

Release notes XBeam2D 2027

Patch version 2027-05-02 (19-08-2026)

Improvements

General

- 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 2027-05-01 (13-08-2026)

New

General

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

Version 2027-04-01 (25-07-2026)

Improvements

General

- The chessboard load has already been generated, but it can be regenerated using the new menu option, "Load Generator".
- 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 2027-03-02 (07-07-2026)

Improvements

General

- When calculating chessboard load, the momentaneous factor in the SLS combinations was incorrectly set to 1.00. This issue has been resolved. (helpdesk #30)

Concrete

- The calculation of VR_{dc} in the output table was incorrect for plate reinforcement (diameter-ctc). The surface area of the longitudinal reinforcement was 0 mm². As a result, the VR_{dc} was underestimated.

This issue has been resolved. The calculation and design were otherwise correct. This error was therefore a cosmetic output error. (helpdesk #23)

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.
- **** Chessboard loading **** It is possible to perform calculations for live loads using a chessboard loading pattern. The various loading scenarios (load cases and combinations) are generated automatically. (helpdesk #3160, #829, #2748)

Improvements

General

- 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)
- 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. (help desk #25)
- 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)
- When multiple beams were inserted (via copy/paste), the beam loads might have been placed multiple times because the system incorrectly used the "selected beams" for this purpose. This has been resolved. (helpdesk #18)

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 / inity 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.

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.

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, XBeam2D** is fully prepared for the next generation.

Thanks to .NET 10, XBeam2D 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.

Concrete

- The Dutch national annex NEN-EN 1992-1-1+C2:2011+A1:2015/NB+A1:2020+A2:2025 is provided. (helpdesk #2866)
- The stirrups distances that can be selected in the design are standard 300, 250, 200, 175, 150, 125, 100, 75, and 50. What is new is that these values can now be set by the user. (helpdesk #3007)

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.
- When ground pressures could not be calculated, the program crashed when displaying them. This issue has been resolved. (helpdesk #2890)
- 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)

Concrete

- The shear force coverage line for a plate over multiple support points was not determined correctly. This has been resolved. (help desk #2853)
- The design of stirrups has been improved. (help desk #2862)
- The coverage entered at the bottom was not printed correctly. This has been resolved. (help desk #3053)
- In the case of a plate, the number of stirrup sections and the angle of compression strut are no longer printed. (helpdesk #3053)
- The requirement for total and additional deflection was fixed but can now be set in the beam data. (help desk #3026)

Timber

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

Patch version 2026-02-02 (24-06-2025)

Improvements

Concrete

- With a spring-supported beam, the program crashed during the concrete deflection calculation. That problem has been resolved. Concrete deflections cannot be calculated for beams on a bedding. Spring-supported beams are also excluded. (helpdesk #2842)

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)

Release patch version 2026-01-02 (24-06-2025)

Improvements

Steel

- It did not succeed in placing point loads graphically, for example. That problem has been fixed. (helpdesk #2792)

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. (help desk #1518, #1868, #2565 and #2575)
- 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

- The graphical representation of soil pressure was found to be incorrect at widths other than 1m. This problem has been resolved. (helpdesk #2532)
- 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)

Concrete

- The longitudinal reinforcement apparently could not be entered as "4X16" (capitalized) but could be entered as "4x16". This has been corrected. Both syntax are now accepted. (help desk #2783)

Version 2025-04-01 (25-11-2024)

new

General

- ****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 load. 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)
- The status bar shows the own weight and paint area. You can also have this determined from a selection of beams.

Concrete

- It is now possible to enter the old reinforcement diameters Ø5.5, Ø7.5, Ø8.5, Ø9.5, Ø10.5, Ø11.5, Ø11, Ø13, Ø15, Ø19 and Ø28. This is of course useful if you want to test an existing construction. (help desk #2541).
- It is now also possible to enter covers smaller than the minimum required cover. A checkbox "Diviant cover" has been added. By default, it is off. When the cover does not comply, a warning is now given instead of an error message. (helpdesk #2541)

improvements

General

- The graphical representation of soil pressure was found to be incorrect at widths other than 1m. This problem has been resolved. (helpdesk #2532)

Concrete

- The default side reinforcement was changed from 1Ø10 to 1Ø8.

Patch version 2025-03-02 (09-10-2024)

improvements

General

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

Version 2025-03-01 (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.

improvements

Concrete

- In the main reinforcement dialog box, there was incorrect text for the transverse reinforcement. This has been corrected.(helpdesk #2335).
- It is now possible to enter side reinforcement. That side reinforcement is also included in the calculation of the resistant bending moment. The user can set whether that side reinforcement should be included in the calculation or not. By default, it is off (help desk #2365)

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).

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.
- 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.**

Concrete

- Concrete strength classes C28/35 and C55/67 have been added.
- The length of the supplement reinforcement is now also displayed graphically. (helpdesk #1725)

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

- In the built-up section dialog box, the angle radio button was not displaying correctly. This has been fixed. (helpdesk #1963)
- In the Geometry tables, Gridlines/Levels tab was visible. This has been fixed. (helpdesk #1966)
- 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)
- When you adjusted the beam load graphically by dragging, no undo step was saved. This has been fixed. (helpdesk #2039)
- Setting the number of springs for a beam on an elastic soil has been improved. (helpdesk #1964)
- When a new entry did not use the wizard and draw a beam, the program broke down. Also when a new entry was made, the wizard was not displayed immediately. This has been fixed. (helpdesk #1994)
- 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)

Concrete

- Longitudinal reinforcement: the diameters Ø5, Ø7, Ø9 and Ø14 have been added. (helpdesk #2017)
- Cover: side cover was tested against the largest minimum cover top side and bottom side. This has been modified. Now the side cover is tested against the smallest minimum cover. (helpdesk #2053)

version 2024-02 (24-08-2023)

new

General

- It is now possible to drag all **beam loads graphically** (with 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 colors 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 are now entered in a separate dialog as in **XFEM4U**. The grids are drawn with a bullet including dimension lines.
- 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.
- 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)

Concrete

- Concrete coverage: concrete coverage is automatically increased when it does not meet the minimum coverage. (helpdesk #1614)

improvements

General

- Dialog box beams: For a tapered beams, additional check that the cross-sectional shape at the beginning and end are the same. (helpdesk #1921)

Concrete

- Art. 7.3.2 Minimum reinforcement areas were incorrectly considered in the reinforcement design. In fact, the reinforcement is designed based on Art. 7.3.3 Crack control without direct calculation. This has been modified. (helpdesk #1924)
- Concrete coverage: For an uncontrolled surface, the minimum coverage was not properly determined. This has been improved. (helpdesk #1614)

version 2024-01 (21-06-2023)

new

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)
- 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)
- When a uniformly distributed load was incorrectly entered as type q-px, this load was not included. This issue has been resolved. (helpdesk #1685)

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)

Timber

- In the stress check, the beam height was not correctly determined in all cases. This problem has been resolved. (helpdesk #1674)

Concrete

- Users rightly found it inconvenient and unintuitive that they had to redesign the reinforcement of the beam themselves each time they changed the geometry and/or load. This has now been greatly improved. When the beam has to be recalculated because the geometry and/or load changes, the reinforcement is now automatically redesigned as well. Is the reinforcement manually modified by the user (longitudinal reinforcement and/or braces) then there is "prescribed reinforcement" and of course no reinforcement design is done. (helpdesk #1570)

version 2023-01 (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

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)
- 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)

version 3.07.01

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.

improvements

General

- In the case of a concrete beam, the profile name could not be changed. This has been improved. (helpdesk #1399)
- The paragraph numbering for the detailed calculation output per beam was missing. This has been added. (help desk #1414)
- The placement of an internal hinge has been improved. The beam is automatically split including all beam loads. (helpdesk #1454)
- 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)

Steel

- Built up section: In the graphical view, profiles that partially overlap are shown in red. (helpdesk #1430)

Concrete

- In the determination of the minimum concrete cover, a fixed life of 50 years was incorrectly used and not the set value. This has been fixed. (helpdesk #1466)

version 3.06.02

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)
- When generating a new input, the profile data was not initialized properly which could cause a steel profile to have a concrete quality. This has been resolved. (helpdesk #1001 and #1060)
- 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)
- 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)

Concrete

- For old reinforcing steels with low yield strength (FeB220), no resistant moment MRk could be determined (check on cracking). This has been resolved. (helpdesk #1210)

version 3.06.01

new

General

- In the display options you can now set whether the unity check should be displayed incl. or excl. deflection. (helpdesk #1018)

improvements

General

- In the wizard the combination "SLS permanent" was not generated automatically. That combination is necessary to test the additional deflection. This has been solved. (helpdesk #898)
- 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)
- 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

- 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).
- 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)
- Equilateral and unequal angle samples in the profile database have been expanded. (helpdesk #1007)

version 3.05.01

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 startup, 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

Concrete

- In order to determine the minimum coverage, the environmental classes XF1, XF2, XF3, XF4, XA1, XA2 and XA3 were automatically converted. This conversion is no longer done. In the input, for the environmental classes XF2 and XF4, an XS or XD class must always be selected additionally. Furthermore, the environmental class XC has been made mandatory. This is solved in the UI. (helpdesk #888)

version 3.04.03

new

General

- T

improvements

General

- The program crashed when creating the output. This has been solved. (helpdesk #874 and #881)

version 3.04.02

new

General

- T

Concrete

- A radio button has been added to the screen with which you can easily switch between the graphical representation of the reinforcement's

envelop lines. (Longitudinal reinforcement (UGT), longitudinal reinforcement (BGT) and stirrup reinforcement).

improvements

General

- 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 beam on an elastic soil, the beam width could not be adjusted anymore. This has been solved. (helpdesk #843)
-

Concrete

- When the profile dimensions were adjusted by using the text fields (instead of clicking on the dimension lines) the section properties (A , I_y , etc.) were not recalculated. This has been solved. (helpdesk #783)
- The output of reinforcement and deflection can be turned off. (helpdesk #844)
- The stirrup design in the case of a slab (on an elastic soil) has been improved. (helpdesk #856)

version 3.04.01

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).
- 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

- 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)
- 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)
- In the case of a beam on an elastic soil, the wizard used an incorrect width so that the spring values were incorrect. This has been solved. (helpdesk #763)
- Profiles that have not been used will no longer be printed.
- Zero loads are automatically removed.

Concrete

- In the case of a beam on an elastic soil, the moment M_{Ed} and the x in the tables Longitudinal reinforcement - Extreme boundary condition / Usability boundary condition were not displayed correctly. This has been improved. (helpdesk 2020012001)
- Interactively changing reinforcement: Changing the stirrup spacing and the reinforcement area has been greatly improved. Now this can be done graphically by clicking on the relevant dimension line. (helpdesk #738)
- The dimension lines were drawn with arrowheads. The texts of the dimension lines for stirrups were mixed up. This has been improved. Now dots are drawn.
- For the calculation of the shear resistance V_{Rdc} the longitudinal reinforcement A_{sl} is taken into account. Only the basic reinforcement is taken into account here because this reinforcement extends into the supports. This is a conservative assumption. The smallest value of V_{Rdc} was assumed. That was also a conservative assumption. New is that now is investigated which bending occurs locally, and the relevant basic reinforcement is taken into account top or bottom for the determination of V_{Rdc} .
- In the case of a beam on an elastic soil, the shear force V_{Ed} and the x in the tables Shear reinforcement were not shown correctly. This has been improved. (helpdesk #761)
- The deflection calculation is now optional. By default the deflection calculation is off. (helpdesk #749)
- For a concrete cross-section of, for example, 1000x150, the horizontal dimension line was drawn through the cross-section. This has been improved.

