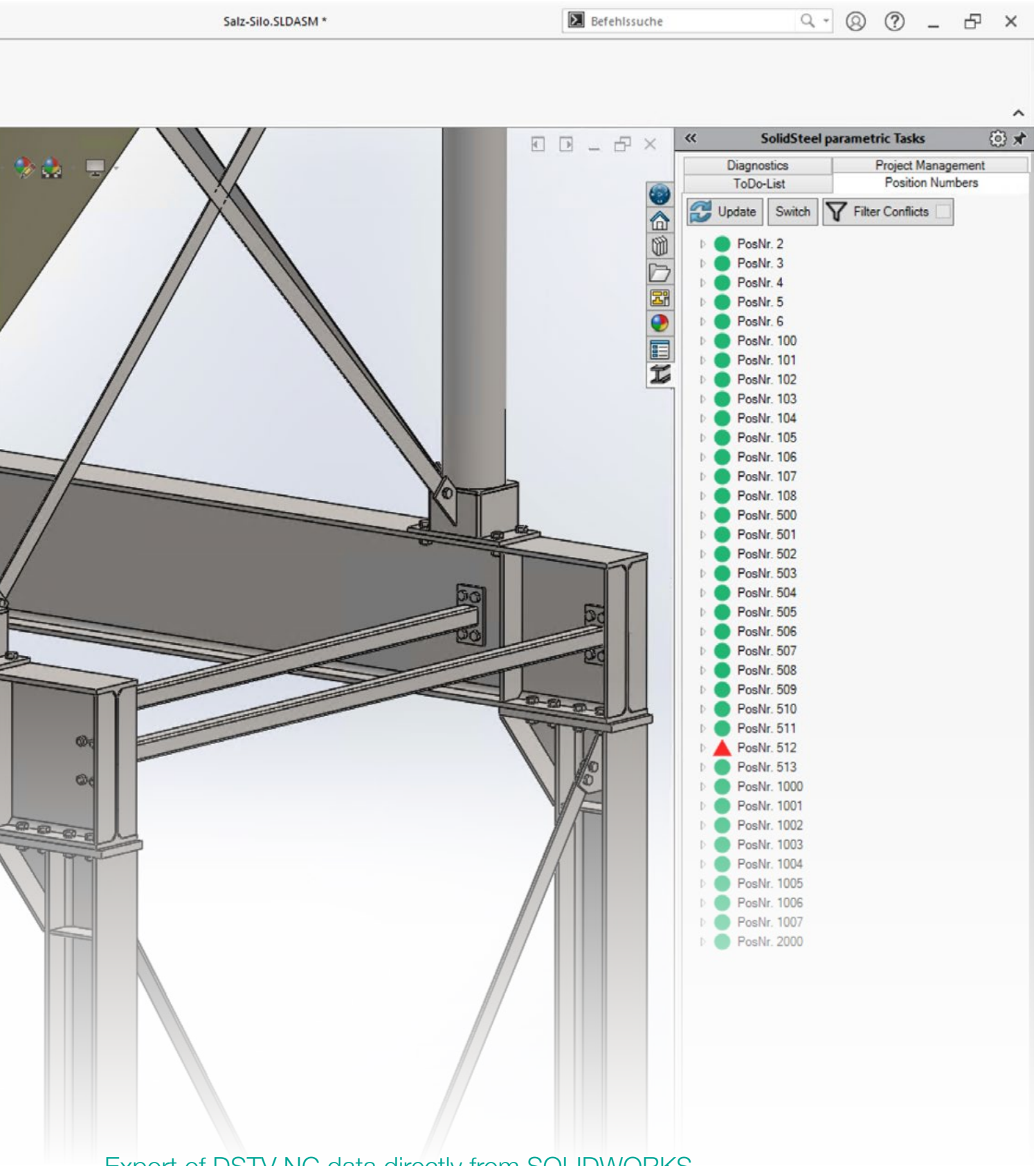
In the third part we look at the fast, design-accompanying calculations of individual components with STAHL 2000.

The STAHL 2000 software series also originates from our company, offers specialized individual programs for many frequently required calculations and leads quickly to the result through intuitive operation.

(14:38 min.)



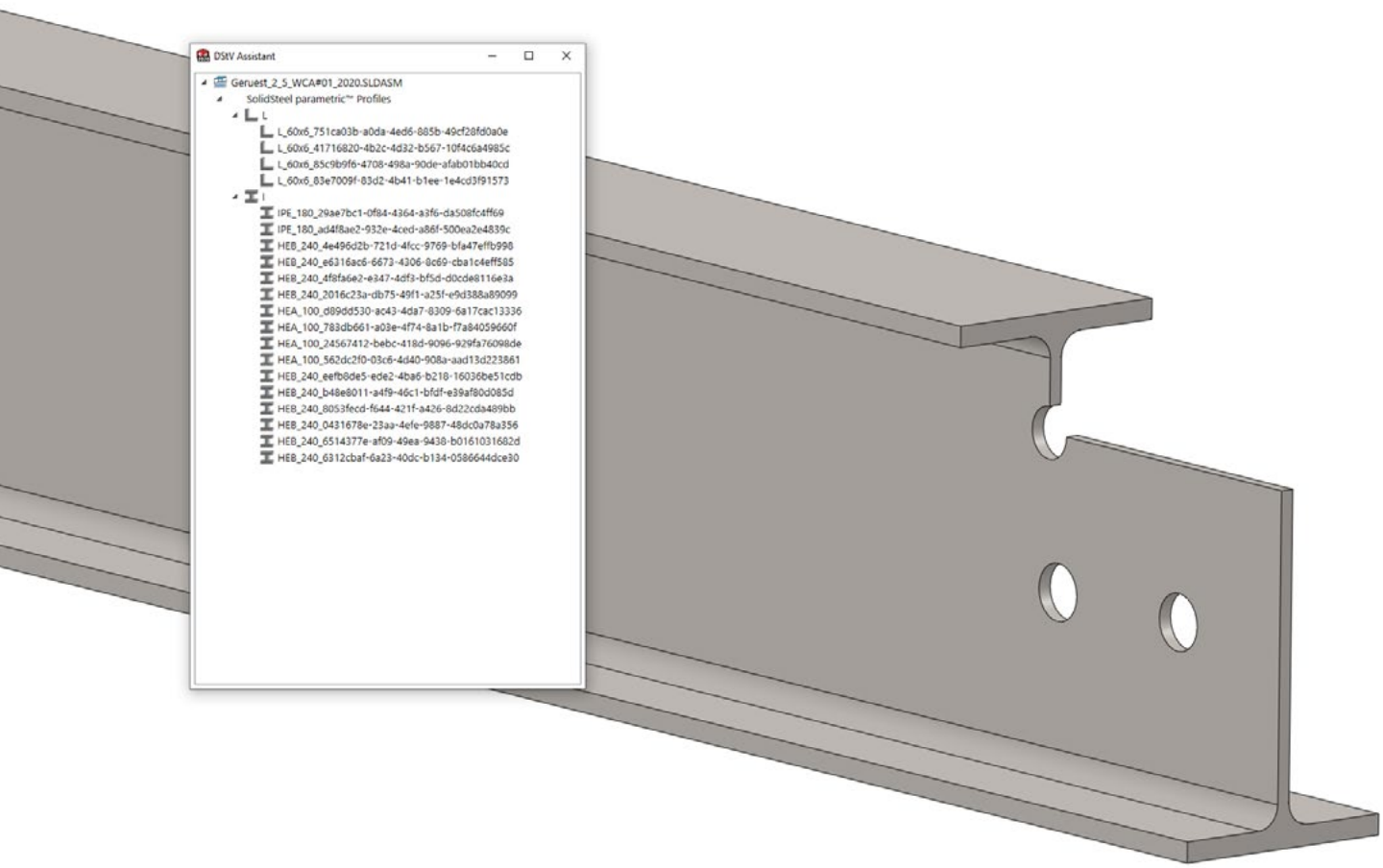
DSTV Assistant for SOLIDWORKS



Export of DSTV NC data directly from SOLIDWORKS

With the DSTV Assistant for SOLIDWORKS, you can export NC data from steel components (profiles and sheets) in DSTV format directly from SOLIDWORKS and pass them on to machine production.
You can transfer NC data directly from SolidSteel parametric, SOLIDWORKS weldments, SOLIDWORKS Structure System or export any imported files such as STEP, IGES and others.

The DSTV Assistant can be used either directly in the design department or in the classic way in work preparation and can be seamlessly integrated into internal processes.

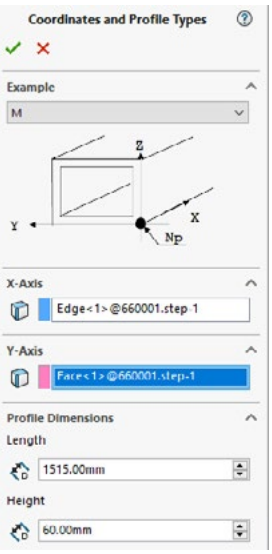


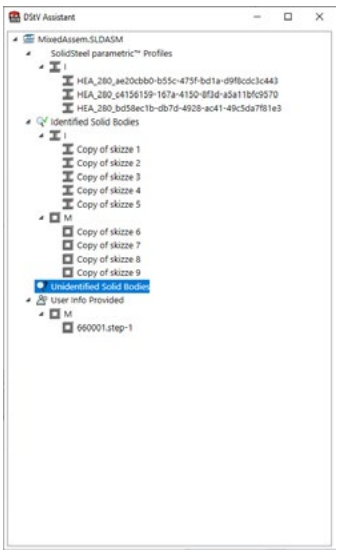
Clever functions

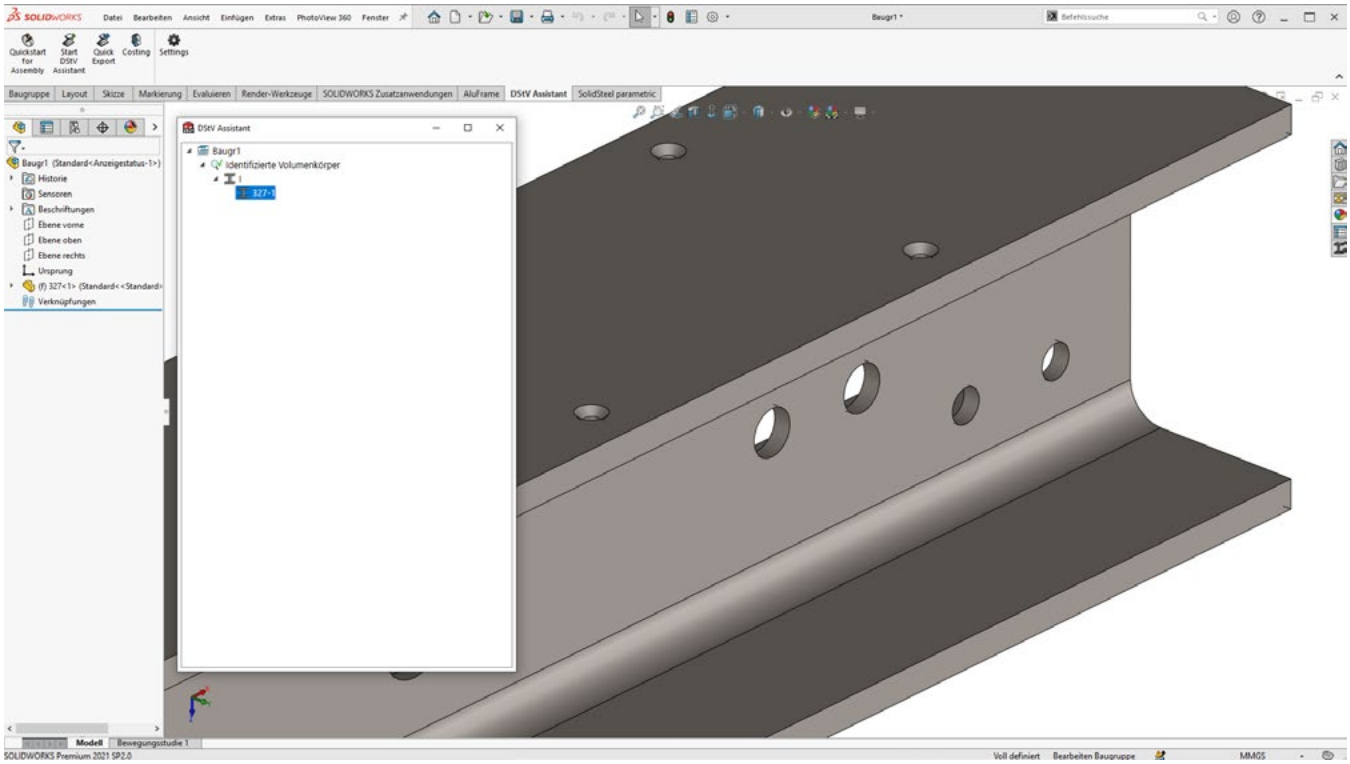
If NC data needs to be generated from a 3D solid body, the DSTV Assistant analyzes the component and determines the profile type or sheet orientation as well as all processing such as holes, cutouts, chamfers and many more.

Thanks to the intelligent functions, the NC data in DSTV format can be generated very quickly and automatically. However, if, for example, the profile type is not known or the sheet metal alignment is not clear, manual entries by the user may be necessary. Here too, the DSTV Assistant provides you with the best possible support through clearly structured queries for the missing information.

In summary, the DSTV Assistant is the perfect addition for all users who work with structural steel components in SOLIDWORKS and offers automatic functions for exporting NC data in DSTV directly from SOLIDWORKS.
The source of the data is almost irrelevant as long as SOLIDWORKS can import the data.







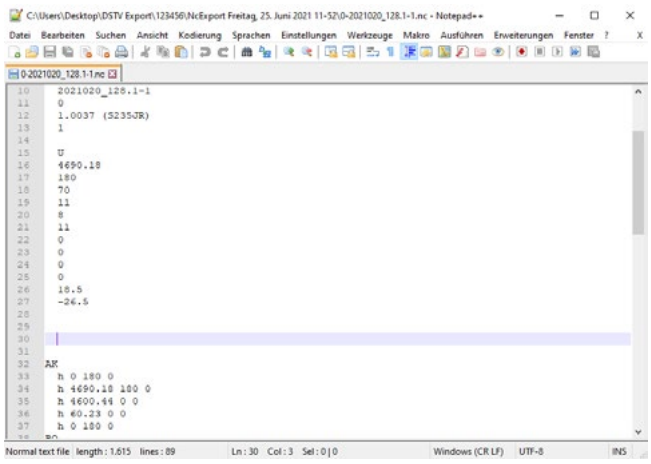
DSTV NC data in a few minutes

The DSTV Assistant automatically checks all parts in the assembly to determine whether they are steel profiles or sheet metal parts.
It doesn‘t matter whether the data comes from SOLIDWORKS Welded Construction, SOLIDWORKS Structure System or SolidSteel parametric.

Of course, equal parts are also combined directly in this process and the required number of parts is written in the header of the NC file.

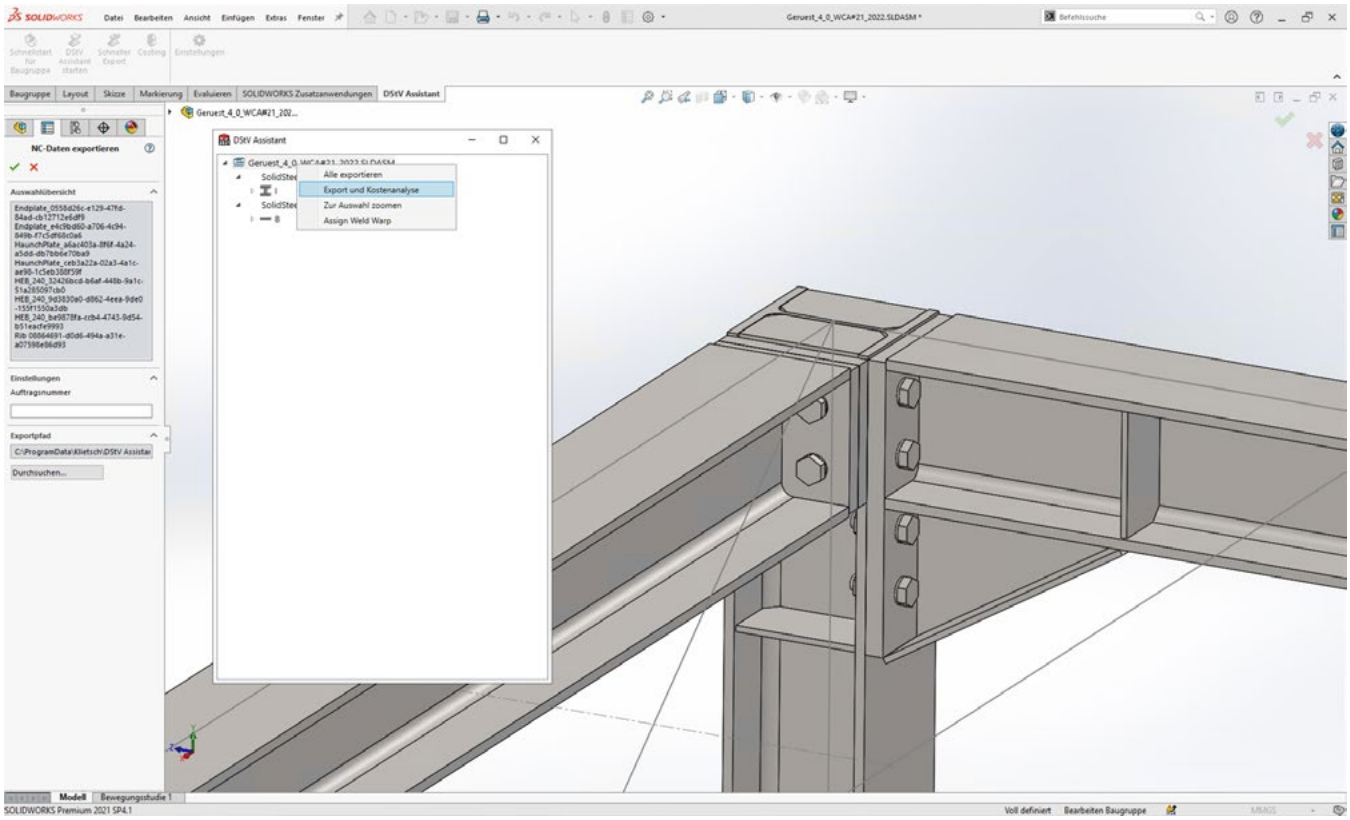
After all parts have been recognized, only a storage location and an order number have to be assigned and the DSTV NC data is exported within a few seconds.

Optionally, the files can also be saved directly to your PDM system. In this case, the names of the files are created according to your specifications and you can pass the data directly to production.



Did you know?

You want to create DSTV NC data, but the information like item number, name or other metadata comes from your ERP system? That‘s no problem – DSTV Assistant can take all the header data of the DSTV NC file individually from the user-defined properties of the part and then automatically insert it into the NC file.



Knowing component costs beforehand?

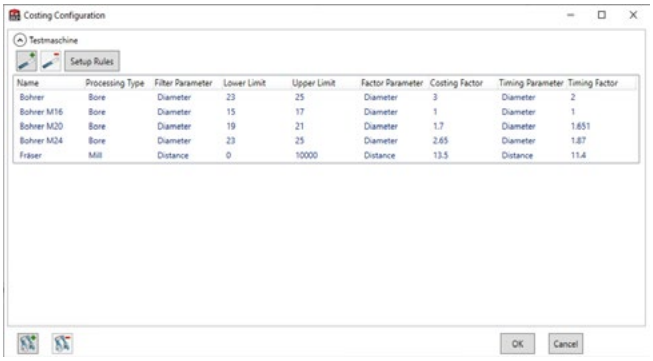
With the costing module of the DSTV Assistant this is no problem!

The task of this function is to estimate the costs for machining in advance. Once a part has been analyzed with the help of the DSTV Assistant for SOLIDWORKS, a list is created with the necessary machining operations and the corresponding cost factors.

Using an intelligent mapping table, a price and machining time is determined for each machining operation and saved accordingly.
Subsequently, the list of machining operations including the corresponding prices as well as external information such as set-up times, times for tool changes, gross and net weights, etc. is written to an XML file and can be processed manually or automatically by ERP systems.

Tool change

Tool changes result from the different types of machining in the DSTV document. If two M20 bores and four M24 bores have to be produced, a tool change is required.
The basis for this is a tool table that contains information about which machining operations can be performed with which tool.



Key data for the price calculation

The price calculation takes into account or includes:

- Profile type and size
- Type of machining
- Time of machining
- Tool change, if necessary, according to routine
- Output of net and gross weight
- Manual input of correction factors
- Manual input of single or all times / costs
- Output of results in neutral XML format
- The output can be stored in the PDM system

The entire process chain in steel construction

Steel construction has tradition, steel construction endures, as can be seen well in buildings that have defied the times for centuries.
But even traditional construction methods are subject to the changing times and with our apps for SOLIDWORKS you are well prepared for the present as well as for the future of steel construction!

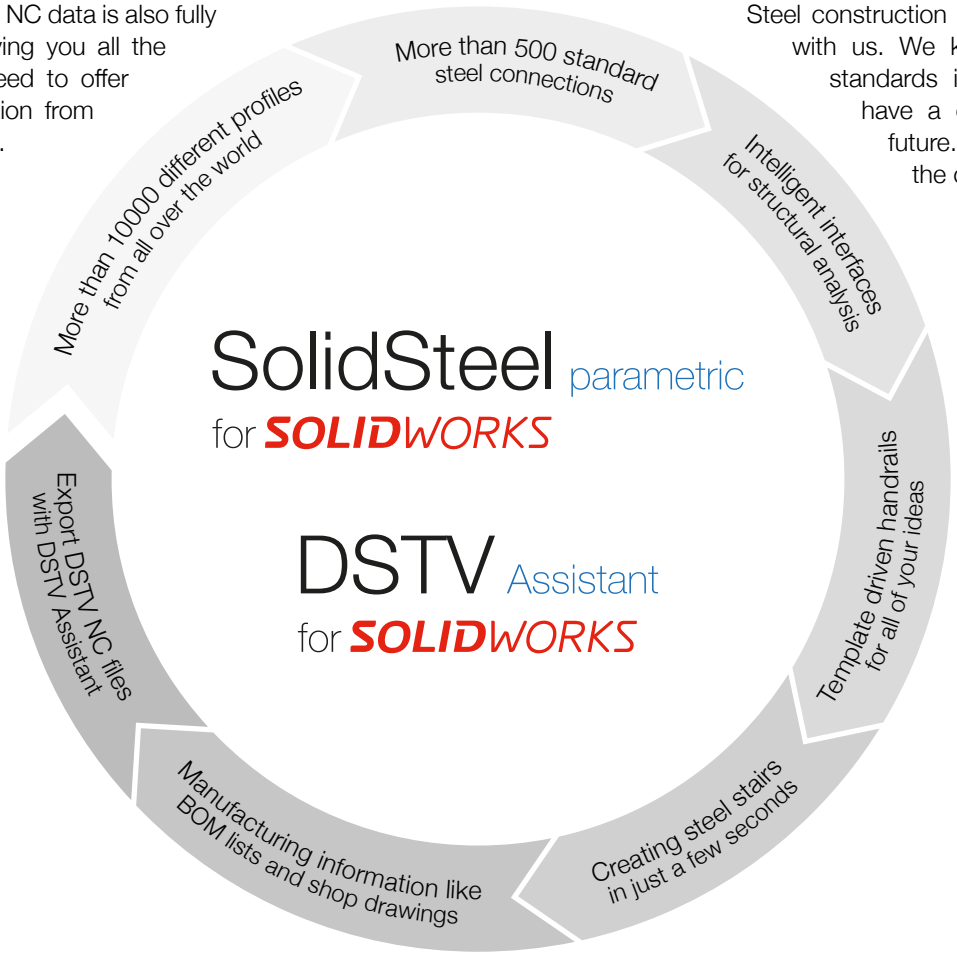
SolidSteel parametric for SOLIDWORKS, is the leading solution for the realization of steel design in SOLIDWORKS and offers the right tools and functions from the first sketch to exchange formats for structural analysis, to the transfer of the completed design to manufacturing.

With the apps for SOLIDWORKS, the process chain does not end when the design is finished as a 3D model in the computer. With the DSTV Assistant, the export of DSTV NC data is also fully automated, giving you all the options you need to offer steel construction from a single source.

Even without the NC data export of the DSTV Assistant, the standard functions of SolidSteel parametric can cover the classic follow-up processes such as the export of bills of materials or the automated export of shop drawings (individual parts and assemblies).

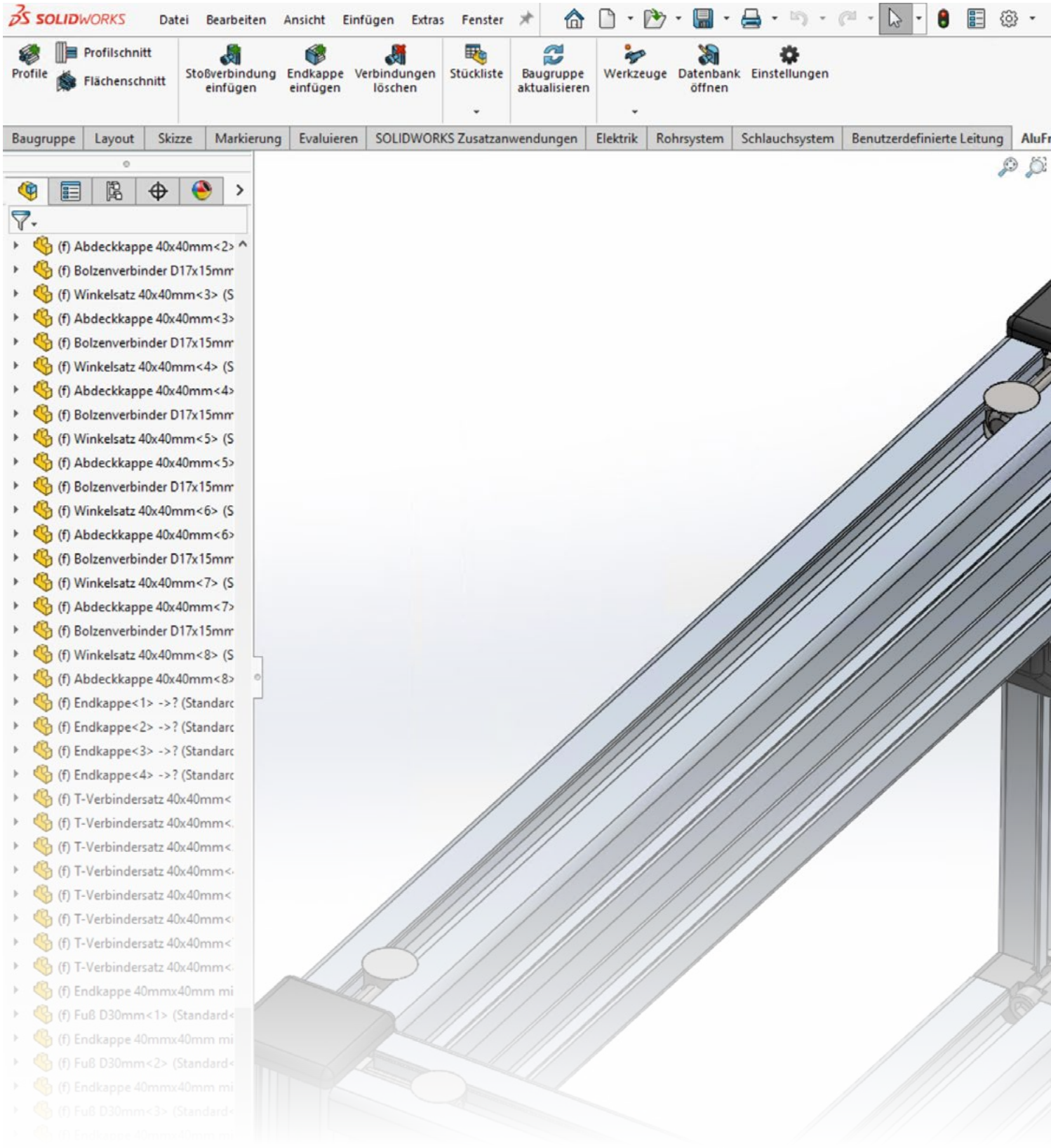
Our apps for SOLIDWORKS are 100% integrated into SOLIDWORKS and the user interface follows the graphical specifications of SOLIDWORKS to such an extent that users tend to forget quickly that they are working with an external solution and no longer with board tools from SOLIDWORKS.

The Ingenieurgemeinschaft Klietsch GmbH is a software development company in the CAD sector and based in Germany for more than 45 years.
Steel construction has a long tradition with us. We know the past, set standards in the present and have a clear vision for the future. Become a part of the community!



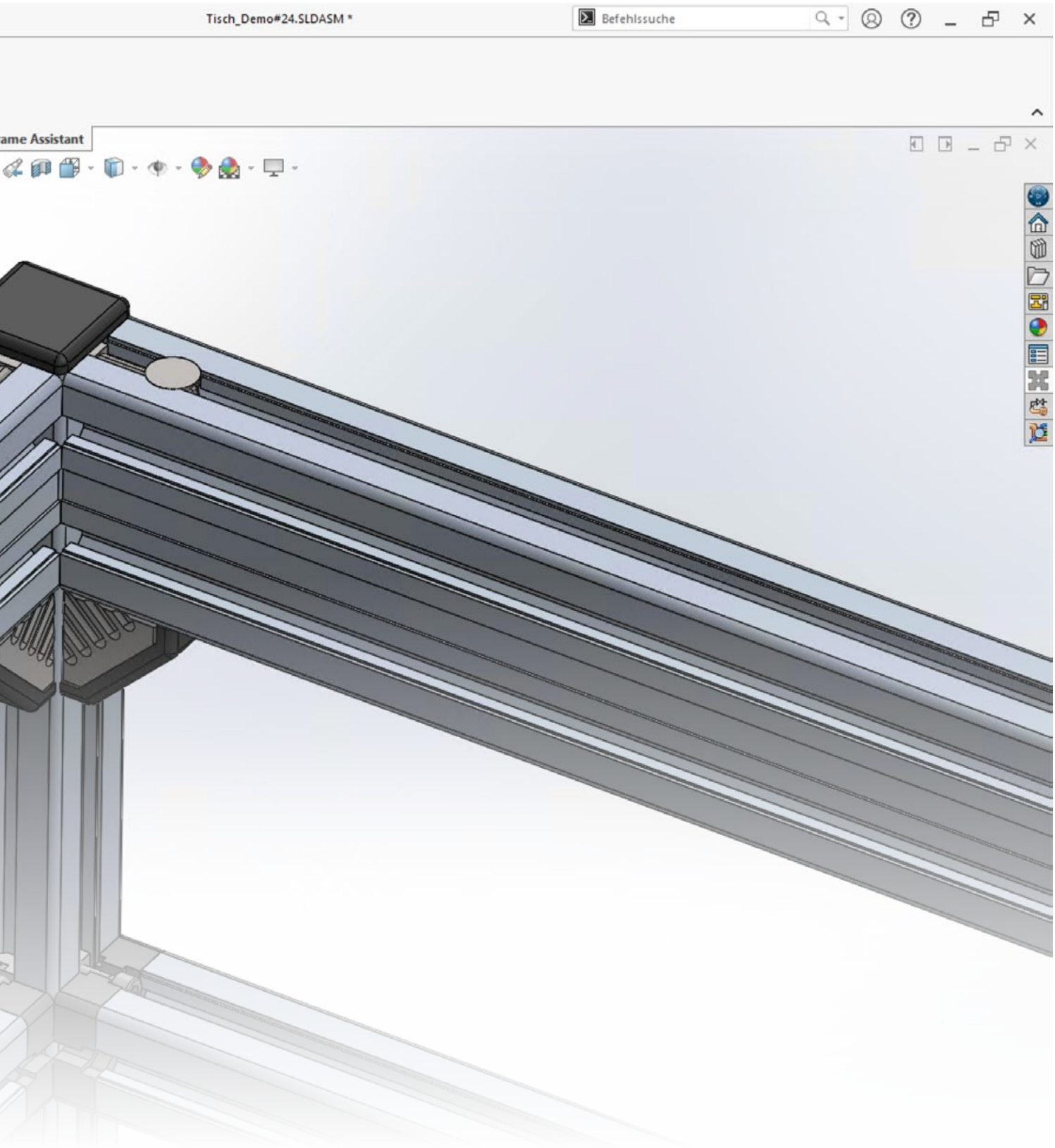
Would you like to see how the DSTV Assistant works?

You have the option to send us one or more 3D models. We are going to convert those models into DSTV NC data with the help of the DSTV Assistant. Then you will receive the test data and can check it internally.
For more information please visit www.klietsch.com



AluFrame Assistant

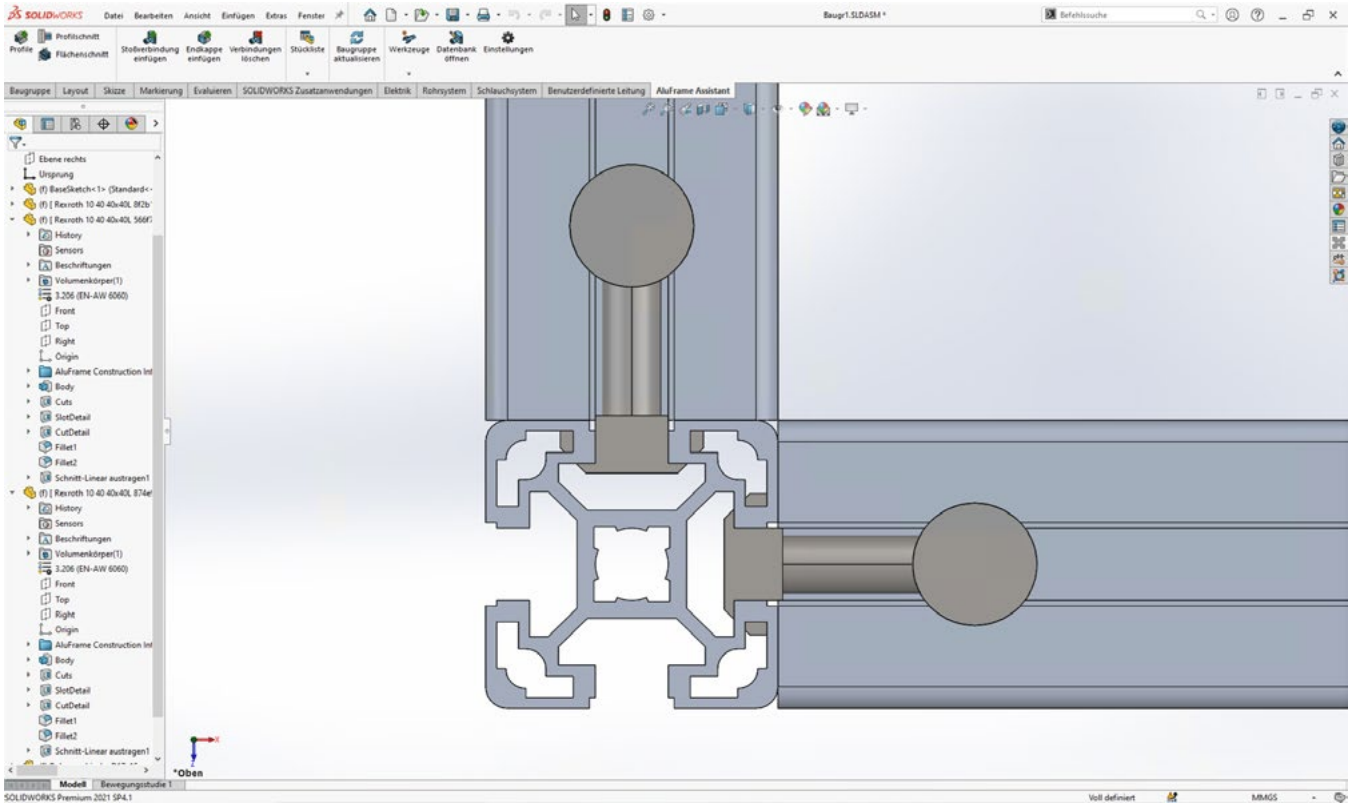
for SOLIDWORKS



Aluminium profile design with SOLIDWORKS

The AluFrame Assistant is a parametric 3D CAD software solution which is integrated into SOLIDWORKS by 100%. With the powerful and user-friendly design functionalities for aluminium profiles and connections of different well-known manufacturers there are no limits to your creativity.

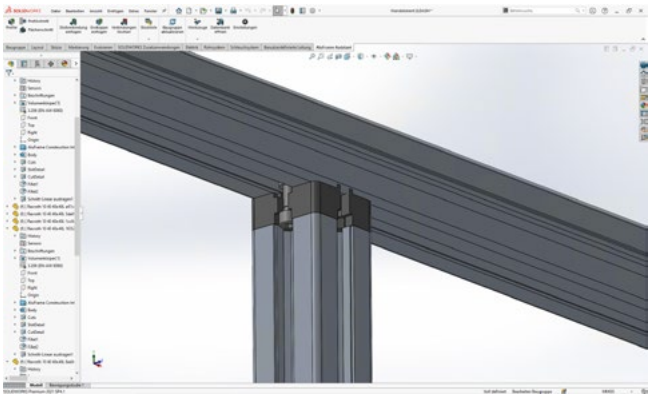
Create complex designs including the typical connection parts with or without boreholes fast and efficiently. After the design there are different possibilities to create bills of materials, order lists, manufacturing drawings or the export of manufacturing data.



Connections

Comfort was very important for the design process. With the AluFrame Assistant for SOLIDWORKS typical connections can be placed within seconds. Additionally all the necessary things like boreholes or profile trims will be done automatically.

In addition to standard connections of many different manufacturers it is possible to add connections to any degree of complexity. This is a one-time process and afterwards the connection is available in your library. That way you are not limited to single manufacturers or given ways of design.

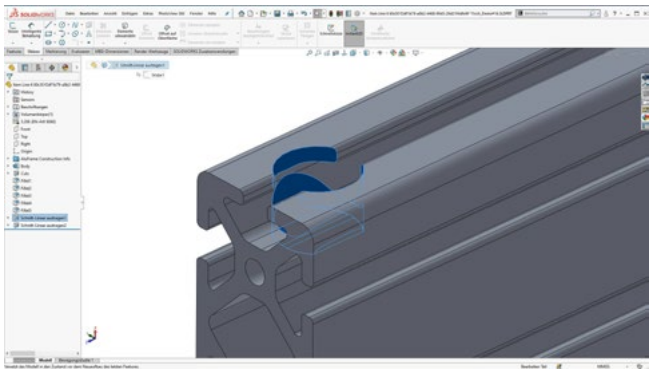


Smart databases

The AluFrame Assistant has a self-learning database. This means that it is possible to teach-in new connections within the graphical interface of SOLIDWORKS.

To do this, the connection is placed and linked once with normal SOLIDWORKS functions and then taught to the AluFrame Assistant.

The profile type, the slot size, the position of the connection and also all user-defined properties are then automatically stored in the database.

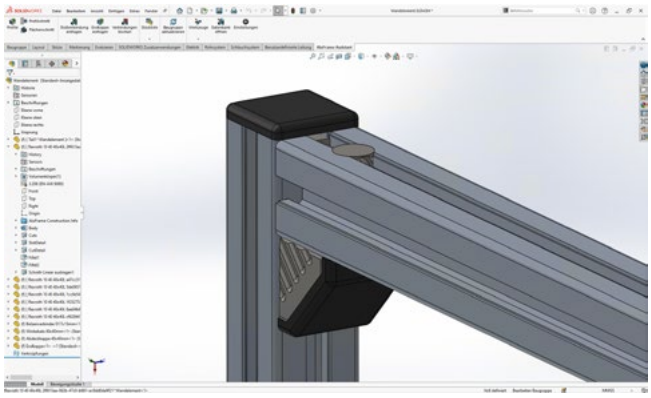


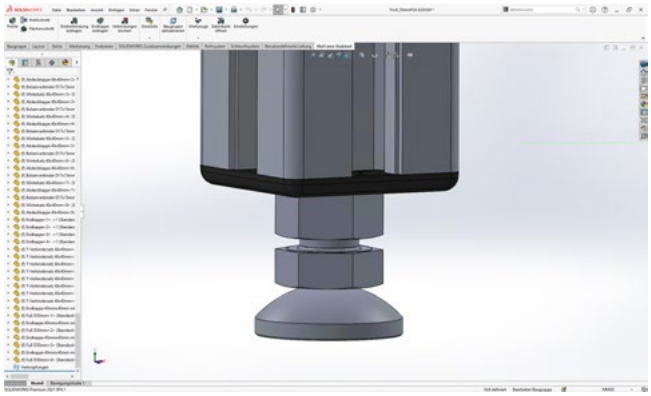
Thus, not only is the connection placed automatically, as described above, but additional information such as part numbers, order numbers and the like are also managed automatically.

Standard connections

Standard connections can be added to the design with just a few clicks.
Simply select both profiles, the software then suggests all the connections stored in the database that match the profile manufacturer and the slot size. Once the connection has been selected, the position can be determined.

Afterwards, everything happens fully automatically.
All parts of the connection are placed, a subassembly is created (only if the connection consists of several individual parts) and any necessary processing, such as drilling the profile, is carried out.





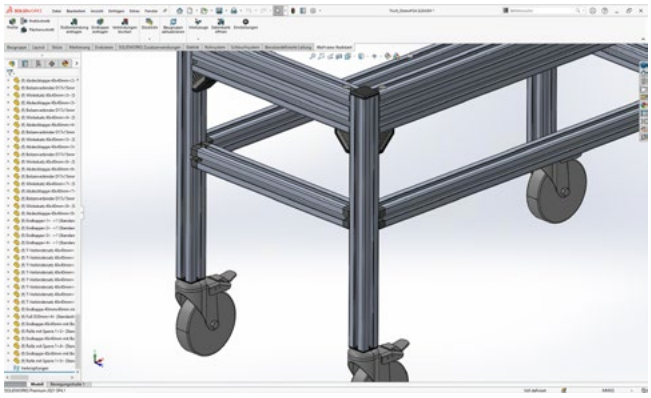
Wheels, feet and end caps

Similar to the standard connections, the AluFrame Assistant for SOLIDWORKS can also be used to place wheels, feet, end caps or similar components.

All that is required, is to select a profile in the 3D model, select a component from the database and decide whether the component should be placed at the beginning or the end of the profile.

The component is then automatically loaded into the 3D model, placed appropriately, any necessary machining is automatically performed on the profiles and the user-defined properties of the component are written.

The intelligent training mechanism of the AluFrame Assistant is also available for wheels, feet, end caps and other components. This gives you, the user, all the options you need to teach it new components. You are also free to choose the manufacturer.

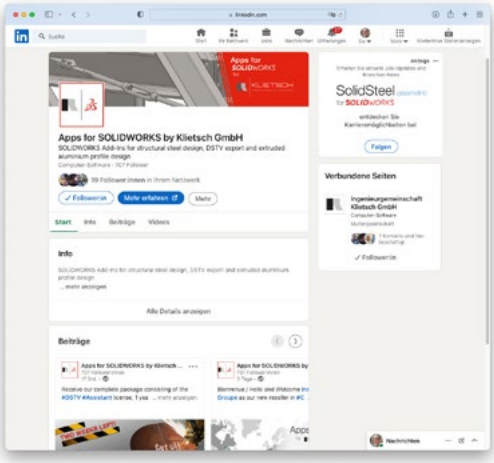


Free trial

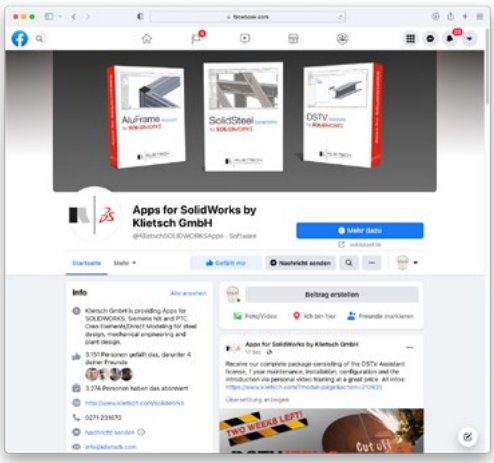
The AluFrame Assistant helps to save time in construction, impresses with simple operation and a lot of comfort. Do not hesitate and contact us. More information and trial version: **please visit www.klietsch.com or scan QR Code.**

Informative and fast

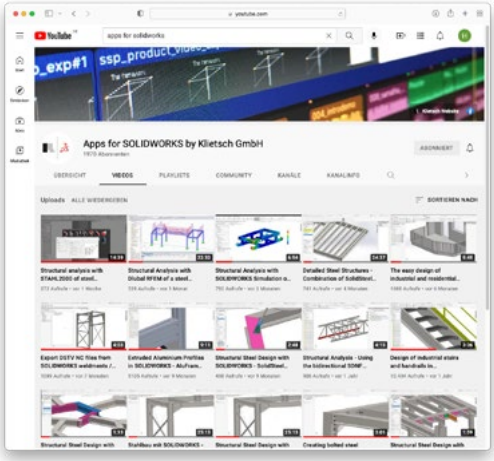
The fastest and most interactive way to stay in touch with us and our software is via our social media channels on LinkedIn and Facebook or via our YouTube channel. There you will be constantly informed about software updates, new features or promotions.



Apps for SOLIDWORKS by Klietsch on LinkedIn:
https://www.linkedin.com/showcase/apps-for-solidworks-by-klietsch-gmbh



Apps for SOLIDWORKS by Klietsch on Facebook:
https://www.facebook.com/KlietschSOLIDWORKSApps



Apps for SOLIDWORKS by Klietsch on YouTube:
https://www.youtube.com/@apps4swx

How to get our software

All of our Apps for SOLIDWORKS are sold both directly by us as the manufacturer and also by SOLIDWORKS resellers (VAR) in order to offer you, the customer, the best possible support.

Currently there are resellers in the countrys marked darker in the world map above, which are:
Argentina, Australia, Austria, Belgium, Bosnia & Herzegovina, Canada, China, Colombia, Croatia, Czech Republic, France, Germany, Greece, Hong Kong, Hungary, Ireland, Italy, Malaysia, Morocco, Netherlands, New Zealand, Poland, Portugal, Republic of Türkiye, Singapore, Slovakia, Slovenia, South Africa, Switzerland, United Kingdom, USA, Vietnam...

Your country is not listed?

Like to see a complete list of resellers?

You want to become a reseller?

Please contact us via solidworks@solidsteel.de, visit our website www.klietsch.com or simply scan the QR code.

Imprint and contact information

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