

Release notes XFrame2D 2027

Release version 2027-07-01 (20-08-2026)

Improvements

General

- In certain cases, a memory leak could occur, causing the PC to crash. This issue has been resolved.
- Memory deallocation in the solver was missing, causing a memory leak. This has been resolved.
- Improved text labels for moment and shear force lines. (#40)

Release version 2027-06-01 (13-08-2026)

New

General

- ****Fire resistance****
It is possible to calculate the fire resistance of your steel structures.

Version 2027-05-01 (25-07-2026)

Improvements

General

- Drawing became slow when there were a large number of beams. This has been greatly improved. (help desk #31)
- Copying beams was no longer working properly. This has been fixed. (help desk #24)
- For beams on an elastic foundation (Shallow foundation), spring supports that can only absorb compression are automatically generated. These springs are iteratively removed or added. This was also done during the calculation of the individual load cases and caused various computational problems, such as endless iterations. Consider, for example, the load case in which primarily horizontal forces are dominant. The number of iterations can increase significantly, and the results are often unusable. Just as in XFEM4U, members on an elastic foundation—as well as tension and compression beams—are now solved iteratively only in the calculation of load combinations. (helpdesk #34)
- The SLS combinations were no longer being generated correctly. This has been resolved.

- If the beam type was not entered (which is, of course, an input error), the program would crash while generating the output. This has been fixed.
- There was an error in the automatic renumbering of nodes (which is done by default) that caused beams to be incorrectly removed. This has been fixed.
- When a new version was announced, the "Download" and "Release Notes" buttons no longer worked in .NET 10. This issue has been resolved. That is, when subsequent new versions are announced.

Version 2027-04-01 (29-06-2026)

Improvements

General

- Due to a programming error, the calculation of tension and compression beams was no longer performed correctly in certain situations. Despite our extensive testing procedures, this error unfortunately went unnoticed and was therefore not detected by us earlier.
As soon as we became aware of this issue, we immediately investigated it and developed a solution. The error has since been resolved. (help desk #27)

Version 2027-03-01 (19-06-2026)

New

General

- **** New solver **** To determine the analysis, a large number of equations must be solved. The stiffness matrix of the structure with the load matrix must be solved a large number of times, especially in the geometrically nonlinear calculation.
A completely new solver has now been implemented that solves that stiffness matrix at lightning speed. This greatly reduces the total calculation time especially for large models.
- Draw a beam perpendicular to another beam: There is also snapping to the point of perpendicularity. The bullet node is drawn marked and the status bar shows that you connect perpendicular to the beam. This allows you to draw beams perpendicular to other beams. (help desk #3207)

Improvements

General

- The program crashed when entering the beams in the table. This has been resolved. (help desk #3101)
- Wind load generator. For perpendicular wind (wind parallel to ridge), the distance $e/4$ for zone F was calculated from the ridge. This should have been measured from the gutter. This has been corrected. (help desk #3114)
- When starting up, a dialog box opens with the option to open a recent model or start with a new model. When a model is saved, an image is now also saved as a png file in the project folder. This image is used in the dialog box mentioned above and also makes searching with Windows Explorer easier.

However, some customers do not like the fact that an image is saved as a png file in the project folder. That is why this is now optional. It is enabled by default, but you can now also disable it. (help desk #3093)

- Design working life is printed in the output. (help desk #3081)
- For example, when you draw a new beam, the dimension line appears, and you can type in that distance right away. That's existing functionality. What's new is that when you start typing, the mouse no longer responds to being moved. (help desk #3149)
- XML export/import: The lateral supports on the top and bottom flanges could not be exported to or imported separately from an XML file. This has been fixed.
- In some cases, a double image (two images overlapping) appeared after the project was saved and a bitmap was exported. This issue has been resolved. (help desk #13)
- Exporting and importing a profile's material type to XML was not working properly. This has been resolved. (helpdesk #16)

Steel

- Lateral stability check: For continuous beams where bending dominates in an intermediate span (situation according to NEN-EN 1993-1-1+C2: 2011/NB:2011 Figure NB.6, Formula NB.7), resulting in the lower flange being subjected primarily to compression, the critical elastic buckling moment (Formula NB.NB.2) was not calculated correctly. The sign of the factor C2 was incorrectly reversed. This error has been corrected. (help desk #3162)
- Lateral stability check: The references to the formulas have been updated in the text.
- Lateral stability check: The C2 value must be extrapolated for loads applied to the upper flange, since C2 assumes a load at the center of gravity of the upper or lower flange. The effect will not be very significant, but it is still worth noting. (help desk #5)
- Lateral stability check: The lateral stability check must be performed for each part of the beam with an unsupported length L_{st} between two end fork conditions, between one end fork condition and one lateral support, or between two lateral supports. This is also provided for in our software. Only the decisive check (to limit the amount of output) is printed. The maximum moment must be determined for each girder section. There was an error in this. A maximum moment could also be found outside the beam part under consideration. As a result, the check / unity check could incorrectly yield a higher result. This has been resolved. (help desk #6)
- The determination of the correction factor k_c according to Section 6.3.2.3, Table 6.6, is now based on the formula by Serna et al., which uses M_{max} and moments at five locations. This method is the most accurate.

Timber

- The check for lateral buckling stability with a tapered profile was incorrect. The calculation used an incorrect cross-section height. This error has been corrected. (helpdesk #19)
- In art. 6.3.2, the value in the unity checks table was not displayed in red when the unity check exceeded 1.00. This has been resolved. (helpdesk #15)

Version 2027-02-01 (19-01-2026)

Improvements

General

- It was not possible to place an order and make a payment online. This issue has been resolved.
- It was possible to display enveloping reaction forces in the graphical representation of load cases. This has been resolved. (help desk #3070)

Patch version 2027-01-02 (16-01-2026)

Improvements

General

- The moment factors (ψ_0 , ψ_1 , and ψ_2) were missing from the load case table. It was also not possible to enter these values. This issue has been resolved. (help desk #3072)

Timber

- The deflection was incorrectly displayed in red in the output, even though it was compliant. (help desk #3015 and #3071)

Version 2027-01-01 (12-01-2026)

new

The world of civil engineering calculations is constantly evolving. New challenges call for more powerful solutions, more flexible software, and a stable foundation for the future. At Struct4U, we have therefore taken a major step forward: with the introduction of **.NET 10, XFrame2D** is fully prepared for the next generation.

Thanks to .NET 10, XFrame2D runs faster, more stably, and more securely than ever before. This not only means that calculations are performed more efficiently, but also that the program effortlessly keeps pace with the latest developments in IT and engineering. Think better performance, a longer platform lifespan, and easier integration with other modern tools.

General

- When starting up, the last model was opened by default. This has now been improved.
A new dialog box now opens with the option to open a recent model or start with a new model. All recent models are now displayed in a table sorted by date and including an image, making searching even easier.
- When a model is saved, an image is now also saved as a png file in the project folder. This image is used in the dialog box mentioned above, but also makes searching with Windows Explorer easier.
- To open the dialog mentioned above box again later, Start has been added to the drop-down menu.
- It is possible to convert an XFrame2D model to an XFEM4U model via XML export/import. (help desk #2864)
- It is now possible to exclude certain beams from a code check. (help desk #2787)

Improvements

General

- Export to MS/Word has been improved. (helpdesk #2866)
- It was not possible to purchase a Network Premium license online. This has been resolved.
- The reaction forces were mixed up in an envelope. This issue has been resolved. (help desk #3020)
- The angle of the profile is now displayed graphically and in the output. (help desk #3016)
- When the deflection was not satisfactory, this was displayed as a warning in dark green in the output. This is now printed as an error in red. (help desk #2688, #3015)
- Texts of stacked q-loads went through the arrow. This has been improved. (help desk #2998)
- The tab order in the load generator dialog has been improved. (help desk #3064)
- When displaying gridlines graphically, the z-levels were not printed with the correct sign. This has been resolved. (help desk #2600)
- Double node loads in the same node were drawn on top of each other, making them difficult to see. This has been improved. (helpdesk #2889)
- Graphic images in the output were not scaled correctly. This has been improved. (help desk #2653)

Steel

- Moment connection: When the number of bolt rows was adjusted, all bolt distances were incorrectly reinitiated. This has been resolved. Now, when a bolt row is added, the other bolt distances remain intact. (helpdesk #2606)
- Lateral stability check: It was not possible to create a beam group consisting of beams with different profiles. This has now been made

possible, so that a good length between the forks (L_g) can be determined. (help desk #3033)

- Moment connection: The pressure stress in the column is now maximized to the yield strength. (help desk #2811)

Timber

- The additional deflection was not calculated with the total final deflection including creep. This has been corrected. (helpdesk #3047)

Patch version 2026-02-03 (04-09-2025)

Improvements

Concrete

- The automatic opening of one of the examples during the first startup did not work properly. This has been improved.

Release patch version 2026-02-02 (26-08-2025)

Improvements

General

- ****Microsoft Word**** The program crashed when exporting to Word. This has been resolved. (help desk #2836).
- Program crashes after XML import. This has been resolved. (help desk #2839)

Release version 2026-02-01 (24-08-2025)

new

General

- ****Microsoft Word**** It is now possible to generate the output as a Microsoft Word document (.docx)*.
All (Eurocode) formulas in our transparent output are converted to equations in Microsoft Word.
- When a beam is copied, all beam loads on that beam are now also copied. (help desk #2813)

Improvements

Steel

- The determination of lateral distances of a beam group where node numbers were used did not go well. That problem has been fixed. (help desk #2788)

Version 2026-01-01 (17-06-2025)

new

General

- ****Keyboard shortcuts**** It is now also possible to work with keyboard shortcuts. Almost all commands can now be called with 2 letters just like in Autocad - Revit.
The two letters, often an abbreviation of the command (such as BE for "Add **B**eam" or LQ for "Add beam **L**oad **Q**"), make it **logical and quick to learn**, even for beginners. We've already created those shortcuts for you. And you can use the menu or the letters KS ("**K**eyboard **S**hortcuts") to show that list.
With keyboard shortcuts, you don't have to navigate to menus or buttons with the mouse every time. This saves several seconds each time, which adds up to **hours of time saved** in a workday.
We are quite convinced that you will like this new functionality.
- When you select multiple beams you can enter the beam load on those beams at once. (helpdesk #2632)
- When you select multiple nodes you can enter at once the node load on those nodes. (helpdesk #2632)
- Table combinations: In each cell you enter 2 numbers in the syntax "combination factor x load factor". You can now also enter one value and that is the load factor. In that case, the combination factor is set to 1.00.

improvements

General

- Module built-up sections: The self-weight of a built-up sections with different moduli of elasticity was not properly determined. This problem has been resolved. (helpdesk #2650)
- When a recovery file is found, a message box appears. This message box was behind the startup splash screen. This problem has been fixed. It is now always visible.
- Profiles dialog box: A rotated profile on the right side fell partially out of view. This has been corrected. (help desk #2778)
- When NEN8700 was used, the consequence class incorrectly changed from CC1 to CC3 when the project was re-opened. That problem has been fixed. (help desk #1592 and #2782)

Steel

- Lateral supports at bottom flange: The setting "as top flange" was on by default. This has been changed. It is now off by default. This appears to be more practical and is also safer. (helpdesk #2669)

Patch version 2025-05-02 (27-11-2024)

improvements

General

- Load generator: Wind load at wind perpendicular was not properly generated in specific cases. This has been resolved. (help desk #2574)

Version 2025-05-01 (25-11-2024)

new

General

- Snapping has been extended. Now it also snaps to intersections of beams and grid lines. (help desk #2517)
- ****Tip**** To repeat the last drawing command just press the **Enter** key or the **spacebar** like in AutoCad, or **F4** key like MS Word. This is super convenient because you often want to quickly repeat the drawing of, say, a new beam. The big advantage is that you don't have to move your mouse to the function in the ribbon menu so you can input your model even easier and faster. Try it. You will like it. (help desk #2367)
- Load generator: It is now possible to specify the basic wind velocity V_{b0} by yourself as well. (help desk #2403)
- The status bar shows the own weight and paint area. You can also have this determined from a selection of beams.

improvements

General

- Load generator: When the height above ground level was entered in [mm] instead of [m], no wind load was generated. Input control has been improved. (helpdesk #1113, #1773)
- The graphical representation of soil pressure was found to be incorrect at widths other than 1m. This problem has been resolved. (helpdesk #2532)

Steel

- Beam-column moment connection: Bolt spacing was not saved in specific situations. This problem has been resolved. (helpdesk #2539)
- Beam-column moment connection: In those cases where a combination with as type "SLS Quasi-permanent" occurred, the connections were not displayed on screen nor in the output. This issue has been resolved. (helpdesk #2537)

Patch version 2025-04-02 (09-10-2024)

improvements

General

- Enveloping reaction forces were erroneously no longer drawn in the output. This problem has been fixed. (helpdesk #2516)

Version 2025-04-01 (03-10-2024)

New

General

- **Company logo:** It is now possible to include your company logo in header. If you choose this then the page numbering is moved to the footer to save space. The time is no longer printed. (help desk #2281)
- For beams on elastic foundation (Shallow foundation), it is now possible to graphically display the soil pressure. And just like the bending moment diagram, it is also possible to show the envelope soil pressure as well. (helpdesk #1917, #1965, #2078, #2365)
- In the ribbon menu, the page group File has been removed, making the menu more clear.
The functions print, pdf output and print preview are added to the Quick Access Toolbar.
- The axial force diagram can now also be drawn in the spy.
- The buckling length around the y-axis can now optionally be entered as well. Just like in XFEM4U (help desk #2373 and #2393)

improvements

General

- In specific cases, the removal of a beam incorrectly also removed a node load. This has been resolved. (help desk #2334)
- When using the new functions *Determine intersections of beams*, *Extend beams* and *Divide beam*, this was not seen as a change and, as a result, was not automatically recalculated. This has been fixed.
- When mirroring beams, the lateral supports are placed on the wrong side. This is solved by reversing the beam orientation. (helpdesk #2361)

Version 2025-03-02 (09-06-2024)

New

General

- When selecting beams you can right click → ✂ **Determine intersection points of beams**, determine the intersection point and a new node is generated.
- When selecting beams you can right-click → ✂ **Extend beams**, have the intersection point determined and the beams extended.
- When selecting beams you can right-click → ✂ **Divide beam**, easily divide your selected beams into equal beam parts.

improvements

General

- When beams occurred where one of the nodes no longer existed, the program crashed. This has been fixed. (helpdesk #2262)
- The automatic collapsing and unfolding of menu groups (Geometry, Loads, etc.) in the menu depending on the tab that is active no longer worked. This has been fixed. (helpdesk #2295).
- From Excel, parameterized input can now be created very easily and quickly. From Excel an XML file is created which is automatically detected by **XFrame2D**. This makes the input immediately visible in **XFrame2D**. A number of components have now been added to the XML file, further improving the linkage to Excel and Python.

Version 2025-02-01 (28-04-2024)

improvements

General

- It occurred that the program would abort on exit when the "Results" tab was displayed. This has been resolved. (helpdesk #2241)
- Drawing profile cross section: The rounding radii (r1 and r2) with r=0 were not drawn and therefore could not be adjusted. This has been improved. (helpdesk #2251)

Timber

- The determination of the decisive timber grade specifically for a build-up sections was found to be flawed. This has been resolved. (help desk #2258).

Patch version 2025-01-02 (21-03-2024)

improvements

General

- It was not possible to enter a concrete profile. This has been resolved. (helpdesk #2221)

Version 2025-01-01 (28-02-2024)

New

General

- **** User defined materials **** It is now possible to enter own materials (steel, concrete, rebar and timber). How? Very simple. Copy one of the standard materials. Give it its own name and adjust its properties. It is not possible to modify the default Eurocode materials but you can define and add your own material. Then you can use this material in other programs as well. The materials are saved in an XML file so you can put that on the network and share the materials database with your colleagues.
- In the transparent calculation output of steel and timber, the decisive check is shown in **blue** for $u_c < 1.00$ (or **red** for $u_c > 1.00$). This allows you to quickly see which check is decisive so that you can more easily optimize your structural design.
- From Excel, parameterized input can now be created very easily and quickly. From Excel an XML file is created which is automatically detected by **XFrame2D**. This makes the input immediately visible in **XFrame2D**.
- Tables load cases and load combinations: It is now possible to select and delete multiple cases and combinations, respectively. (helpdesk #2038)
- When you change the beam load type to roof then it is taken over right away with the deflection. (helpdesk #2051)
- Of course, a software error should never occur. Let there be no misunderstanding about that. If an unexpected software error does occur, an error report is now automatically generated. And you can send this report very easily (with one click on the send button) including your input and preferably your explanation to us so that our Struct4U support team can investigate it. That way you help us make the software even better.

Steel

- Our profile database has expanded the old German steel profiles **N.P.** and **B.**

Timber

- Load case type "Live load" has been changed to "Live load (medium-term)." And a new type "Live load (short-term)" has been added. This allows you to test your timber structure in both medium-term and also short-term load-duration class. (helpdesk #2051)

improvements

General

- Network license / Floating license: When the number of concurrent users exceeds the number of network licenses an error message now appears including a list of current users (CPUId, UserName and UserDomainName). (helpdesk #2014)
- The calculation of spring connections was not going well when there were nodes on the spring beam. This has been fixed. (helpdesk #2046)
- In specific cases it could happen that uc values in a symmetrical construction did not give the same symmetrical image. This has been resolved. (helpdesk #2045)
- Load generator: When additional loads were manually added to the load cases that were automatically generated by the load generator, and the beam properties of one of the beams was modified, those loads disappeared again because the load generator was automatically restarted by mistake. This has been resolved. (helpdesk #2043)
- When you adjusted the beam load graphically by dragging, no undo step was saved. This has been fixed. (helpdesk #2039)
- For example, when you drew a new beam, the view tab switched to Loads. This annoying error has been fixed. (helpdesk #2020)
- Setting the number of springs for a beam on an elastic soil has been improved. (helpdesk #1964)
- The beam type (loads) was not displayed graphically correctly in all cases. Also when the profile was rotated, it was not displayed. This has been resolved. (helpdesk #2000)
- When you drew a beam and clicked Cancel in the Beam dialog box, adding beams was not canceled. This issue has been resolved. (helpdesk #2081)
- Blank sheets appeared in the output for relatively small constructions. This problem has been solved. (helpdesk #2033)
- When you drew a beam on an elastic soil from right to left, the springs were generated outside the beam. This problem has been solved. (helpdesk #2153)
- The u.c. symbol is now drawn at 0.45 x beam length instead of the center. This makes the symbols less likely to be drawn on top of each other (helpdesk #2155)
- Beam dialog box: The settings related to deflection were not remembered when entering a new beam. This problem has been solved. (helpdesk #1818)
- If the license is not found immediately after startup, no calculation can be made. This is, of course, confusing. Now a message appears saying that the license has not yet been found.
- Output selection: total page count can now be suppressed. (helpdesk #2069)

Steel

- Built-up section: The self-weight was not calculated correctly in all situations. This has been resolved. (helpdesk #2007 and #2064)
- The steel check, of course, implicitly includes cross-sectional classification. (1,2,3 to 4) Section class 4 is converted to section class 3 with a reduction of the yield strength. The output for those situations has been improved. (help desk #2142)
- In the output, in the table of EN1993 checks, the determined cross-sectional class was not printed correctly. This has been corrected.

Patch version 2024-02-02 (09-10-2023)

improvements

General

- In the latest version 2024.02 (as a new functionality) in the beam dialog it has been made possible to change the start and end node. However, an annoying error occurred here, unfortunately. This has been fixed. (helpdesk #1977)
- When mirroring, coinciding nodes were not removed. That has been improved. (helpdesk 1050)
- In the built-up section dialog box, the angle radio button was not displaying correctly. This has been fixed. (helpdesk #1963)
- In the Geometry table, Gridlines/Levels tab was visible. This has been fixed. (helpdesk #1966)

version 2024-02 (23-08-2023)

new

General

- It is now possible to drag all **beam loads graphically** (using the mouse) and in the case of a q-load also to extend them. This supports snapping to points every 10 mm but also to other beam loads. This works very intuitively. You can now enter or adjust beam loads even easier and faster.
- In the beam load dialog box, you can now also change the distances (a, b and L) by clicking on the dimension lines. This also works very intuitively.
- The user interface has a new **look and feel**. Modern and functional. We are now using the new "skin" "The Bezier" which renders its elements with vectors. This means resolution independence and sharp rendering on Hi-DPI displays. This is not only nice to work with but also less tiring. It is also possible to set the colours yourself.

The menu will contain more items. To keep it clear for the user, the menu groups *Geometry* and *Loads* are automatically minimized depending on which tab is active. All to make the user-friendliness of the software even better.

- Gridlines and levels are now entered in a separate dialog as in **XFEM4U**. The grids are drawn with a bullet including dimension lines. You can enter the labels of the levels yourself or have them drawn automatically.
- XML export/import: It is now possible to export the model including load cases (including beam and node loads) and combinations to an XML file as well as import from an XML file.
- With the scroll wheel pressed, you can now pan.
- Spy: In the dialog you could already graphically retrieve the values (f,M or D) at a certain x-distance. What is new is that you can also enter the x-distance numerically and thus exactly. (helpdesk #1933)
- Dialog of beams: It is now possible to change the start node and end node as in **XFEM4U**.
- With the new menu option "Multiple Copies" beams can easily be copied multiple times as in **XFEM4U**.
- With the new menu option "Mirror" beams can easily be mirrored or flipped as in **XFEM4U**.

improvements

General

- When drawing a new beam, multiple beams were automatically generated when intermediate nodes were found. This did not prove convenient and has been removed for that reason. It is now identical to the method used in **XFEM4U**.
- Copying and moving now works almost identically as in **XFEM4U**. It works more intuitively. (helpdesk #1644 and #1811)
- Also, measuring distances now works almost identically as in **XFEM4U**.
- Tension only beams are drawn in red on the Geometry tab.
- In the Preference dialog box, the User defined profiles tab has been removed.
- When the node of a beam on elastic soil was moved the springs were not re-generated. This has been resolved.
- Dialog box beams: For a tapered beams, additional check that the cross-sectional shape at the beginning and end are the same. (helpdesk #1921)
- Zooming with the mouse wheel has been improved. You also no longer need to press the control key.

version 2024-01 (21-06-2023)

new

General

- Load generator: It is now possible to enter the base wind velocity V_{b0} for other countries so that wind loads can be generated.

Steel

- Our profile database has expanded the old steel profiles DIN, DIE, DIL and DIR.

improvements

General

- When the display options was turned off it could not be opened. An error message "The dock manager is not yet initialized" appeared. This has been resolved. (helpdesk #1710/#1711)
- Load generator: The snow load shape coefficients were not written out correctly in the output. This has been resolved. (helpdesk #1715)
- To make communication via our web API with our License Server even more secure, TLS 1.2 (Transport Layer Security) is now being used. (helpdesk #1737)
- For a large number of combinations, the table combinations was not printing correctly in all cases. This has been resolved. (helpdesk #1756)

Steel

- In specific cases, reduction factor for lateral-torsional buckling χ_{LT} could become greater than 1.00. This is obviously not possible and has been adjusted. (helpdesk #1745)
- Beam-column moment connection - butt joint: The stiffness of the end plate was charged 1x instead of 2x. This has been resolved. (helpdesk #1666)
- Beam-column moment connection - butt joint: In the stiffness calculation the bolt length was not correctly determined This is solved. (helpdesk #1667)
- Beam-column moment connection - butt joint: In the stiffness calculation a stiffness modification factor of 2 instead of 3 was used.
- The program broke down when beam numbers occurred in the beam group that no longer existed. It has been resolved. (helpdesk #1791)
- Beam-column moment connection - The program broke down when creating an output when moment connections were entered but not used and thus not calculated. An attempt was made to draw the connections and that caused the error. It has been fixed. (helpdesk #1810)

Timber

- In the stress check, the beam height was not correctly determined in all cases. This problem has been resolved. (helpdesk #1674)
- Columns were also always tested according to Art. 6.2.4. This turned out to be incorrect. Formulas (6.19) and (6.20) should be used only when $\lambda_{rel,z} < 0.3$ and $\lambda_{rel,y} < 0.3$. This has been fixed. (helpdesk #1698)
- No out of plane buckling length could be specified that was greater than the beam length. This problem has been resolved. (helpdesk #1771)

Patch version 2023-01-02 (17-11-2022)

new

Steel

- Moment connection: In a butt connection, the factor k_{10} was erroneously not taken into account in the calculation of bending stiffness. (See helpdesk #1606) The calculation of moment-curvature diagram used to determine the bending stiffness was not going well anyway, causing the analysis to crash. This has been resolved. (helpdesk #1663)

version 2023-01-01 (03-11-2022)

new

General

- On the geometry tab, profiles can now be displayed. There is a new display option "Profiles" (helpdesk #1455)

Steel

- Lateral buckling stability: I-shaped sections were tested according to formula (6.56) of article 6.3.2.2. This is a conservative and therefore safe calculation. However, for I-sections it is also possible to use formulae (6.57) and (6.58) of article 6.3.2.3. Generally this leads to lower / more favourable unity checks. And we have now provided that. This can further optimize the design of your steel structure and save steel kilos. (helpdesk #1603)

improvements

General

- When changing the beam load widths for each beam, it appeared as if the widths were being adjusted at all beams. This has been fixed. (helpdesk #1548)
- Graphical representation beam point load was completely outside the beam. This has been fixed. (helpdesk #1431)

Steel

- For symmetrical structures with tapered profiles and symmetrical loads, it could happen that the unity check was not symmetrical. This has been resolved. (helpdesk #1576 and #1582)
- Tension beams were incorrectly tested for buckling at normal stresses less than -0.1 N/mm^2 . This has been resolved. (helpdesk #1514 and #1573)
- When the analysis was set to geometrically linear, the stiffnesses of the joints are not taken into account and the joint is also not tested. This has been modified. When steel or timber testing is selected, a geometrically nonlinear calculation is always started. (helpdesk #1498, #1547 and #1598)

- Moment connection: When the profile changes, the number of bolt rows is also changed. This was not desirable and has been adjusted. (helpdesk #1530)
- Beam-beam moment connection: For a butt joint, the factor k_{10} was incorrectly not considered in the calculation of the bending stiffness. This has been resolved. (helpdesk #1606)

version 3.12.01

new

General

- ***Tapered profiles / non-prismatic beams***: It is now possible to enter a tapered profile. You can do this very easily by choosing 2 different profile cross-sections at the beginning and end of the beam.
- Wind load generator: It is possible to enter the structural factor $C_s C_d$. It is also possible to take into account the correlation according to Art. 7.2.2(3). (helpdesk #779, #1253 and #1259)
- The number of last used input files has been increased from 10 to 25.

improvements

General

- Index out of range when calculating. This has been fixed. (helpdesk #1269)
- When adding a new beam and turning off the beam's own weight, this setting was not adopted. This problem has been fixed. (helpdesk #1272)
- The texts of reaction forces were drawn mixed in some cases. This problem has been fixed. (helpdesk #1271)
- When you remove a beam then that beam must also be removed from all beam groups. This was missing and has now been added. (helpdesk #1306)
- The paragraph numbering for the detailed calculation output per beam was missing. This has been added. (help desk #1414)
- In the output, in the table load combinations, the load factors were partially missing in case of a large number of load cases. This problem has been solved. (helpdesk #1158 and #1439)
- The placement of an internal hinge has been improved. The beam is automatically split including all beam loads. (helpdesk #1454)
- When the last file was automatically opened, the imperfections setting was incorrectly re-enabled. This issue has been resolved. (helpdesk #1473)
- Every time you switch between imperfections on - off then the type of load case changes from permanent excl own weight to permanent incl own weight. This problem has been fixed. (helpdesk #1474)

- Load Generator: The snow load was not determined correctly in other reference periods. This has been resolved. (helpdesk #1484)
- Test deflection: The total deflection and the additional deflection are tested. Only one combination was printed. Namely, the one that was decisive for total deflection or additional deflection. This has been expanded. Now the decisive combinations for total deflection and additional deflection are printed separately. (helpdesk #1472)

Timber

- The testing of Beams subjected to either compression and bending according to art. 6.3.3 did not proceed properly in all cases. This has been resolved. (helpdesk #1308)
- There was a textual error in formula (6.19). This has been solved.

Steel

- Built up section: In the graphical view, profiles that partially overlap are shown in red. (helpdesk #1430)
- Beam-column moment connection: When the angle of the truss leg is adjusted, the bolt distances are automatically corrected (help #1433)

version 3.11.01

new

General

- In geometry, the dimension lines have been added. With the new display option "Dimension lines" you can make the dimension lines visible or not. (helpdesk #1043)
- In the case of an enveloping moment line, in a number of cases, the moments at the supports were not properly represented. The sign turned around. This has been solved. This was a drawing error. (helpdesk #1062)
- At the beam data "Dead weight automatically generated" has been added. It is now possible to set per beam whether or not the own weight should be generated. (helpdesk #1082)

improvements

General

- Beams on an elastic soil: the default value of the modulus of sub-grade reaction was reduced from 40,000 kN/m³ to 10,000 kN/m³. (helpdesk #1044)
- A support with a local coordinate system specifically at a 90 degree angle was not processed correctly. This has been resolved. (helpdesk #1046)
- Print Preview: The output screen could be resized with the Minimize menu option. That menu option has been removed. (helpdesk #1080)

- Load cases. When opening an existing project, the "Automatically generate combinations" setting was not filled in in the dialog. This has been resolved. (helpdesk #1091)
- Beam loads: In the combo box of evenly distributed load, the capital letter Q is shown while in the drawing there is a lower case letter. This has been fixed. (helpdesk #1112)
- When you choose an output without paragraph numbering and table of contents, the usability limit states still show the chapter number while the description is missing. This has been fixed. (helpdesk #1238)

Steel

- When the yield strength is reduced so that cross-sectional class 3 can be calculated, the test is printed according to Table 5.2. You can see how the reduced yield strength was determined. (helpdesk #1124)
- Beam-column moment connection: Adjusting vertical bolt spacing has been improved. (helpdesk #740)

version 3.10.02

new

General

- In the output of the table beam forces you miss the beam forces at intermediate nodes. New is that these are now also printed. (helpdesk #974)
- In the display options you can now set whether the unity check should be displayed incl. or excl. deflection. (helpdesk #1018)
- **Load width per beam:** In the load generator, the load width of the truss is specified. Now you can "override" this load width per beam by specifying a different load width at the beginning and end of the beam. (helpdesk #1027)
- The way you enter beams or plate edges and also how you move beams or plates has been extended with an extra option. While you are drawing, you can **press the space bar** to display a new dialog where you can enter relative or absolute coordinates directly. This works intuitively so you can enter your construction even faster and easier.

improvements

General

- The calculation results are not printed. The cause was that when creating the output, the load generator automatically reversed the beam orientation of beams when necessary. This has been resolved. (helpdesk #993)

- Load generator: The sign of the default values of the over- and underpressure coefficients in the case of no openings turned out to be incorrect. This error was minimal, by the way. This has been resolved. (helpdesk #1017)

Steel

- When the entered pre-camber was larger than the deflection the test did not proceed correctly. This has been fixed and a warning will be printed in that case. (helpdesk #991)
- When a beam was copied the beam group was not created properly. This has been fixed. (#995)
- Equilateral and unequal angle samples in the profile database have been expanded. (helpdesk #1007)

Timber

- The table deflection has been dropped because the deflection is done per beam. The results are visible in the table for the unity checks. (helpdesk #1029)

version 3.09.01

improvements

General

- If the rotation angle was entered in the profile data for built-up profiles, this was not taken into account. This has been solved. (helpdesk #917)
- When logging in as an existing customer with the Struct4uKey this was not visible in the status bar. This has been solved. (helpdesk #922)
- In the output, not all tables were displayed correctly. This has been fixed. (helpdesk #927)
- Load generator: The combination "SLS Permanent" was not created. That combination is necessary to test the additional deflection. This has been solved. (helpdesk #928)
- When the account credit is too low, the administrator login screen was shown. The program broke down when you tried to login. This has been fixed. (helpdesk #939)

Steel

- Beam-column moment connection: When a shear force was entered, the bending in the head plate was incorrectly calculated using the column flange thickness. This has been fixed. (helpdesk #948)
- In specific cases, the MyEd in the stability control according to 6.3.3 was not determined correctly. This has been resolved. (helpdesk #954)
- Based on NEN-EN 1993-1-1+C2:2011/NB:2011 formula (NB.74) or (NB.75), it is determined what the pressed flange is (top or bottom flange) and which lateral supports should be taken into account. It is assumed here that the clamping moments are opposite in sign to the field

moment. When the moments are equal in sign, the pressed flange was not determined correctly. This has been resolved. (helpdesk #953)

- For L-profiles the section classes 1 and 2 are converted to section class 3. In other words, they are tested elastically and are no longer plastic. (helpdesk #918)
- A textual error in the output: Formula (6.42) incorrectly contained the factor "a". This has been removed. (helpdesk #967).
- In specific cases, for beam groups, the moments for the determination of lateral stability were not properly determined. This has been resolved. (helpdesk 971)

version 3.08.01

new

General

- **Deflection:** Beams are now also tested for additional deflection and final deflection. It is also possible to specify a pre-camber for a steel beam.
- At start-up, the last open project is opened.
- Output selection: In the output selection dialog, the default text "Page" (in the page header) can be changed to, for example, "Attachment A".

improvements

General

- Load generator: The wind load was not properly generated in specific cases (flat roof in combination with a pitched roof). This has been solved. (helpdesk #894)

version 3.07.02

new

General

- Profiles that have not been used will no longer be printed.
- Zero loads are automatically removed.
- In the output selection dialog a new "Save settings" button has been added. This allows you to save the output selection used for each new input.

improvements

General

- When the table profile data was output, the column for I_y proved to be too narrow, so that one digit was omitted. This has been solved. (helpdesk #761)
- It was possible to suppress the graphical output of the profiles. This has changed. Profiles are now always displayed graphically so that the steel grade or strength class is always in the output. (helpdesk #788)
- In the graphical view, all internal forces are displayed with the sign character.
- In the case of a new project, the project data and the calculation settings are requested.
- When you switch combinations in the combo box, while the combo box was unfolded, the description of the combination on screen was not updated. This has been solved. (helpdesk #880)

- In the output for the graphical representation of the enveloping displacements there was a wrong text. This has been solved. (helpdesk #879)

Steel

- Connection: Connections in cantilevers were classified according to Figure 5.4. This was not correct. The boundary lines derailed in the graphical representation. This has been resolved. Connections in cantilevers are no longer classified because the classification is not relevant either. (helpdesk #744)
- In specific cases, the program could crash when generating the output of connections. This problem has been solved. (helpdesk #803)
- Beam-column moment connection: A hole clearance of 2 mm was taken into account. This was not correct. The hole clearance depends on the bolt diameter according to EN 10190-2. This has been resolved. (helpdesk #833)
- In specific cases, nodes could refer to connections that no longer existed. As a result, the output crashed. This has been resolved. (helpdesk #861)
- An error has been found in the calculation of k_{zz} from table B.1. This has been solved. (helpdesk #890)

version 3.07.01

new

General

- The gravitational acceleration g can be specified. The standard value is 9.81 m/s^2 . (To generate the own weight bar loads, 10 m/s^2 was taken into account. This has now been improved).

Steel

- Beam-column moment connection: The test of the shear force was still missing. This check is added. The shear force is carried by the bolts in the lower rows of bolts, which are barely loaded under tension. (helpdesk #702)

improvements

General

- The outer and inner radius of a box section could not be adjusted. This has been solved. (helpdesk 2019111501)
- Rounding radius of a T-shape could not be adjusted. This has been solved. (helpdesk #717)
- When on the Results tab the Project data were changed, the calculation results were erroneously deleted. This has been resolved. (helpdesk #709)

Steel

- In the case of tension beams, small normal pressure forces may occur and a compression and bending test may have been carried out. This has been resolved. (helpdesk 2019111503)
- Beam-column moment connection: The check of the connecting beam turned out to be too good, so the end plate was initiated with the default dimensions. (helpdesk 2019112502)
- Steel/Lateral buckling stability: Based on the sign of the moment in the middle of the beam (group) it was determined which lateral supports should be taken into account, the lateral supports at the top flange or at the bottom flange. This has been further refined by using the test rules according to NEN-EN 1993-1-1+C2:2011/NB:2011 figure NB.6 resp. figure NB.7. (formulas NB.74 and NB.75) (helpdesk 2019111201)

