Release notes XFEM4U 2025

Release version 2025-03-01 (31-08-2024)

New

General

- Analysis: Analysis was automatically restarted after every (minor) change. During design, this can be annoying because it takes time. What is new is that XFEM4U now asks you if that analysis should be restarted or not.
- **Code design check**: During each analysis, the code design check of all beams and plates was performed. During the modelling of a structure and temporary calculations, the code design check is often not relevant yet and requires unnecessary calculation time.
  New is that the code design check can be set in the menu. In new models this is off by default.
- Also. The code design check is now adjustable for each beam and plate. Not in all cases is that check relevant. By default this is on but can thus be turned off. (helpdesk #2342)
- Surface load: When you hover the mouse over a surface load, the status bar displays the value, area and total force.
- 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.
- A new Select drop-down menu has been added to the ribbon menu with the 9 new functions, "***Get previous selection s***", "***Select by coordinate range***", "***Select beams in XY plane***", "***Select beams in XZ plane***", "***Select beams in YZ plane***", "***Select beams parallel to***", "***Select all free nodes***", "***Select all***" and "***Invert selection***".
- The ribbon menu now includes all editing functions in a drop-down menu. So you can now use either the right mouse / context menu or the ribbon menu. It just depends on what you like.
- You can use the new edit menu button function and/or right-click function "***Split beam***" to have beams split into individual beams. All beam loads are, of course, split in the process as well.
- And the way back is also there. You can use the new edit menu button function and/or right-click function "***Merge beam***" to merge the beams into one beam again. And again, all beam loads are neatly included. (help desk #1902)
- **Company logo**: It is now possible to include your company logo in the page 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)
- In the previous release, a new and much faster solver was built in. The menu option “Design calculation / Final calculation” is therefore much less

relevant and also dropped. By default, the analysis is geometrically nonlinear.

Improvements

General

- Analysis: The geometric nonlinear analysis is determined with an iterative calculation. With the introduction of the new superfast solver in the previous release, that iteration no longer ran correctly in all cases and has therefore been improved. Also the iteration on tension and compression beams has been optimized.
- Plates: An error was found in drawing the moments in the intersection line, among other things. This has been resolved. (help desk #2316)
- Beam loads: The texts of coinciding (derived) beam loads were often/almost always mixed together. This has been improved a lot. (help desk #2300)
- Load combination table: it was no longer possible to use Remove to extract a load case from the combination. This has been resolved. (help desk #2312)
- Grid lines: automatic dimensioning of grid lines is now optional. Also, the distance of the dimension lines from the model has been made dependent on the size of the model. This distance was 1500 mm but now depends on the size of the model.
- Display plate results: The number of decimal places for moments, forces and displacements can be set, but this was not used when displaying plate moments and forces. This has been fixed. (help desk #2311)
- It was not possible to change the start and end node of a dummy beam and a rigid element. This has been fixed.
- The radio buttons in the environmental classes concrete dialog box were no longer listed below each other. This has been fixed. (help desk #2326)
- When a beam was removed, the nodes were not removed. This has been improved. The nodes are now removed as well.
- Renumbering: When renumbering nodes and beams, all beams are now put in the same main direction as much as possible. (help desk #2325)
- All screenshots (*.png) were stored in the project file folder. This has been modified. Now the screenshots are stored in the subfolder "XFEM4U_Images". This is much more convenient for the user. (help desk #2331)
- For rigid links, the hinged beam connections were not drawn as spheres as they were for beams. That has been fixed. (help desk #2294)
- Display option plates: When switched off plate results were still drawn. This has been fixed. (help desk #2293)
- The tool (struct4u_speckle_export.exe) to export a model to Speckle server has been updated and digitally signed. (help desk #2306 and #2308)

- When a model was opened at startup with a view in it referring to the Results tab, the message "License not found" appeared. That has been fixed. (help desk #2329)
- The status bar shows the own weight and paint area. You can also have this determined from a selection of beams and/or plates. New is that it now also checks which layers are on or off. (helpdesk #2346)
- In the ribbon menu the combo box for the load cases is filled dynamically a large number of times. In specific cases, a software problem occurred in the process, causing the application to abort. This problem has been resolved. (help desk #2356)

Concrete
- Display plate reinforcement: the legend of the plate reinforcement was not displayed correctly in all cases. This has been resolved. (help desk #2314)
- In solid view, the restraints of the plate edge were drawn by mistake. This has been fixed. (help desk #2338).

Steel
- When testing only torsion of hollow sections, it could happen that the cross-sectional classification was not done so the reduced yield strength was also not determined and a unity of infinity occurred. This has been resolved. (helpdesk #2336)

Release version 2025-02-01 (30-05-2024)

New

General
- ** **New solver** ** To determine the analyse, 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.
- ** **64 bits** ** To enable the analysis of very large (plate) models, we have made the switch to 64 bits. *Why?* Even though there is a lot of external memory available in your computer. Under 32 bits, only a limited amount of this memory can be used. Under 64 bits, that limitation actually no longer exists.
- The rendering of the 3d texts is much improved. Although the text is drawn in 2D but always parallel to the projection of the beam or force. (help desk #1826)

- Dimension Lines: It was already possible to add dimension lines. Dimensioning has been made a lot easier with 'continuous dimensioning'. *How*? Very simple. Insert the first dimension line and then click sequentially on the nodes that must also be dimensioned. Using escape or the right mouse button stops the dimensioning. (help desk #2115)
- When selecting beams you could already right-click →  ***Extend beams***, have the intersection point determined and the beams extended. And when selecting beams you could already right click →  ***Determine intersection points of beams***, determine the intersection point and a new node was generated.
  New is that this now also works for guide lines and a combination of beams and guide lines.
- Gridlines: Gridlines are now automatically dimensioned.
- When you select beams and/or plates, their own weight is displayed directly in the status bar. If nothing is selected then the total weight is displayed.
- Plates/Surface Loads/Shells. The x-axis of the local coordinate system is always parallel to the first side to be drawn. With the new right-click menu function: ***Rotate around local z-axis*** the coordinate system can now be adjusted. (help desk #1706 and #2196)
- 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. (help desk #2251)
- Display Options: By default, the Display Options screen is automatically hidden on the left side. When you hover the mouse over the Display Options, the screen automatically becomes visible and you can adjust all settings.
  You can also lock this screen so that all display options will always be in view as it was. It just depends on what you like.
  The advantage of auto-hiding is that there is more space for the model and fewer buttons in view. This also makes the whole thing look calmer and clearer.
  You can no longer close the display options screen, which is why the corresponding button in the ribbon menu has also been removed.

<u>Concrete</u>

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

Improvements

<u>General</u>

- Plates: A beam on a bedding automatically generated intermediate nodes on a spring. The plate contour was incorrectly automatically divided with those nodes. This has been fixed. (help desk #2098)

- When displaying beam groups, the nodes were not displayed. This has been resolved. (help desk #2108)
- When the description for the combinations was not filled in, the program broke down when creating the output. This has been fixed. (help desk #2113)
- Plates: In a single case, nodes of the plate contour could no longer exist. Therefore, the program broke down. Now those nodes are removed from the plate contour. Although the plate is then not drawn correctly, the user is able to fix it. (help desk #2112)
- Backups were stored in the project file folder. This has been modified. Now the backup-ups are stored in the subfolder "XFEM4U_Backup". This is much more convenient for the user. (help desk #2110)
- Adding beams: When drawing a (dummy) beam for the first time it is asked to add a profile. When Cancel was chosen here, a beam was added without a profile reference, causing the program to abort during analysis. This has been improved. (help desk #2126)
- Surface loads: In a linear running surface load, the load at any point was not calculated quite correctly. A slight deviation was noted. This error has been fixed. (help desk #2103)
- In specific cases, the program broke down when renumbering. Beams whose nodes no longer exist were taken out. This process has been improved. (help desk #2129)
- Rigid links: it will be possible in release 2025.1.1 to define the connection at both ends, just like beams. (help desk #2040). Reading old projects did not go well as a result and has been improved. Both ends are taken equally for old projects.
- Beams / Rigid links: You can no longer adjust the Ty and Tz. It is always set to "A". This prevents arithmetic problems.
- The program automatically creates backups with a sequence number. However, when this file already existed, the program broke down. This has been improved. (help desk #2136)
- Dialog box plates/reinforcement: The combo box of c.t.c. distance appears to be too small, causing part of the number to be omitted. This has been improved. (help desk #2137)
- Dialog box plates/reinforcement: The default c.t.c. distance of the base reinforcement has been reduced on request from 250 mm to 150 mm (help desk #2138)
- When the number of load cases became larger than the number of load combinations, the program broke down when creating the output. This problem has been resolved. (help desk #2140)
- In specific cases, the program broke down when adding a new combination using the small dialog box (momentaneous factor x load factor). And then only on the last load case. This problem has been solved. (help desk #2143)
- XFEM4U broke down when a projection surface load was not properly entered. This problem has been resolved. (help desk #2145)

- Apparently it could happen that beams referred to profile data that no longer existed causing the program to abort. Now those beams are automatically removed. (help desk #2158)
- When you change the view, the combobox load case is changed but not the text in the tab at the bottom. This issue has been resolved. (help desk #2159)
- With the introduction of the Navigation Cube, the texts behind a transparent plane such as a q-load or moment line were no longer drawn. This has been fixed. (help desk #2159)
- 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. (help desk #2069)
- Beam dialog box: When modifying multiple beams with different angles around the x-axis or different deflection requirements, an error message occurred resulting in the OK button being disabled. This issue has been resolved. (help desk #2165 similar to help desk #1970)
- When automatically generating the combinations, the numbering did not go correctly. This has been resolved. (help desk #2166)
- The (re)generation of spring supports at beams on a bed did not always go well. This has been improved.
- Surface loads: For an inclined slab with multiple nodes on the edge, the introduced surface load was not taken into account.
  What was going on? When you add nodes on the edge of the plate, the coordinates are rounded to 0.1 mm. So in fact, the plate is then no longer pure plane. To investigate whether a surface load falls on a plate or not, the surface load is transposed to the local coordinate system of the plate. Then the z-coordinate is tested. The tolerance of that z-coordinate test turned out to be too small. We increased it. This solved this problem. (help desk #2191)
- If you double clicked on a plate, beam, beam load or node load after calculation without changing anything, it was mistakenly seen as a change and the calculation had to be restarted. This has been improved. (help desk #2200)
- When you selected multiple plates of different thicknesses, and you changed the mesh size of those, the plate thickness of the first plate became 0. This problem has been solved.
- A memory problem could occur when the cursor had to be recreated each time. This was solved by generating all cursors only once and holding them in memory. (help desk #2224)
- Display Options: The perspective / orthographic setting was not saved to the views. This has been resolved. (help desk #2236)

<u>Concrete</u>

- Dummy beams were incorrectly included in the combo box of beam groups on the concrete tab. This has been corrected. (help desk #2161)

- Plates: The crack width requirement is increased by the factor kx. This calculation was not easily seen in the output. This has been improved. (help desk #2172)

Steel

- 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.

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).

Release patch version 2025-01-03 (06-02-2024)

Improvements

General

- In specific cases, the surface load was put on the plate in the wrong direction. What was going on? Based on the contour, the direction of the normal is calculated. And herein, for the surface load, was the problem. It also has to do with where you start drawing. Another peculiarity is that it only goes wrong when the contour lines run inward. This has been solved. (help desk #2148)

Release patch version 2025-01-02 (26-01-2024)

Improvements

General

- Dialog box node loads: When you wanted to change the load case of multiple selected node loads, the program broke down. This has been fixed. (help desk #2127)
- Dialog box concrete cross section: The coverage was no longer automatically adjusted. Also, the reinforcement was not adjusted right away in the drawing. This has been resolved. (help desk #2121)

Release version 2025-01-01 (16-01-2024)

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.
- ** **Navigation Cube** ** In AutoCad Revit, the *View Cube* at the top right of the screen lets you quickly switch views by clicking on it. ***XFEM4U*** now also features a similar *Navigation Cube*. You can use the right mouse button to show or hide the *Navigation Cube*.
- In the transparent calculation output of steel and timber, the decisive check is shown in **blue** for uc < 1.00 (or **red** for uc > 1.00). This allows

you to quickly see which check is decisive so that you can more easily optimize your structural design.
- Tables load cases and load combinations: It is now possible to select and delete multiple cases and combinations, respectively. (help desk #2038)
- Table load combinations: It is now possible to select multiple combinations and change them to active or inactive respectively. How? Select multiple combinations and right-click to choose menu option Active.
- Surface loads: It is now possible to copy Surface loads. (help desk #1424, #1495 and #2026)
- It is possible to create a construction parametrically using Microsoft Excel. You can define your own parameters in Excel. From Excel an XML is generated that is also directly loaded into XFEM4U. So you see the result immediately and you can start calculating that construction right away.
- 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. (help desk #2051)

Improvements

General
- In the built-up section dialog box, the angle radio button was not displaying correctly. This has been fixed. (help desk #1963)
- Beams dialog box: When modifying multiple beams with different angles of the profile, an error message occurred resulting in the OK button being disabled. This issue has been resolved. (help desk #1970)
- There were problems with internal merging of nodes. This problem has been resolved. (help desk #2004)
- The program broke down when drawing a section line when there were no plates in the model at all. This problem has been fixed. (help desk #2028)
- Node loads on nodes with eccentricity were not included. This problem has been resolved. (help desk #2037)
- When an orientation node that no longer existed was used, an error occurred and the construction was also not calculated.

This has been improved. Now it is checked per beam whether the orientation node exists. If not. Then the "Orientation node" flag is set off for that beam(s) and a message appears on the screen. The user has to take action himself and select a correct orientation node for that beam(s). (help desk #2034)

- When a global beam load was entered and an angle was also specified, the calculation did not proceed correctly. For global bar loads, the angle is now automatically set to 0. (help desk #2008)
- 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). (help desk #2014)
- Rigid links: it is now possible to define the connection at both ends, just like beams. (help desk #2040)
- Plate: When an opening was specified resp. changed in a plate, the new mesh was not immediately redrawn. This has been fixed. (help desk #2031)
- Setting the number of springs for a beam on an elastic soil has been improved. (help desk #1964)
- Node loads smaller than 0.1 kN and 0.1 kNm, respectively, were incorrectly not drawn. They were being calculated. This has been resolved.
- Dialog beams: When you turn off "Split into structural member sections for analysis" you will now immediately be asked if you actually want to do that. This will hopefully prevent this from being turned off accidentally. (help desk #2021)
- Load generator / surface load: A number of important rules regarding the use of the load generator or surface load are described in the manual. (help desk #1978)
- The number of load cases in the table combinations was limited to 30. This has been expanded to 40 maximum load cases. (help desk #2076)
- Load generator / surface load: When multiple shells of different shapes and load bearing directions were applied in the same façade, wind loads were not generated properly in all cases. This has been improved. (help desk #2075)
- Plate: In some cases it fails to generate a good mesh. This may be because the nodes are too close together or the mesh size is chosen too rough or too fine. The user must then adjust this. However. In specific cases, the visual representation (with OpenGL) of the mesh succeeded but the overall model disappeared when rotating due to a faulty internal OpenGL control. This problem has been solved. (help desk #2013)
- For rigid links, "Split into structural members for analysis” is turned off by default. (help desk #2074)
- Rotating plates: To rotate a plate, the nodes must be selected. If no node of the plate is selected all the nodes of the plate are now selected automatically.

- Mesh a plate: When a node lies within a hole of a plate, the mesher thinks that node should be included and leads to an incorrect mesh. This issue has been resolved. (help desk #2032 and #2080)

Steel
- Built-up section: The self-weight was not calculated correctly in all situations. This has been resolved. (help desk #2007 and #2064)

Concrete
- Longitudinal reinforcement: the diameters Ø5, Ø7, Ø9 and Ø14 have been added. (help desk #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. (help desk #2053)

Release version 2024-03-02 (21-09-2023)

Improvements

General
- Problems were identified regarding copying beams. This has been resolved. (help desk #1952, #1957, #1958 and #1959)
- When beams had no start or end node, the program crashed during (internal) renumbering. This has been solved. Those beams are now deleted automatically. (help desk #1958)

Release version 2024-03-01 (02-09-2023)

New

General
- ** **Hinged connection in plates** ** Plate elements were always calculated as continuous and moment fixed. New is that plate edges can be entered and calculated as hinged. (help desk #1600)
- In the beam load dialog box, you can now also change the distances (a, b and L) by clicking on the dimension lines. This works very intuitively.
- It is possible to enter drawing guide lines. (help desk #1105 and #1905) Guide lines can be very convenient when entering your construction. A display option has been added so you can show or hide these guide lines.
- 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.

- Plates/walls: The entry has been modified. Now the dialog box will be displayed on closing the contour line. This way of working is more intuitive. And it also means that, among other things, the mesh size can be set or adjusted directly.
- Display option: The color scale profiles now list the material name after the profile name. (help desk #1922)
- The Geometry tab displays information about the model (number of beams, plates, etc.) right at the bottom.

Concrete

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

Improvements

General

- Surface loads: In specific cases, not all beam loads were determined. This has been resolved. (help desk #1900 and #1913)
- Drawing q-beam load specifically in global x-direction was not right. The plane of q-load was drawn in the origin instead of at distance "a". This has been fixed. (help desk #1923)
- When the node of a beam on an elastic soil was moved the springs were not re-generated. This has been resolved.
- Plates/walls: When plates are (partially) on top of each other, those plates are drawn in red. (help desk #1562)
- Plates/walls: Meshing of plates has been optimized.
- Dialog box beams: For a tapered beams, additional check that the cross-sectional shape at the beginning and end are the same. (help desk #1921)
- The 'snapping' of a node to a plate has been improved. When multiple plates lie behind each other, the closest intersection point is now taken.

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. (help desk #1924)
- Concrete coverage: For an uncontrolled surface, the minimum coverage was not properly determined. This has been improved. (help desk #1614)
- In the dialog box where the reinforcement is drawn, a new menu option (radio button) has been added that allows you to switch between ULS and SLS envelopes. This allows you to quickly check if the reinforcement meets strength and cracking requirements. (help desk #1889)

Release version 2024-02-01 (03-07-2023)

New

General

- ** **Elastic supporting beams / beams on an elastic soil** **. It is possible to specify an elastic soil for each beam. The elastic soil is schematized as a series of springs that can only absorb positive reactions. Possible "loosening" is therefore provided. The spring spacing can be influenced. The modulus of subgrade reaction (kN/m^3) is automatically converted to springs with the appropriate spring value (kN/m). The generated springs are drawn transparently. (help desk #875, #1527, #1552, #1574, #1748, #1825)
- For plates on an elastic soil, the generated springs are drawn transparently per mesh node. In the view options, turn on "Node of plate mesh" to have all springs drawn. The elastic soil can be displayed properly that way.
- Display of plate stresses and forces: It is now possible to display values within a certain range by entering the maximum and/or minimum value. For example, you can display only those elements with a bending stress greater than 100 N/mm2. (help desk #1819)
- Display Options has been expanded to include an additional "Plates" option. You can use this to quickly draw or not draw the plates. And that's useful with modelling. Sometimes the object plate just gets in the way.

Improvements

General

- Surface loads: It is checked whether the surface load falls into a plate. If so, no beam loads are generated. However, the detection of whether a surface load fell into a plate was not good which in some cases also resulted in no beam loads being generated. This has been resolved. (help desk #1829)
- In version 2024.1, a problem was found regarding plates on an elastic soil. This has been fixed. (help desk #1837)
- Nodes from a hole in a plate do not show up in the layer of the plate. If you then switch off the layer, these nodes remain visible. This has been fixed. (help desk #1823)
- Outside the Section Box, the beams were drawn invisibly and could therefore be selected. This is obviously not desirable and has been fixed. (help desk #1843)
- When copying beams, any orientation nodes were not updated. This has been fixed. (help desk #1847)
- Determining the default orientation node at specifically a column was improved. It was tested for a difference in x and y coordinates of the start

and end node of 0.5 mm. This turned out to be too accurate and has therefore been improved. The difference tested for is now the beam length/1000 with a minimum of 0.5 mm. (help desk #1694)
- On startup, sometimes the error message "Crash Dock Manager is not yet initialized" occurred. This has been resolved. (help desk #1848 and #1856)
- The number of undo steps has been increased from 50 to 250.
- When you removed supports, only the supports (node-restraints) were removed. You had to select the nodes again to also remove the nodes. This has been changed. The nodes including its restraints are now removed at once.
- The Error handler has been improved. (help desk #1874)

Steel
- Testing bending and normal force: 10^3 and 10^6 were missing from the formula of σ_{Ed}. The sign of the normal force was also incorrect. This has been resolved. (help desk #1830)
- Buckling: Nx was printed with an incorrect sign. This has been resolved. (help desk #1830)
- In the testing of the torsion according to article 6.2.7, the stresses in the extreme fibers in x-direction were not properly calculated. This has been resolved. (help desk #1873)

Concrete
- The detailed concrete calculation could no longer be made per node because the right mouse function "Properties" was disabled. This has been fixed. (help desk #1853)
- The number of stirrups sections was printed incorrectly in the plate data. This has been fixed. (help desk #1886)
- It was apparently not possible to display only the reaction forces in a view. When switching between views, the displacements checkbox was turned on each time. This has been fixed. (help desk #1887)

Release patch versie 2024-01-03 (03-05-2023)

Improvements

General

- When copying, nodes that merged were no longer merged. This has been fixed. (help desk #1833)

Release patch versie 2024-01-02 (21-04-2023)

Improvements

General

- When the dimension line referred to a node that did not exist (anymore), the program broke down when drawing that dimension line. This has been fixed. (help desk #1827)

Release version 2024-01-01 (20-04-2023)

New

General

- ** **Water accumulation on roofs** ** It is now possible to test water accumulation on flat roofs. Water accumulation in steel structures of flat roofs is and remains a hot topic. Every year a number of roofs in the Netherlands fail due to water accumulation. Many publications have been written about it and research has been done. The latest calculation rules and insights have now been implemented in ***XFEM4U***.
- ** **Orthotropic slabs** ** Calculating isotropic slabs was already possible. What is new is that orthotropic plates can also be calculated.
- ** **Section Box** ** as in AutoCad Revit, you can use a Section Box to limit the visibility of the model. The objects of the model inside the section box are still visible, what is outside it is no longer visible.
- ** **Match properties** ** (a new right-click function) Match properties allows you to quickly and easily copy all beam data from a selected beam. The cursor changes. Now click on the other beam to copy that data. Right-click or ESC disables this function.
- Plate meshing has been greatly improved in a number of aspects.
- In the graphic you could already display the (Min./Max.) reaction forces. What is new is that you can now display the (Min.) reaction forces and/or the (Max.) reaction forces. (help desk #1691)
- DXF underlay: In the view options a button has been added to add the DXF underlay immediately. An existing overlay can also be deleted with the same button. (help desk #1683)
- DXF underlay: You can move and rotate a DXF underlay. Often the coordinate system in the DXF file is different than in your model. So now you can move and rotate it. (help desk #1730)
- To make communication via our web API with our License Server even more secure, TLS 1.2 (Transport Layer Security) is now being used. (help desk #1737)
- XML export/import: It is now possible to export load cases (including beam loads, node loads and surface loads) and load combinations to an XML file as well as import from an XML file.
- Snap perpendicular to the beam: There is also snapping to the point of perpendicularity. The bullet node is drawn in red and the status bar shows that you connect perpendicular to the beam. Watch the demo.
- The keyboard shortcuts **P** has been added for **P**roperties. For example, if you have selected multiple beams, you can change the properties directly by pressing key P. No need to use the right mouse menu (context menu). This saves clicking with the mouse.

Improvements

General

- For new projects, the snapsize is read in from the ini file. (help desk #1681)
- The snapsize can also be read from the XML file. (help desk #1681)
- 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. (help desk #1710/#1711)
- Load generator - snow load: The roof pitch was not properly determined resulting also in mu1 and subsequently snow load. For steep roofs, this caused the magnitude of snow load to be too high (conservative). This has been solved. (help desk #1716)
- The torsional moment of inertia Ix for rectangular cross section was not properly determined. This has been resolved. (help desk #1720)
- Surface loads: Selecting surface loads has been improved. When you click on one of the components, all are selected as a "block".
- Q beam load: Selecting the q load has been improved. Clicking on any of the components will select all as a "block"
- Large shear forces developed in a tapered profile. The cause: A tapered profile is approximated by dividing it into a number of sub beams, each with a different moment of inertia. A beam is also divided when nodes are found on the beam. In this case, it occurred that the nodes generated were too close to existing nodes, making the partial beams very short and causing computational problems. This was solved by checking that the nodes to be generated do not get too close (within snap distance) to existing nodes. (help desk #1738)
- Adding a plate: When you ended drawing a plate after entering 1 or 2 nodes with the right mouse button, the program hung. This has been fixed. (help desk #1451 en #1734)
- You could already set the number of decimals of forces and displacements. The sum of forces and largest displacements (bottom right of the screen) did not change with it. This has been solved. (help desk #1416)
- On the Load tab, it could happen that you unintentionally deleted also a beam, for example, due to a wrong selection. This has been improved. Now the question is always asked when you did not select only loads. (help desk #1754)
- For a large number of combinations, the table combinations was not printing correctly in all cases. This has been resolved. (help desk #1756)
- When using the "Hinge" menu option, the beam load(s) were not split properly. For example a q-load was running through across the 2 beams causing erroneous mechanics results. This has been resolved. (help desk #1763)
- A serious error was found with a surface load on a roof surface at exactly 45 degrees. Mathematically, the sign of the load flipped, causing the load to be in the wrong direction. This error has been corrected. (help desk #1766)

- After a zoom selection the selection remained. This has been improved.
- XML import: Import of nodes was not quite correct. This has been improved.
- In specific cases beam and node numbers of transparent drawn beams were shown. This has been improved. Beam and node numbers are never shown when the beam itself is drawn transparent. (help desk #1775)
- When joining nodes within the snap distance (CheckOnCoincidingNodes function), the cross-sectional size of the connecting beams in the node is now considered. In this way, the nodes of the main supporting structure are given more priority (weight). (help desk #1789)
- For views of grids and levels, the remaining beams were drawn transparently. For large construction, this was not clear. This has been improved. Now other beams are no longer drawn transparently. You can still show them by enabling "Draw hidden layers layers transparent". (help desk #1782)
- Meshing of plates: When the nodes of the plate contour were not completely in a plane, the mesh nodes were not taken together with the nodes of the plate contour. This has been improved. The testing of the distance between nodes has been increased from 1 mm to 5 mm. (help desk #1799)
- When you copy a beam with the function *Visibility* switch on the new beam is not drawn visible. This has been fixed. (help desk #1804)
- Plates: The modulus of elasticity at material *Other* could not be adjusted. This has been resolved. (help desk #1802)
- Snapping to beams has been further extended to include a snap to the intersection with the lines of the DXF underlay, and the grid lines. (help desk #1764)
- When a plate is removed and re-entered, all data (slab thickness, concrete grade, etc.) is remembered. (help desk #1753)
- A triangular plate was not drawn as solid. This has been fixed. (help desk #1736)

Concrete

- The determination of the maximum value in the color scale of the reinforcement quantities (Asxt, Asyt, Asxb, Asyb and Asw) did not examine whether the plates (using layers) were visible or not. This has been improved. (help desk #1742)

Steel

- When the steel standard was off in the calculation settings, the program broke down when printing the yield strength. This problem has been resolved. (help desk #1697)
- 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. (help desk #1745)

- Beam-column moment connection - butt joint: The stiffness of the end plate was charged 1x instead of 2x. This has been resolved. (help desk #1666)
- Beam-column moment connection - butt joint: In the stiffness calculation the bolt length was not correctly determined This is solved. (help desk #1667)
- Beam-column moment connection - butt joint: In the stiffness calculation a stiffness modification factor of 2 instead of 3 was used.

Timber

- 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. (help desk #1698)

Patch version 2023-02-02 (05-01-2023)

improvements

General

- Surface loads: The beam loads were not correctly determined in all cases. This has been resolved. (help desk #1704)

Release version 2023-02-01 (09-12-2022)

New

General

- A new element, ***Rigid link*** has been added.
- Portuguese (PT) and Portuguese (BR) languages are provided.
- When you select beams you can now right-mouse click → ***Extend beams***, have the intersection point determined and extend the beams.
- Surface loads: In the beam dialog, you can enter factor by which the generated beam loads from the surface loads are increased or decreased to account for the static indeterminacy of the imposed plate. By default, this factor is set to 1.00.
- Load generator: It is now possible to enter the base wind velocity Vb0 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

- Non ASCII characters (e.g. Spanish characters) could not be displayed in the 3D screen. This has been fixed. (help desk #1641)
- Load generator wind: It was found that the height ground level was not included in the determination of the height. This has been resolved. (help desk #1645)
- Slab calculation concrete: The starting values of the layer thicknesses are reduced so that the initial internal lever arm corresponds to 0.9d. For relatively thin and low loaded slabs, this significantly reduces the amount of longitudinal reinforcement required. When the height concrete pressure zone proves too small to absorb the concrete pressure force, the layer thickness is automatically increased in 1-mm increments. (help desk #1637)
  https://forum.struct4u.com/t/check-reinforcement-calculation-concrete-plate/42/5
- When selecting beams, you could already right-mouse click ***Determine intersection of beams***, have the intersection points determined and a new node was generated. What is new is that this intersection point does not have to be on the selected beams.
- When a DXF overlay was used, the menu option ***Insert internal hinge*** did not work. This has been resolved. (help desk #1669)
- When the grid lines were not drawn, they were incorrectly snapped to. This has been changed.
- When the DXF overlay was not drawn it was incorrectly snapped to. This has been changed.

- For deflection testing, the buckling length about the strong axis (y-axis) is now taken. (help desk #1665)

Steel
- Lateral stability check (Art. 6.3.2.3 table 6.6): In specific cases, psi=M_1/M_2 became NaN. This problem has been solved. (help desk #1668).

Concrete
- Beam grid: Users found it inconvenient and unintuitive that they had to redesign the reinforcement of the beams each time they changed the geometry and/or load. This has now been greatly improved.
  When the beam grid has to be recalculated because the geometry and/or load changes, the reinforcement of all beams is now automatically redesigned as well. For those beams where the reinforcement (longitudinal reinforcement and/or stirrups) has been manually changed by the user (prescribed reinforcement), of course, no reinforcement design is done. (help desk #1570)

Timber
- In the stress check, the beam height was not correctly determined in all cases. This problem has been resolved. (help desk #1674)
- For glued laminated wood, the kh for bending on the weak axis was not properly determined. This problem has been solved. (help desk #1675)
- The test for deflection was incorrectly printed 2x when the combination for total deflection was the same as the combination for additional deflection. This has been corrected. (help desk #1676)

Patch version 2023-01-02 (06-11-2022)

improvements

General
- Plate meshing has been greatly improved in a number of aspects.
- The internal buffers have been enlarged so that large models (over 10.000 plate elements) can be computed.
- When the analysis of large models (more than 1000 nodes) starts, XFEM4U will ask if the structure should be renumbered first. Renumbering significantly reduces the computation time (time to solve the global stiffness matrix).

version 2023-01 (03-11-2022)

New

Steel

- The American steel standard "**ANSI/AISC 360-16**" has been implemented.
- 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. (help desk #1603)

improvements

General

- Built up profile was not drawn correctly in the deformed state when the individual profiles were rotated. This has been resolved. (help desk #1563)
- When the display option "Auto save all display options in current view" is set off and layers disappear when creating a new view. This has been fixed. (help desk #1575)
- Wind load generator: For a saddle roof, zones I and J were incorrectly reversed. This has been fixed. (help desk #1611)
- Wind Load Generator: In the dialog box, the wind pressure was determined (informatively) at ground level. This has been modified. Now the wind pressure is determined at ground level + construction height. (help desk #1567). By the way, the wind pressures on the different surfaces were determined correctly.
- Wind Load Generator: A problem was encountered when generating the load planes. This has been resolved. (help desk #1616)
- Load generator: Generated snow load was displayed with the value (mu). When you turned off "Show wind zones" the load size was displayed. This was confusing. Now snow loads are always displayed with the load size. (help desk #1623)
- Plates: The modulus of sub-grade reaction was not printed for a plate on an elastic soil. This has been resolved. (help desk #1613)

Steel

- Beam-beam moment connection: For a butt joint, the factor k10 was incorrectly not considered in the calculation of the bending stiffness. This has been resolved. (help desk #1606)
- The torsional moment of inertia of some cold-formed profile sections was not calculated. This has been resolved. (help desk #1631)

Release notes XFEM4U

version 5.12.04 (03-10-2022)

improvements

General
- Surface loads: When a surface load was applied to a plate, beam loads were incorrectly generated on the edge beams as well. This has been fixed. (help desk #1590)
- Index out of range when opening the combination table: Opening of old projects was not going well because the number of load cases per combination was increased from 30 to 40. This problem has been fixed. (help desk #1554, #1594 and #1595)

version 5.12.02 (01-09-2022)

improvements

General
- Ground pressures: There appeared to be an error in the translation of the reaction forces to ground pressures. The ground pressure was exactly 2x too large. This has been fixed. (help desk #1557)

version 5.12.01 (11-08-2022)

new

General
- ***Wind and snow load generator***: A load generator has been added to generate all wind loads and snow loads according to the Eurocode. The user draws the all outer shells (panels). Then the load generator can be started.
- With installation of this new release, in addition to XFEM4U, ***XFEM4U-viewer*** is also installed. The use of ***XFEM4U-viewer*** is completely free of charge. The viewer has exactly the same functionality as XFEM4U. Only you cannot save the model or create a pdf output. When you send your input (*.xfem) to your customer, he can view your model with this viewer and examine all results he wants to see.
- The program icon has been renewed.
- Plates: The direction in which a plate is drawn (Clockwise or counter clockwise) determines the direction of the z-axis. With the right mouse button the contour could already be flipped. New is that now you can do

that also in the plate's dialog. The same functionality has been implemented for surface loads and outer shells.
- During startup of the program, a splash screen is shown with pictures of some sample projects.
- Display Options / Layers: Layers can now be made visible or invisible with the click of a button.

improvements

General

- The height of the beams and output selection dialog boxes was too large for a screen resolution of e.g. 1366x768 or 1280x720. This has been adjusted. Now XFEM4U can also be used on any laptop as well.
- There appeared to be a small error in the determination for the spring value when a plate is placed on an elastic soil. This has been resolved.
- Surface loads: When edge beams were partially missing, the beam loads were not generated properly. This has been fixed. (Help desk #1507) Now dummy beams are internally automatically added.
- Surface loads lying on top of each other were not drawn stacked. You couldn't clearly see that multiple surface loads were stacked on top of each other. This has been improved. Now the surface loads that (partly) lie on top of each other are drawn stacked. (Help desk #1088)
- It could happen that surface loads on plates were internally applied mathematically incorrectly. This has been solved. It also no longer matters how the plates and/or surface loads are drawn. Clockwise or anti-clockwise. (help desk #1525)
- For surface loads, beams loads are generated. Unless the beam is lying in a plate. Then it is skipped. When checking whether the beam falls into a plate, it was incorrectly checked whether the plate was visible or not. This has been fixed. (help desk #1529)
- Profile data of dummy beams were printed in the output. This raises questions. Now the profiles of dummy beams are no longer printed. (help desk #1503)
- In the output, the profile names of dummy beams are omitted in the table of beams.
- The input data of the plate was missing in the output. They are now printed. (help desk #1233, #1502)
- Surface load: The load bearing direction is only relevant for beam structures. This has of course to do with the generation of beam loads. Therefore, when the surface load is on a slab or wall, the direction of load bearing in the dialog box is dis-disabled and arrow of load bearing is no longer drawn.
- Surface load: In a number of cases, beams loads were not generated. This has been resolved. (help desk #1493)
- Splitting a beam did not work properly in all cases. This has been fixed. (help desk #1532)
- Output of slabs on springs: In the table of spring values, mesh nodes were not skipped. This has been improved. (help desk #1504)

Steel

- If the first profile was a concrete profile (no steel and no timber) then the EN1993/EN1995 fields were wrongly disabled in the case of a non-tapered beam as a result of which, among other things, the buckling length could not be adjusted. This was of course wrong and has been solved. (help desk #1509)

Timber

- Display option: Decisive unity check excl. deflection worked for steel profiles but not for timber profiles. This has been resolved. (help desk #1534)

version 5.11.01 (23-05-2022)

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.
- Plate number is now displayed graphically when profile names are shown.
- With the new right-click option ***Generate Plate Load***, a surface load over the entire plate can be generated quickly and easily.
- As you know, you can use the spy to retrieve the moment in any cross section of the beam, for example. New is that when you choose an envelope (moment line) the spy shows you 2 values (max/min). (help desk #1443)
- Views: It is possible to change the order of the views afterwards. The views also appear in that order in the output. So now the order can be changed. (help desk #1477)

improvements

General

- When drawing a surface load was terminated with escape key, a linear load could not be specified immediately. This has been fixed. (help desk #1469)
- A number of missing UB and UC profiles have been added. (help desk #1442)
- The support symbols of a plate edge were drawn small and those of a normal node large. This has been improved. Now those support symbols are drawn just as large. Node supports of beams are always drawn larger though. (help desk #1440)
- When only shell elements were entered the headers of the beams and also of the profiles were still printed. This has been improved.
- In the output of permanent load, only the dead weight of the beams was mentioned. The dead weight of the plates was missing from this. This has been added. (help desk #1483)
- Hall wizard: the load cases and combinations were created properly from the wizard. In the combinations table, the automatically generated load cases were not visible as columns. This issue has been resolved. (help desk #1485)
- Hall wizard / Load Generator: The snow load was not determined correctly in other reference periods. This has been resolved. (help desk #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. (help desk #1472)

- After shifting (move up/move down) the profiles in the profile table, and then renumbering of all beams and nodes, beam loads were incorrectly removed. This issue has been resolved. (help desk #1487)

Steel

- Built-up profile: Web plates were not always taken into account in the calculation of shear force surface (stress test). E.g. depending on where the plate is turned. In the selection profile dialog or in the built up section dialog. This problem has been solved. (help desk #1489)

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. (help desk #1466)
- Plates: Changing the reinforcement data of multiple selected plates did not work. This has been resolved. (help desk #1468)
- Plates: The diameter 9 is added in the combo box of the reinforcement data of the plate. (help desk #1476)
- Plates: The bottom reinforcement in y-direction was not read in correctly. This issue has been resolved. (help desk #1475)
- Representation of plate reinforcement: When there were plate elements for which reinforcement could not be designed (because either the pressure zone or the pressure shear diagonal failed), those plate elements were drawn in red. The remaining plate elements were drawn transparent. Disadvantage of this was that you did not see any designed reinforcement drawn at all. Therefore, this was modified. Those plate elements for which reinforcement cannot be designed (because either the pressure zone or the pressure shear diagonal collapses) are now shown in black and the mesh nodes in red. The designed reinforcement in the other plate elements are now shown with a color. (help desk #1471)

version 5.10.02 (11-03-2022)

improvements

General

- The beam loads were not generated properly in all situations when surface loads were applied in one direction. This issue has been resolved. (help desk #1445)

version 5.10.01 (11-03-2022)

new

General

- It is now possible to use a DXF as an underlay. When you enter your construction nodes are snapped to the lines of the DXF underlay. You do not have to enter the coordinates or distances manually. This is not only pleasant but also efficient and reduces the risk of errors considerably. This is a really important addition that allows you to enter your construction even faster.
  In the menu, press the program icon and choose: *Import/Export → DXF-Import DXF → Import DXF underlay*.
- It is now possible to copy beam loads to one or more beams. (help desk #1383). How? Select the beam load(s), choose Copy and click on the beam / beams to which the load should be copied.
- The number of last used input files has been increased from 10 to 25.
- ***Surface loads:*** For slabs and walls, plate loads could be entered (uniformly distributed load over the entire plate/wall) and for beam structures, any surface load from which beam loads can be generated. What is new is that any surface load can now be used for plates and walls as well.
- Surface loads: Surface loads can now also be entered as linear progression by defining the load on 3 points of the load contour. So for example also earth pressure or water pressure can now be entered. Or in the case of a beam construction a running wind pressure.
- Surface loads: It is possible to move the points of the surface loads graphically as you can with nodes.
- Surface loads: Double-clicking on a point of the surface load opens a dialog box where you can adjust the coordinates. (help desk #1402)

improvements

General

- When there is a newer version the installation could not be started from the pop-up screen. This has been fixed. (help desk #1368). The same was true for the release notes.

- In the beam dialog, the torque reduction is only relevant for concrete beams and is disabled for the other profiles. (help desk #1376)
- In the dialog box for the beam, the lateral stability is not relevant for concrete beams and is disabled. (help desk #1376)
- Surface load: Short beams that were not in the load plane were incorrectly included, making it impossible to generate beam loads. This has been resolved. (help desk #1395).
- Beam loads were already drawn "stacked". Any coinciding point loads were not yet drawn stacked but that has been improved. (help desk #1397)
- The tab order in the nodes dialog box has been improved.
- Plate meshing: The meshing has been improved. The minimum number of points/mesh nodes of a (dummy) beam was 3. This has been reduced to 2 which also makes the mesh much more regular with very short beams. Also, prior to meshing, all (dummy) beams with a length of 0 are taken out. (help desk #1390)
- Surface loads: Beam loads were not properly generated when a beam section fell into a field. (help desk #1283, #1292)
- Surface loads: On the Loads tab, all non-loadbearing beams are shown in red. This makes it easier for the user to check his input.
- When you save a project, XFEM4U creates a backup copy of the previous version of the project (that is, the project file before the current save). This backup was named "<project_name>. <nnnn> .xfem", where <nnnn> is a 4-digit number indicating how many times the file was saved. This was found to be confusing. The name of the backup was therefore changed to "backup<nnn>_<project_name>.xfem".
- When drawing global q-beam loads in y direction, the transparent plane was not drawn properly. This has been solved.
- The drawing of q-beam loads has been improved. At the beginning and end 3 arrows are drawn.
- Drawing of surface loads has been improved. Now transparent edges are drawn including 3 arrows at the beginning and at the end.
- Double-clicking on the unity check bullet opens the detailed calculation. This did not work when, for example, you also had the moment lines turned on. This problem has been solved. Double clicking on the bullet now works in all situations..
- In the output selection screen, the combo box for the beams only added the steel beams. The wooden ones were missing here. This issue has been resolved (help desk #1413)
- The paragraph numbering for the detailed calculation output per beam was missing. This has been added. (help desk #1414)
- Graphical representation of enveloping reaction forces: The arrow and the corresponding values were not drawn when the actual force (in the selected load combination) was 0 kN / very small. This error has been resolved. (help desk #1429)
- When the system is potentially unstable, a message appears showing the (up to 10) largest node displacements. (Help desk #1432) Also, those

affected nodes will be slightly larger and drawn in red in the graphic. (help desk #1126)

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. (help desk #1308)
- There was a textual error in formula (6.19). This has been solved.

Concrete
- The longitudinal reinforcement in 2 layers was not printed properly in table "longitudinal reinforcement". The string was truncated. This has been fixed. (help desk #1411)

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

version 5.09.01 (29-11-2021)

new

General
- Wind load generator in wizard: It is possible to enter the structural factor CsCd. It is also possible to take into account the correlation according to Art. 7.2.2(3). (help desk #779, #1253 and #1259)

Concrete
- In a concrete slab calculation, the decisive nodes per slab are detailed in the output. You could already show a detailed reinforcement calculation per node in ULS and SLS, but new is that the decisive nodes are automatically included in the output.
- In the table Longitudinal reinforcement - UGT, the required longitudinal reinforcement (As;req) is now also printed. This can be used to determine how much longitudinal reinforcement is "left over". (As,prov - As,req) And that amount can then be used for that side as torsion reinforcement.

improvements

General
- The colours (in beams and plates) can now be displayed smooth or banded. It could happen that the radio button was not filled causing the program to abort. This has been solved (help desk #1244).

- When a beam was selected without the nodes also being selected, it could not be moved, rotated or mirrored. This has been solved by automatically selecting the beginning and ending node in that case. This works even more intuitively. (help desk #1260)
- In Solid View, nodes are not drawn visibly. What is new is that you can select those nodes with a window of crossing. This makes it possible to move the nodes or to change their properties. This also works even more intuitively.
- When you draw a new beam, the beam data of the last entered beam is copied. This is useful anyway.
  It could happen that the layer was off so the beam was added but not visibly drawn. That was confusing. Now the layers that are not on are no longer checked in the combo box. You will now always see the new beam drawn. (Help desk 1262)
- With large construction, the surface load planes were no longer drawn as planes. This has been fixed. (help desk 1264)
- When you drew a surface load plane and you clicked 2x on the same node causing 2 same coordinates to occur in the contour, you got the error message "Assertion failed". This problem has been solved. All duplicates are now automatically removed. (help desk 1268)
- When you choose "envelope" the minimum and maximum reaction forces are now automatically displayed. (help desk 1273)
- When moving and copying objects (nodes, beams, plates, etc.), you had to click on an existing node as a base point (starting point). This has been improved. You can now take any point as a base point. So also a non-existing point. Moving and copying has been made even more intuitive. (Help desk 1268 and 1277)
- An orientation node can be used for beam orientation. Orientation nodes could be removed while still in use. The program broke down as a result. This has been resolved. (help desk 1229). Also, the orientation nodes were not always visible. This has also been resolved.
- When renumbering, the orientation nodes were not included. This has been resolved. (help desk 1230).
- For large structures, it was possible that the moment lines were not drawn as planes. This had to do with the scaling. This has been solved. (help desk 1279).
- The display of the color scale of unity-check now takes into account any filter values entered. (min./max.) (help desk 1282).
- The iteration process with respect to the tensile rods in a geometrically nonlinear (GNL) analysis (final calculation) did not work properly in all situations, causing the calculation to stop and no results to be displayed. This has been improved. (Help desk 1284).
- The monitor dialog of the analysis has been improved. Now the description of the load case or combination is displayed.
- In the nodes input table, node coordinates are shown with 1 decimal place. (help desk 1285)
- Plates: The color rendering of reinforcement Asw, Asx, and Asy could still become slightly asymmetrical in a symmetrical model. Quadrangles are

drawn as 2 triangles and the direction of the diagonal affected the color gradient in the element. This has been solved. (help desk 1289)
- For a large number of load combinations, it could happen that when making envelope displacements, the program broke down. This has been solved. (help desk 1287)
- Combinations whose load factors and instantaneous factors are missing are skipped in the calculation. When filling the combo box, those combinations were incorrectly not skipped. This has been solved. (help desk 1293)
- When printing reaction forces per load case, all nodes were incorrectly printed. This has been resolved. (help desk 1294)
- Changing multiple selected beams at once did not work properly in all cases. This has been resolved. (help desk 1300)
- Drawing a surface load does not go well when, for example, one of the selected nodes does not lie the plane. The program broke down because of this. This has been solved. An error message is displayed. (help desk 1302)
- Moment lines at long beams with many intermediate nodes (sub beams) were not always well represented. In particular, the last point was missing. This has been solved by adding extra drawing points. (help desk 1311)

<u>Concrete</u>
- Slabs: The color rendering of the reinforcement did not go well everywhere. Elements that did not require reinforcement (longitudinal and/or transverse reinforcement) in one of the nodes were incorrectly not drawn. This problem has been solved. (help desk #1270)

version 5.08.02 (16-08-2021)

improvements

General

- When rotating, beams that were parallel to the axis of rotation were rotated about the x-axis. It was not examined whether those beams had an orientation node. This has been resolved.
- Security of online payments increased by changing TLS (Transport Layer Security) from 1.0 to 1.2.

version 5.08.01 (16-08-2021)

new

General

- The menu now includes **Zoom extents** and **Zoom Selection**.
   **Zoom extents** displays the entire construction with the default camera position.
   With **Zoom selection** the selection is displayed full screen. If nothing is selected, the entire structure is displayed with the last camera position
- The colors (in beams and plates) can now be displayed smoothly or banded. It is also possible to set the number of colors in the color scale.

Concrete

- Beam Grids - Pile Optimization: In the side view of the concrete beam, it was already possible to change the pile distances by clicking on the dimension lines. This is of course still possible.
  What is new is that it can now also be done graphically. You can now graphically "drag" the support. After changing this, the beam grid is automatically recalculated and you can see the result immediately. (help desk #1218)

improvements

General

- It could happen that bars were not visibly drawn. This issue has been resolved. (help desk #1205 and #1207)
- Slabs: In the case of a slab on a bed, the ground pressure could not be displayed. This issue has been resolved.
- The beam loads of non-visible beams were drawn. This has been fixed. (help desk #1200)
- 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. (help desk #1112)
- The 3d visualization of deformations, N-, D- and M-lines has been improved and accelerated.
- Surface loads: Surface loads were not properly adjusted in all cases when nodes were moved. This has been resolved. (help desk #1180)

Steel

- Beam-column moment connection: Adjusting vertical bolt spacing has been improved. (help desk #740)
- Beam-column moment connection: When the connection was adjusted, the analysis was not restarted. Of course, it should be. This has been fixed. (help desk #1204)
- When testing tensile beams, only the tensile is evaluated. (help desk #1173)

Concrete

- Piled concrete foundations: For old reinforcing steels with low yield strength (FeB220), no resistant moment MRk could be determined. This has been resolved. (help desk #1210)
- Beam grids: In the 3d view, the beams were drawn as "Solid" by default. This has been modified. (help desk #1206)
- Plates: The reinforcement data disappeared on the 2nd opening of the screen after the creep coefficient was calculated. This issue has been resolved. (help desk #1217)

version 5.07.01 (25-07-2021)

new

General

- **The 3d navigation is greatly improved!** The 3d navigation and selection now works like in Autodesk-Revit resp. Tekla structures. You can configure which method (Revit or Tekla) you want to use.

  Note: As you know ***XFEM4U*** also supports the use of a 3d mouse (from e.g. 3dconnection) which makes working even easier and faster.

- The menu option "Zoom extents" has been expanded and greatly improved. When you select objects and click on "Zoom extents" they are zoomed in. Also the navigation pivot point is taken in the center of the selected ones and the camera position is now held.
- As you know you can use the menu options "Visibility" and "Search" to easily make a selection of beams and/or plates visible. New is that now you can select only the visible beams, nodes and/or plates which you can also easily change. (help desk #1119)
  That means you can now make sub-selections. How? Select a section and press "Visibility". Rotate the model. Now select a section again and press "Visibility".
- Copying has been greatly improved: A new feature is that the COPY command is repeated. So you can place your copy in several places and thus enter your model even faster and easier. (help desk #1138)
- In the dialog boxes of the beams, plates and surface loads, you can now also edit or add the layer names directly. (help desk #1121)
- Plates: at the location of holes, nodes are now generated allowing you to enter (dummy) bars and line loads more easily. (help desk #1049)
- Display options: The list combo box views also automatically contains all grids and levels. The depth is not (yet) adjustable and is now 500mm in front and 500mm behind. Only the objects within these distances will be visible. (help desk #1183)

improvements

General

- Print Preview: The output screen could be resized with the Minimize menu option. That menu option has been removed. (help desk #1080)
- Surface load: When a beam was copied that coincided with one of the sides of the surface load, the surface load was moved incorrectly. This issue has been resolved. (help desk #1089).
- Wizard: In the hall wizard you can now also enter the design working life. That was not possible and was set to 50 years. Now you can. (help desk #1092)

- The color scale for displacements and plate stresses was shown as an envelope. Now the color scale is displayed per load case or load combination. (help desk #1098, #1122, #1141)
- The color scale for the unity check now matches the visible beams. (help desk #1141)
- The sphere of a hinged beam connection is drawn smaller and is no longer selectable. This allows you to select the node more easily.
- In the output selection dialog you can now also edit the project data. The part item is now a combo box and the previously entered texts are saved. (help desk #1139)
- If after calculation it turns out that there are unstable beams, they can now be made visible with "Show unstable beams".(help desk #1189)

Steel

- Changing the lateral distances of multiple selected beams did not work properly. This issue has been resolved. (help desk #1114)
- 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. (help desk #1124)
- In the case of a tube, in the testing of the torsion, the longitudinal stress sigma,x was not calculated correctly. In the formula (My.e)/Iy, for the e the diameter was taken. And that should of course be the radius. This has been fixed. (help desk #1190)

Concrete

- It was possible to display unity checks (0.00). This was confusing. For concrete beams, unity checks are no longer displayed. (help desk #1118)
- In the calculation of plates and walls, the compressive strength was reduced to 0.6 fcd and 0.8 fcd for some cases. After recent discussions between Johan Blaauwendraad and René Braam, it was decided not to apply that reduction anymore.

version 5.06.02 (27-05-2021)

new

General

- As you know, with the menu option "Visibility" you can easily make a selection of beams and/or plates visible. New is that when you now create output, only the visible beams and/or plates are printed. So you can very easily and quickly make an output of a part of your construction.

Steel

- A new (context) menu function "Create bill of materials" allows the creation of a Excel file of all visible beams. (help desk #1038). So you can quickly create a bill of materials with this.

Concrete

- Slabs: For slabs on an elastic soil, soil pressures can now be displayed in colour. (help desk #747, #942, #1040)
- Beam Grids - Pile Optimization: In the side view of the concrete beam, dimension lines are drawn between the nodes and piles. You can now easily adjust the pile distances by clicking on the number in the dimension line. After changing this, the beam grid is automatically recalculated and you can therefore immediately see the result. (help desk #1042)

improvements

General

- Nodes and node reactions of plates that are not visible were drawn. This has been fixed. (help desk #1035).
- For very complex models, the required "bandwidth" for solving the stiffness matrix could be too small. This has been resolved. (help desk #1036)
- For plates, the plate thickness was always displayed. This has been improved. When at display option "Profile names" is on, the thickness is displayed. (help desk #1038)
- Slabs on an elastic soil: If the display option "Profile names" is on, the value of the modulus of sub-grade reaction is now also displayed. (help desk #1071)
- Slabs 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. (help desk #1044)
- When editing from the profiles table of a built-up section, the profile name was not visible. This has been resolved. (help desk #1045)
- Plates: It was not possible to remove a node from a plate edge. This has been resolved. (help desk #1004)
- View options: A checkbox "Auto save all display options in current view" has been added that allows you to set whether all view options should be

automatically saved in the view. By default, this is "on." (help desk #1070)
- When nodes coincided, no calculation was done and no input error messages were displayed. This has been improved. Any input errors are now made visible. (help desk #1063)

Steel
- The beam deflection of a bar group with a kink (buckled roof girder) did not proceed properly. This has been resolved. (help desk #1037)

Concrete
- Beam Grid: In the top 3d view, a Solid View is now always shown and deflections are turned off. This is clearer. It also makes it easier to switch beams by clicking on a beam in the 3d view.
- The minimum cover was not properly determined for environmental class XC4, among others. This has been resolved. (help desk #1047)

version 5.05.01 (3-05-2021)

new

General

- **Plates and shells** - In earlier versions, we used a triangular hybrid element. On the advice of **Prof. Johan Blaauwendraad** we switched to a quadrangular hybrid element. This element gives even better results arithmetically. Johan Blaauwendraad supported us in the realization and implementation of this element.
- **Plates and shells** - Because of this transition from triangular plate elements to quadrangular plate elements, an entirely new and powerful mesh tool has also been implemented. This new mesh tool is very fast and generates a nicely regular element grid which of course greatly increases calculation accuracy.
- **Plates and shells** - Mesh refinements had to be specified at the plate edges. At the connections with other plates the plate edges had to be selected with a crossing. That did not work fine and was therefore adjusted. Mesh refinements are given at nodes. That works much more intuitive so you can do your input even faster.
- **Line Loads on plates -** Using newly added *dummy* beams, any line load on the plate can now be charged.
- **Linear supports under plates -** Using newly added *dummy* beams, any linear supports under the plate can now be accounted for.
- It is now also possible to export the model via SDNF format. The model can be imported from and exported to Tekla Structures via SDNF format.
- In the output of the table beam forces you miss the beam forces at intermediate nodes. New is that these are now also printed. (help desk #974)
- **Spy**: A spy function has been added. Moving the mouse over the beam axis displays all the forces (Nx, Mx, My, Mz, Vy and/or Vz) in that cross section. You don't have to set anything. Just move the mouse. It works very intuitively.
- The "snapping" to a beam has been improved. Now it "follows" the beam that is closest to.
- In the display options you can now set whether the unity check should be displayed incl. or excl. deflection. (help desk #1018)

Concrete

- Plates and walls: The reinforcement is calculated according to Dutch report "***Wapenen van Schijven en Platen***" prepared by **Prof. Johan Blaauwendraad**. The reinforcement is designed for strength (ULS) and for cracking (SLS).
- For concrete profiles, you can now change the profile name in the dialog box. (help desk #983)

improvements

General

- Display options: The check mark for the envelope was not saved. This has been solved. (help desk #925)
- 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. (help desk #939)
- Beam loads: Q-loads with length L=0 are not drawn but were charged. This has been fixed. Now those beam loads are automatically removed. (help desk #982)
- In the ribbon menu, the combobox "Design calculation / Final calculation" was outside the ribbon page block "Calculate". This has been fixed. (help desk #984)
- Display unity checks: Double clicking on a node in the display of unity checks incorrectly displayed the calculation of a beam. This has been fixed. (help desk #988)
- The beam load qxy is always a global load, that is, relative to the main axis system. It turned out to be possible to enter that load as a local load. This has been solved. (help desk #987)
- Rotate: Beam that are parallel to the axis of rotation are rotated about the x-axis.

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. (help desk #948)
- In specific cases, the MyEd in the stability control according to 6.3.3 was not determined correctly. This has been resolved. (help desk #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. (help desk #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. (help desk #918)
- A textual error in the output: Formula (6.42) incorrectly contained the factor "a". This has been removed. (help desk #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. (help desk #991)
- Equal and unequal angle profiles in the profile database have been expanded. (help desk #1007)

Concrete

- Beams with offset works in the line view but not in the solid view. This has been fixed. (help desk #973)
- Entry of reinforcement is made easier. (help desk #1016)

Release notes XFrame3d

version 5.04.02 (1-12-2020)

improvements

General
- Moment lines were not immediately visible. This has been solved. (help desk #929)

version 5.04.01 (1-12-2020)

new

General
- **Plate** - It is possible to determine forces in a cross section of a plate. The cross section line can be drawn randomly.
- 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 use 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.
- **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.
- The display option has been extended. When displaying the results, the units can optionally be omitted. (help desk #869)
- Dialog of beams: It is now possible to change the start node and end node.
- Output selection: In the output selection dialog, the default text "Page" (in the page header) can be changed to, for example, "Attachment A".
- **Import** - It is possible to import another XFrame3d model.

improvements

General
- Plates: When a node is added and the node falls into a plate, the plate is re-mesh-t.
- Plates: When a beam is added and both nodes fall into the same plate, the plate is re-mesh-t.
- Menu functions ***Rotate*** and ***Multiple Copies*** have been improved and accelerated. (The construction was drawn too often).
- It was not possible to turn the weight of multiple selected plates on/off. This has been solved.

- Plates: Node loads could be entered on mesh nodes. This has been adjusted. This is no longer possible. If you want to enter a node load, first place a node in the plate which you can then load.
- There was a wrong text in the dialog box of the grid lines. This has been solved.
- After renumbering, dimension lines were drawn incorrectly. This has been solved. (help desk #891)
- Problems have been identified with regard to the generation of views of load cases in the output. This has been solved. (help desk #897)
- The node displacement (dx,dy,dz and dxyz) is displayed more clearly. (help desk #902)
- When moving, the surface loads were not taken into account. This has been solved. (help desk #913)
- If the rotation angle was entered in the profile data for built-up profiles, this was not taken into account. This has been solved. (help desk #917)
- When logging in as an existing customer with the Struct4uKey this was not visible in the status bar. This has been solved. (help desk #922)

Steel
- An error has been found in the calculation of k_{zz} from table B.1. This has been solved. (help desk #890)

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. (help desk #888)

version 5.03.01 (26-03-2020)

new

General
- **Hybride plate element** - XFrame3d has been extended with a hybrid plate element.
  As you know it is already possible to calculate plates and walls in XFrame3d. In the preparation to be able to accurately determine the reinforcement as well, we have built in another triangular plate element. This is a triangular hybrid element. **Prof. Johan Blaauwendraad** supported us in the realization and implementation. This new plate element offers the advantage over the earlier element that the shear force is determined more accurately.
- With the new menu option "Visibility" a selection of beams and/or plates can easily be made visible.

How does it work? Make a selection and click on this menu option. Only the selected beams and/or plates will now be drawn. The other beams and/or plates will be drawn transparently. To undo this click on this menu option without making a selection. Watch the demo.
- Node numbers of a mesh can now also be displayed.
- The color texts of nodes, beams, plates, loads, results and others can be set. Node and beam numbers are displayed in different colors.

improvements

General

- After using the Wizard, generated loads were erroneously deleted. This problem has been solved. (help desk #830)
- Calculation settings: The gammaP and gammaQ factors were not adjusted when changing the consequence class. This problem has been solved. (help desk #837)
- Entering the psi and gamma with the dialog did not go well. This problem has been solved. (help desk #842)
- Plates: Plate edges were drawn as transparent tubes. Plate edges are now drawn non transparently, which makes selecting easier.
- Dockpanel *Display options* is now scrollable.
- The display options are disabled / enabled depending on whether plates and / or beams are entered.
- Plates: While you draw the contour, the plate is drawn transparent with a thickness. Nodes inside the plate could therefore not be selected anymore. This is solved. While drawing, the plate is drawn with a thickness of 1mm
- Plates: The notations Mxx, Myy, etc. have been changed to mxx, myy, etc.
- Chapter numbering did not go well when exporting multiple layers. This problem has been solved. (help desk #853)
- Surface loads: In specific cases, beam loads were generated outside the beam. This problem has been solved. (help desk #857)
- The symbols of the loads could not be scaled up. This has been resolved. (help desk #863)
- The window sizes and positions are saved in the ini file (....\AppData\Local\Struct4u\XFrame3d.ini). In specific cases, the width of the window for 3d display could become 0 at startup, causing the program to abort. This has been solved. (help desk #864)
- Double node loads were not always drawn stacked. This has been solved. (help desk #722)
- Wizard: The created load cases were removed. This has been solved. (help desk #867)
- Plates: In the dialog for the holes the axes of the plate are now drawn. (help desk #871)

Steel

- 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. (help desk #833)

Concrete
- The default size (wxh) for a concrete beam has been changed to 300x400.

Patch version 5.02.02 (14-7-2020)

improvements

General
- Pictures in output - In version 5.02.01 problems have been found with regard to the generation of pictures in the output. This has been fixed. (help desk #827)

version 5.02.01 (14-7-2020)

new

General
- **Earthquake calculation** - XFrame3d has been extended with an earthquake calculation according to the Dutch guideline NPR 9998+C1:2020.
- **Dimension lines** - Dimension lines can now be drawn. (several help desk tickets)
- 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
- Display of moment lines has been improved. One scaling factor is now determined for the moments Mx, My and Mz. (help desk #741)
- Display of normal and shear force lines has been improved. One scale factor is now determined for the forces Nx, Vy and Vz.
- Mirroring the construction. Mirroring didn't always succeed while the mirror surface was well drawn. This was because the third point (of the mirror plane) was chosen again by the 'snapping' to other lines. The beam loads were also incorrectly copied. This has been solved. (help desk #743)
- When a plate is removed, the nodes of the plate contour are also removed. This has been adjusted. Those nodes now remain in place.

- When making the output, the program broke down. This is solved. (help desk #754 and #755)
- The visualization of moments in the plate at nodes was not correct. This is solved. (help desk #757)
- When the table profile data was output, the column for Iy and Iz proved to be too narrow, so a digit was omitted. This has been solved. Now an e-power is used. (help desk #762)
- The graphical entry of a surface load was not possible when the table of loads was visible. This has been solved. (help desk #774)
- In the case of the views, the display setting with regard to the plate stresses was not stored correctly. This has been solved. (help desk #777)
- If only a plate load was entered in a load case, the image was not printed. This has been resolved. (help desk #778)
- In the case of an elastic supported plate, the spring values per mesh node were not calculated correctly. This has been solved. (help desk #793)
- In the case of surface loads, it was checked whether the total load corresponded to the sum of the generated beam loads. This check is removed. It is now possible to omit edge beams. For example of the foundation beam in case of wind loads on a facade. Dummy beams had to be entered that also came into the output. And that also raised all sorts of questions.
- When you selected a beam or plate to delete, the nodes were automatically deleted as well. This turns out to be not always convenient and has therefore been removed.
- In specific situations, part of the opengl control on the right side was lost. When you adjusted the size of the main screen (using a maximize or resize) the opengl control was displayed correctly. This is solved. (help desk #799)
- With an enveloping moment line, not in all cases both moment values were displayed. This is solved. (help desk #801)
- For moment lines, small moments (moments smaller than 0.5 kNm) could not be represented numerically. A new feature is that the number of decimals set by the user is now taken into account. Also small moments can now be displayed numerically. (help desk #821)
- In the graphical view, the maximum and minimum reaction forces are displayed with the sign character. (Help desk #819) With an envelope, only the maximum and minimum reaction forces are displayed. (help desk #820)
- In the graphical view, all internal forces are displayed with the sign character.
- Show profiles in color: With a large number of profiles, the list will no longer fit on the screen. This has been improved. (help desk #818)

<u>Steel</u>

- When the profile was rotated, the entered buckling lengths were converted to the axis system of the profile as indicated in the Eurocode, just like the forces. This turned out to be not good. The entered buckling lengths are always in relation to the axis system of the profile as indicated in the Eurocode. (help desk #759 and #760)

Concrete

- 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. (help desk #738)
- When the profile dimensions were adjusted by using the text fields (instead of clicking on the dimension lines) the section properties (A, Iy, etc.) were not recalculated. This has been solved. (help desk #783)

version 5.01.01 (16-04-2020)

new

General

- ***Plates*** – In a plate, holes can now be specified. You can choose from a large number of basic shapes that are parameterized. The position and dimensions of the hole or opening can be adjusted by clicking on the relevant dimension line. The focus in case of multiple holes can be changed by clicking with the left mouse button in the hole contour. If you hold down the left mouse button and move the mouse, you can move the hole graphically. It checks whether holes overlap or cut through the outer contour.
- ***Plates*** – Plates can be supported elastically. The elastic soil is schematized as a number of springs that can support positive reactions only. The spring value of the spring will be calculated automatically.
- When you place the cursor near a displaced node on the Results tab, all the node results will be displayed in a pop-up window. (displacements, plate stresses, or plate forces)

improvements

General

- The primary forces of a beam that is spring-connected on one side and hinged-connected on the other side were not properly determined, resulting in erroneous moments in the hinged connection. This has been solved. Help desk #734
- If the beam is spring-connected on one side and hinged-connected on the other side, an incorrect spring value was printed in the output for the hinged connection. This has been solved. Help desk #735.
- The generation of beam loads from a surface load has been improved. The edge can now also consist of several beams.

version 5.00.03 (26-3-2020)

new

General

- ***Plates*** – When meshing the plate, the number of elements on the edge with one or more connecting plates is automatically equalized. The mesh therefore always connects with the mesh of a connecting slab or wall.
- ***Plates*** – Nodes that fall within the plate contour are incorporated into the mesh after the plate has been meshed. In this way, a point-shaped supported plate can now be inserted very easily. Or a steel construction supported by a plate.
- Nodes coordinates are now rounded to 0.1 mm.

improvements

General

- ***Plates*** – In a number of specific cases the program broke down when you drew a new plate contour. This problem has been solved. (help desk 731)
- ***Plates –*** The generated mesh nodes were also exported to XML. That doesn't make much sense. Those nodes are now skipped.
- XML export: The timber profiles have been added to the XML export.

version 5.00.02 (26-3-2020)

new

General

- ***Plates*** – It is possible to calculate plates. The plate can be entered very easily and quickly by drawing the contours as polyline. With the Escape-key you close the contour. At the plate edges you can create any supports (hinged, fully fixed or spring, etc.).
- ***Calculation*** – It is now possible to stop the analysis and calculation of the beam tests. Especially when calculating large models, this was highly desirable. The calculation monitor now also shows in which phase the calculation is located.
- ***Calculation*** – The calculation engine for solving the stiffness matrix has been optimized. As a result, the speed of calculation has greatly improved and (very) large models can be calculated.
- The stresses and/or forces in the plates are shown in color. Separate colour scales are used for the load cases, ULS combinations and SLS combinations.
- The summation of moments (sMx, sMy and sMz) relative to the zero point are now shown on the bottom right of the screen.
- The color scale for the "unity checks" has been adjusted. The color scale is displayed from 0 to the maximum U.C. instead of 0 to 1.00. This makes

it easier to see which bar is normative and also what the reserve is. (help desk #705)
- The colour scale for the displacements has been adjusted. Now 3 color scales are used. One for the load cases, ULS combinations and SLS combinations.
- Surface loads were always drawn. It is now possible to use layers also with surface loads. With this you can make the surface loads visible or not. (help desk #711)
- The gravitational acceleration g can be specified. The standard value is 9.81 m/s². (To generate the own weight bar loads, 10 m/s² was taken into account. This has now been improved).
- Loads have already been "scaled". Now also the node reactions are drawn "scaled". A large or small reaction force is more noticeable, which makes it even easier to check all reaction forces.
- Load combinations can now be excluded from the calculation by unchecking the relevant combination in the table combination.

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. (help desk #702)

improvements

General
- An error message occurred when the deflection of the load case was requested. (help desk 2019112501)
- Tab results: When you only have the unity check on, the text of the load combination is omitted.
- Tab results: When displaying envelopes, the text of the current combination was displayed. This has been changed. Now the text "Envelop ULS" or "Envelop SLS" is displayed.
- Local axis system of a support did not work. This has been resolved.
- 3 backup files were saved in the project folder. Now 1 backup file is saved. (help desk #706)
- The views in the output were saved as separate PNG files in the project folder. On request, those PNG files will now be deleted again after creating the output. (help desk #707)
- The outer and inner radius of a box section could not be adjusted. This has been solved. (help desk 2019111501)
- Rounding radius of a T-shape could not be adjusted. This has been solved. (help desk #717)
- In a number of cases, it was reported that edge bars were missing in the case of surface loads. This has been improved. (help desk #712)

- The order of the layers in the output did not match the order of the layers entered. This has been improved. (help desk #714)
- The graphical removal of the last node turned out to be impossible. This has been improved.
- Placing beam loads on multiple selected beams was already possible. When another bar load was added on a single bar, the beam load was wrongly also placed on the previously selected beams. This has been solved.
- When working in a large model for a long time, a memory problem (lack of internal memory) could occur. This problem has been solved.

Steel

- 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. (help desk 2019112502)
- When you turn off the steel check control, the beams were calculated anyway. This has been improved. (help desk 2020020503)
- The check of the steel stresses according to formula (6.1) proved to be incorrect in the case of gammaM0 > 1,00. This has been solved. (help desk 2020020602).
- In the case of a composition of different steel grades, the steel grade of the first profile was taken for testing. This has been improved. We are now investigating the lowest steel grade and this is used for the testing. This is a conservative approach. (help desk #703)
- 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) (help desk 2019111201)

Timber

- In the timber check, the general remark on imperfections referred to the steel testing. This has been improved.