



Release notes

Performance

Windows ® 7 and Vista Compatibility

Faster Analysis – Thanks to a new computation algorithm, analysis runs much faster than even before.

Powerful Video Engine – Implemented OpenGL and DirectX engines result perfect 3D rendering.

Work Environment

Live Rotation – Without interrupting work process, any view can be set quickly and simply by mouse.

Additional 3D View Modes – Hidden Line and Shade with Edges are available as well Wireframe and Shade.

Transparency – Transparency of elements helps to find the position of hidden/overlapped elements.

Hidden/Shown Elements – New display technique lets you browse among hidden and shown elements.

Information Pop-up – Moving the cursor over an object its main properties appear in a tool-tip.

New Units – More unit types are available for length, force, mass, stress and displacement.

“Select All” – “Ctrl + A” selection is working inside dialogs, tables and on the Drawing area.

Data Safety – Time save copies are generated by projects and given time period, and a locked copy guarantees the protection of the currently used work file.

Structural Data Input

New Support Symbols – More expressive symbol-system makes motion and rotation directions more clear and differentiates the Single supports from the Group supports.

Quick Definition of Edge Connections – The default type (rigid or hinged) of boundary connections can be set for surface elements (plates, walls and panels).

Non-Linear Support and Connection Behaviors – Compression and tension rigidity can be set separately by motion and rotation components to model for example the “uplift” behavior in 2D/3D structures.

“Infinite” Rigidity – The value of “infinite” rigidity can be set individually by support/connection types and components.

Fictitious Bar Stiffness – The stiffness of a fictitious bar can be set separately by the main directions.

Truss Member – New 3D member carrying axial forces only is presented to model bars with limited compression and tension-only bars like cables, pull rods, diagonal braces etc.

Timber Bar Profiles - Typical timber profiles are available in the Section library to give an initial input of the new Timber Bar Design.

Timber Materials - Three material groups (C, D and GL) with different grades and “application data” are available for the new Timber Design.

Timber Panel Element – Quick definition tool is offered to model prefabricated timber plates/walls for the new Timber Panel Design.

Reduction Values for RC Design – Quick definition of application data for concrete materials is available according to Eurocode 2.

Material Database Export/Import – Customized material database can be shared between projects/users.

Project Materials and Profiles – Materials and profiles used in the current project are separately listed from the other available materials and sections.

Analysis (3D Structure)

Cracked-Section Analysis – Analysis of 2D and 3D structures can be done based on the cracked state of RC surface and bar elements.

Non-Linear Calculation – Non-linear behavior of supports, connections and truss members can be considered in 3D structure analysis.

Continued on next page -->



Release notes

--> Continued from previous page

Design

Design Load – All design and checking calculation works with both load combinations and load groups.

Timber Design – Timber bar and panel designer is available as a new design module.

From Predesign till Final Design – A new interface covers the whole steel, RC and timber design process by starting with automatic predesign and then manual design of the domain structural components and ending with the final design of the complete structure.

Buckling Length Factors – Proposed beta factors are available according to the end conditions of bars.

Design Groups – Set of bars/surface elements can be assigned to a group, and the members in one group will be designed together.

Auto Design – Based on utilization check, a fast and automatic algorithm finds the most suitable design parameters (cross-section, applied reinforcement etc.) of single elements/design groups by design category (steel, RC and timber).

Manual Design – Rapid “on-the-fly” fine tuning of design parameters can be done by elements/design groups according to utilization check. In RC design, Manual design completes the applied reinforcement editing task for both surface and bar elements.

Detailed Result – Results with background calculation formulas, figures and tables can be displayed by single elements or design groups. Quick navigation is powered with zooming details.

RC Design

Bar Reinforcement – Automatic longitudinal bar and stirrup distribution is presented in concrete bar design. Additional reinforcement can be defined manually by design groups or elements. Cracked-section analysis of bars is available for the applied reinforcement in all modules that designs concrete bars/beams/columns.

Surface Reinforcement – According to required reinforcement and utilization, meshes can be designed automatically and fine-tuned manually in concrete shell design. Plate, wall and 3D shell structures can be checked based on the applied reinforcement (cracked-section analysis).

Punching Reinforcement – Design of bars and stirrups arranged in different shapes can be done both automatically and manually inside slabs around their column connections.

Steel Design

Steel Bar – Fast auto design and check are available to find the most suitable steel bar profiles. All section shapes and classes – including class 4 profiles (slender sections) - can be checked for utilization.

Steel Bar-Shell Model – When modeling steel bars with 3D steel shell components, an auto design finds the proper thickness of the shell elements. Also fast check can be run for defined shell thicknesses.

Timber Design

Load-Duration Classes – Duration of loads can be taken into account in the design.

Timber Bar – Both Auto and Manual design are available to find and check the most suitable bar profiles.

Timber Panel – The opposite panel thickness can be detected with the fast Auto design.

Results

Quantity Estimation – A fast process collects all structural elements and sorts them by materials, material qualities, identifiers, main geometric parameters (e.g. profiles), quantities etc. or defined priorities. Summary tables can be attached to the final documentation.

Determinative Load Combination – Load combination that gives the maximum result for load groups is displayed in result tables.