StatiCAD | Structural Analysis and Design Software

Hybrid V9.2.5.0

Release Notes

28 July 2026

AI Assistant, BIM/IFC, steel and light-gauge steel member design, finite element dynamic analysis, DXF/AutoCAD-compatible technical drawing, automation, and precast updates

1 AI Assistant

StatiCAD can be used with an AI assistant embedded directly in the program. You can direct your model in natural language — by typing or speaking via microphone — and the assistant performs member creation, analysis, view, and import/export operations on your behalf.

- **New:** The assistant now adapts to the interface language. If the program is used in Turkish, the assistant replies in Turkish; in other languages it starts in English and switches to whatever language the user types.
- Support for voice commands (push-to-talk).
- Upload files and images for the assistant to evaluate.
- **Edition:** The AI Assistant is available in the Hybrid and Ultimate editions.

2 BIM / IFC — SAP2000 structural model import NEW

IFC (IFC2x3) import has been extended. StatiCAD can now convert not only physical building models but also SAP2000 structural analysis models into StatiCAD members.

- Frame members → imported as columns and beams (fully vertical members are treated as columns).
- Surface members → imported as walls, quadrilateral slabs, and curved / triangular / polygonal shell (finite element) slabs.
- Unit differences (meters/centimeters) and character-encoding differences in the file are handled automatically.
- Corner and elevation placement of shell slabs, along with meshing alignment, has been improved; the model is created in the correct position in the 3D view.
- **Edition:** IFC import and export are available in the Hybrid and Ultimate editions.

3 Terminal and API automation expanded

The command interface that lets AI tools operate the program automatically has been expanded.

- **New:** Analysis and reporting commands can now also be used by AI agents operating through the API.
- Terminal-based automation support with tools such as Claude Code and Codex.
- Terminal commands added from now on will also be automatically usable on the API side.

4 Precast structures — variable-section analysis and parametric sections NEW

The first development package for modeling precast buildings has been released: variable-section roof/truss beams are now solved within the program's own finite element analysis, and parametric precast section templates similar to those in SAP2000 have been added.

- **New:** Beams defined with a variable (nonprismatic) section are now solved with their actual stiffness in the program's native 3D frame finite element analysis (without subdividing the member, using stiffness integrated along the section — the same approach as SAP2000). End releases, rigid end zones, and intermediate section forces are supported exactly as before.
- **Parametric templates:** Precast I (Bulb Tee), Precast U, Precast Box, trapezoidal, and box/tube sections are defined with SAP2000-style dimension input (B1–B4, D1–D7, T1, T2, C1, etc.) and a live preview, then saved to the library as polygonal sections.
- **Parametric editing:** Saved precast sections reopen with their dimensions intact; parameters are stored in the section library and in project files (fully compatible with older files).
- Filtering by precast template type has been added to the section library.

5 Steel structure design — beam and column design NEW

Design of steel structural members according to TBDY 2018 and ÇYTHYE (Turkish Code for the Design, Calculation and Construction of Steel Structures) has been added to the program. Fabricated profiles can be assigned to beams and columns; section forces are taken from the program's finite element solution to perform capacity checks, and calculation reports are produced in Turkish and English.

- **Profile library:** a library of 1008 ready-made fabricated section profiles (IPE, IPN, HEA/HEB/HEM, HD/HL/HP, UPN, UPE, L angle, T, RHS/SHS box, CHS tube). Hot-rolled and cold-formed (CC/ZZ/Sigma purlin) profiles are listed in separate categories; selection is made via search and a live section preview.
- **Section & material:** Profiles are assigned from the beam and column (vertical tie-beam) dialogs; steel-grade (S235/S275/S355, etc.) material assignment can be made through the section, similar to SAP2000. The member's design type is set with the "Steel" option in the form.
- **Steel beam design:** Chapter 9 flexure (yielding, inelastic/elastic lateral-torsional buckling; Lp/Lr limits), Chapter 10 shear, and Chapter 11 M-N-V combined-effect checks. The lateral bracing length is calculated automatically, as with timber, or can be entered manually; the deflection check ($f \le L/K$, with K selectable per member) is recorded in the report.
- **Steel column design:** Chapter 8 compressive strength (flexural buckling about x/y and torsional buckling), Table 5.1A slender-section detection, a $KL/r > 200$ warning, and Chapter 11.3 axial force + biaxial flexure combined-effect equations.
- **Reporting:** A "Steel Member Reports" group (Steel Beam and Steel Column calculations) has been added to the report selection tree. Reports show section properties, capacities, the governing limit state, and the load combination each demand comes from; reports can be generated in Turkish and English.
- **3D view:** Beams and columns with assigned profiles are drawn in the 3D window with their actual section geometry (I/H, etc.); section images are automatically added to reports.
- **Validation and scope:** Flexure, buckling, shear, and deflection results have been verified against hand calculations. Capacity calculations for cold-formed profiles will be provided through a separate "Light-Gauge Steel" module.

6 Light-gauge steel structures — panel stud design NEW

The first development package for member design in light-gauge steel (cold-formed steel) panel structures has been released: studs in light-gauge steel shear-wall panels are designed and reported using the Effective Width Method (EGM) based on TS EN 1993-1-3 (Turkish national standard, Eurocode 3 Part 1-3 — cold-formed steel design), with forces derived from panel analysis.

- **Sigma profile library:** 90 ready-made Sigma sections (S160–S400) manufactured in Turkey. Inputs for stud profile selection, stud spacing, and edge stud count have been added to the "Light-Gauge Steel Panel" tab of the wall dialog.
- **Stud check:** axial compression check for edge and intermediate studs; demands are derived from the panel's vertical load (tributary area) and, for edge studs, from overturning. Local, distortional, and flexural–torsional buckling checks are performed with the EGM; on sheathed faces, the weak-axis buckling length is taken as the stud spacing.
- **Panel shear inputs:** left/right panel sheathing and unit shear strength (vc) inputs are now saved with the project and used in the TBDY 2018 panel shear capacity calculation.
- **Analysis warnings:** Studs with insufficient capacity or no profile selected are shown in the warning list at the end of the analysis.
- **Reporting:** A "Light-Gauge Steel Stud Design (EGM)" report has been added to the "Light-Gauge Steel Member Reports" group in the report selection tree: panel name and profile by story, envelope forces (with combination name), edge/intermediate stud demand–capacity ratios, and calculation assumptions (Turkish/English).
- **Scope note:** Out-of-plane wind bending is not included in the stud check in this version (noted in the report); stud material is calculated assuming S350GD. Older projects open without requiring changes.

7 Finite element dynamic analysis — rigid-diaphragm modal analysis and shell (slab) fixes

The rigid-diaphragm reduction in modal (dynamic) analysis performed with finite element matrices has been rewritten, and important fixes directly affecting results have been made in shell (slab) elements. Results were verified comparatively against SAP2000.

- **New:** Full kinematic-constraint method for rigid-diaphragm modal analysis: in-plane story degrees of freedom (UX, UY, RZ) are linked to the story master node with an exact rigid-body relation, while out-of-plane degrees of freedom are statically condensed (the same approach as SAP2000).
- **Fix:** Resolved an error that caused the shell (slab) element's membrane stiffness matrix to be built with incorrect content instead of being recalculated after previous analysis steps. This fix directly improves the static and modal results of models with slabs.
- **Fix:** Corrected the Poisson (cross) term in the shell material matrix being derived from the wrong component; the membrane and bending material matrices are now symmetric and correctly valued.
- **Fix:** The total mass (ΣMn) value in the dynamic analysis report could appear different from the seismic mass in rigid-diaphragm analysis; it is now reported on the same basis as the total effective modal mass.
- The "Include slabs in rigid-diaphragm modal analysis" option makes it possible to include slab stiffness in the modal solution; the restraining effect of slabs on column end rotations is reflected in the periods.
- **Validation:** Rigid-diaphragm modal periods and mass participation ratios for a test structure with slabs were compared against SAP2000; agreement on the order of 1–3% was obtained for translational modes.

8 Technical drawing — DXF export and AutoCAD compatibility

DXF exchange in the technical drawing module has been overhauled from the ground up. Generated DXF files are now in the modern AutoCAD 2000 (AC1015) format; they have been opened and verified with current CAD software, including AutoCAD 2025.

- **New:** Hatches are exported to DXF. Hatch boundaries, pattern name, pattern lines, scale, and angle are written in full; the drawing appears with the same hatching in AutoCAD.
- **New:** Lineweights are written to and read from DXF. Member and layer lineweights are transferred one-to-one with AutoCAD in both directions; a unit error that caused lineweights to import 10 times thinner has also been fixed.
- **New:** TrueType fonts (Arial, Times New Roman, etc.) are written to and read from DXF along with bold/italic information; text is displayed in AutoCAD with the same font and with Turkish characters shown correctly.
- **Fix:** Resolved DXF files produced by StatiCAD opening to a blank screen in AutoCAD (requiring Zoom Extents); drawings now open with the correct view. Also fixed a text-style definition that caused the letter "İ" to appear as "?" in technical drawing titles.
- **Validation:** All export features have been tested with round-trip export/import tests and independent DXF-inspection tools, and verified with AutoCAD 2025 on real project files.

9 Quantity takeoff — column and shear-wall reinforcement takeoff NEW

Column and shear-wall reinforcement takeoff has been added to the quantity takeoff report. Reinforcement lengths and counts are generated from the same calculation as the actual layout in the detailing drawings; weight tables by diameter (Ø8–Ø32) per story and a bill of quantities per member are reported.

- **New:** Column reinforcement takeoff: longitudinal bars (including lap-splice/starter-bar lengths) and stirrups (including confinement-zone tie spacing) are counted and measured from the layout in the column detailing drawing.
- In addition to reinforced-concrete columns, vertical tie-beams in masonry structures are also included in the column reinforcement takeoff.
- **New:** Shear-wall reinforcement takeoff: boundary-element vertical bars (left/right), web vertical and horizontal bars, boundary-element stirrups, and cross-ties are reported with diameter tables per story and a bill of quantities.
- **Scope note:** Column cross-tie reinforcement is not included in the takeoff in this version; the counts of shear-wall boundary-element stirrups and cross-ties are calculated using approximate formulas.

10 Edition and demo behavior

- **AI and BIM/IFC:** active in the Hybrid and Ultimate editions.
- **Demo version:** the AI assistant and IFC import are available; IFC export is disabled, and files are saved in demo format.

11 Shear-wall/wall shell meshing — block mesh, intersection nodes, and internal-force display

Solving shear walls/walls with shell finite elements has been significantly improved in this version: a new regular block meshing method, mandatory meshing at intersection points, the elimination of solution errors caused by very thin wall strips, and completion of 3D color display of wall internal forces.

- **New:** Wall/shear-wall shell mesh method option (Project General Settings → Mesh): “Thin strip (classic)” and “Block (regular subdivision)”. The block method divides wall panels into regular quadrilateral cells according to the given mesh width and height, reducing the formation of thin, elongated elements. The choice is saved with the project file.
- **New:** "Mandatory mesh at intersection and beam-connection points" option: wall-to-wall intersections and beam-connection projections are forced in as mesh nodes, improving load transfer between intersecting members.
- **New:** Internal forces in the wall strips above doors, above windows, and below windows are also color-coded in the 3D view; wall result display has been made complete.

12 Finite element analysis acceleration — multi-core processing and factor-once solution NEW

The total duration of finite element analysis has been significantly reduced through structural improvements in the solver and model-setup stages. Fast solving is enabled by selecting “Multi-core (fast)” from the “FEM Processor Usage” option on the Project General Settings → Member Forces tab; the classic single-core workflow is retained as the default.

- **New:** Factor-once / solve-many solution: for load cases sharing the same stiffness matrix, matrix factorization is performed only once; subsequent load cases are solved with back-substitution only. Conditions that change stiffness (effective section, diaphragm, damage loading) are automatically detected, and the matrix is rebuilt when necessary.
- **New:** Parallel load-case solving: load cases within the same matrix class are solved simultaneously across multiple processor cores (one core is left free for the system).
- **New:** Model-setup skipping: for successive load cases within the same matrix class, rebuilding the finite element model is skipped and only the loads are updated. Wind and vertical seismic load cases are excluded from skipping, erring on the safe side.
- **New:** The ±5% accidental eccentricity effect can also be applied using story torsional moments equivalent to the mass-shifting method; this method is used automatically in fast solving. Comparative tests confirmed that both methods give the same results.
- **Measurement:** In a sample masonry project, total analysis time dropped from about 195 s to about 135 s (a 31% reduction); the individual time for accelerated load cases fell from about 13 s to about 3 s. The gain is larger in models with shell walls: a solution taking 18–20 s for the first load case drops to about 1.5 s for subsequent load cases.
- **Validation:** Fast and classic solving were compared on the same test projects; internal forces, the list of inadequate sections, and design results are identical.

13 Fixes

- Fixed a character-encoding-related import issue when reading IFC files.
- Fixed shell slab corner points, elevation references, and meshing alignment in structural IFC import.
- The right elevation value for quadrilateral slabs is now assigned correctly together with the left elevation.
- Fixed an issue in the 3D view where one side face of rectangular columns and vertical tie-beams was not drawn (showing the interior of the member); beam end faces have also been closed.