# SAP2000 v27.0.0 Release Notes



## *Notice Date: 30-January-2026*

This document lists changes made to SAP2000 since v26.3.0, released 01-July-2025. Items marked with an asterisk (*) in the first column are more significant.

### Analysis

***Enhancements Implemented***

| * | Ticket | Description |
|---|---|---|
| * | 9866 | The solid element has been enhanced to better model shapes other than the cuboid (hexahedron), i.e., shapes that have fewer than 8 nodes. The shapes include the 4-node tetrahedron, 5-node pyramid, 6-node wedge, and 7-node transition. Previously these shapes could be modeled by duplicating node numbers, creating a degenerate cuboid shape. The new natural shapes are more efficient because they use fewer integration points and smaller stiffness matrices. Note that incompatible modes only provide higher-order behavior for the rectilinear dimensions of the pyramid, wedge, transition, and cuboid shapes. The tetrahedron shape is not affected by incompatible modes. Note that the volume of the natural transition shape is 5/6 of the cuboid, whereas for the degenerate transition shape it was 3/4 of the cuboid. Solid properties can now be defined to use standard or quadratic (incompatible-mode) formulations with natural shapes, or legacy versions of these two options that use degenerate shapes. Older models opened in the new version will use the legacy versions with degenerate shapes so that results do not change. New models will default to natural shapes. Note that the use of the cuboid element is recommended whenever possible for best accuracy, and results for this shape are not affected by this enhancement. |

### API

***Enhancements Implemented***

| * | Ticket | Description |
|---|---|---|
| | 11267 | An enhancement was implemented for the Application Programming Interface (API) to update the functions for ACI 318-11, 14, and 19 to include missing preferences and overwrites, and to add the missing implementations of the GetOverwrite and GetPreference functions for ACI 318-14 and 19. |

### Data Files

***Enhancements Implemented***

| * | Ticket | Description |
|---|---|---|
| | 11695 | An enhancement has been implemented that adds the material library to the program for the Korean region. The library has materials from the following standards 1) Concrete materials per KBC2016, KS19 standards 2) Steel materials per KBC2016, KS22 standards 3) Tendon materials per KBC2016, KS22 standards 4) Rebar materials per KBC2016 standard. |
| | 11697 | An enhancement has been implemented that adds a new steel section database for sections from Korea per their KS21 standard. |

### Database Tables

***Enhancements Implemented***

| * | Ticket | Description |
|---|---|---|
| | 11675 | An enhancement was implemented to add the field "UseMaxFreq?" to the "Case - Direct History 1 - General" database table, corresponding to the ability to enable or disable the Maximum Considered Modal Frequency checkbox in the Direct Integration Damping form (Define menu > Load Cases). |

## Design – Concrete Frame

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| * | 10688 | An enhancement has been implemented to add concrete frame design based on the European EN 1992-1-1:2023 code. |
| * | 11663 | An enhancement has been made to add the ACI 318-25 concrete frame design code. |
| * | 11672 | An enhancement has been made to add the Vietnamese TCVN 5574:2018 concrete frame design. |
| | 11720 | An enhancement is made in ACI 318-19 column shear rebar design where Avmin is not enforced in column design when shear force, Vu without Avmin is <= phi*Vc/2. |
| | 11721 | An enhancement was added for ACI 318-19 beam shear design where axial forces are now accounted in computing beam shear capacity. Axial compression can be ignored in computing beam shear capacity when option "Ignore Beneficial Pu for Beam Design?" is set to Yes in the concrete frame design preferences. Axial tension is always considered in computing beam shear capacity. |

## Design – Steel Frame

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| * | 10548 | An enhancement has been made to implement the design for stability using the Direct Analysis Method for CSA S16-19 steel frame design. In particular, Section 8.4 'Design for Structure Stability' of the CSA S16-19 is implemented in combination with Annex O.2 'Stability effects in elastic analysis' to amplify the required design moments. The effective length method is also included. |
| | 11425 | An enhancement has been made for Chinese 2010 steel frame design in which the design value of fu for nonstandard materials is now taken as specified by the user in the material properties irrespective of the thickness of the material. Earlier the values were taken by interpolation from the tables of standard materials based on fyk and thickness values. No change has been made for standard materials where the values come from tables that are thickness dependent. |
| * | 11705 | An enhancement has been made to add the Canadian CSA S16-24 steel frame design. |
| | 11758 | An enhancement was made to allow modal and buckling load cases to be included in design loading combinations. |

## Documentation

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| | 10723 | The example problems in the documentation have been updated to be consistent with the latest user interface and to utilize the newer XML section property libraries. This was a documentation update only. |

## Drafting and Editing

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| * | 678 | An enhancement has been made to add the ability to store a user-defined mesh, called User Mesh, in a solid object. The analysis model for a solid object will be generated using the User Mesh, if it has been assigned. A solid object with a User Mesh can be added directly by importing from a DXF file (Draw menu > Import User Mesh as Solid). The mesh in the DXF file should be defined as lines denoting the edges of the solid elements. The mesh should be composed of valid solid elements for analysis in SAP2000. The mesh should also be a contiguous region. Multiple regions should be imported separately. Additionally, the following menu commands have been added: - A solid object with a user mesh can be divided into separate objects based on the mesh through Edit menu > Edit Solids > Explode User Mesh to Solids - The User Mesh assigned to a solid object can be deleted through Edit menu > Edit Solids > Remove User Mesh from Solid Object |
| * | 9863 | An enhancement has been made to add the ability to store a user-defined mesh, called User Mesh, in an area object. An area object with a User Mesh can be added directly by importing from a DXF file (Draw menu > Import User Mesh as Area) or by merging multiple existing area objects (Edit menu > Edit Areas > Merge Areas to Create User Mesh). The Area elements in the DXF file should be defined as DXF 3DFace. The User mesh currently is limited to elements in the same plane. The mesh should be composed of valid triangular or quadrilateral elements for analysis in SAP2000. The mesh should also be a contiguous region. Multiple regions should be imported separately. The following Edit menu commands have been added: - The existing auto-mesh of an area object can be saved as User Mesh to lock the mesh from being changed through Edit menu > Edit Areas > Convert Current Mesh to User Mesh - An area object with a user mesh can be divided into separate objects based on the mesh through Edit menu > Edit Areas > Explode User Mesh to Areas - The User Mesh assigned to an area object can be deleted through Edit menu > Edit Areas > Remove User Mesh from Area Object The analysis model for an area object will be generated using the User Mesh if enabled in the User Mesh Option groupbox of the Automatic Area Mesh form (Assign menu>Area>Automatic Area Mesh). This option is enabled by default, but can be manually disabled by the user. |
| | 11831 | A modeling geometry tool feature has been added as a new menu command: Edit menu > Edit Solids > Create Solids using Selected Lines. When multiple lines which form a 4-, 5-, 6-, or 8-node solid is selected, this tool will create the solid object formed by those lines. |
| | 11835 | A modeling geometry tool feature has been added as a new menu command: Edit menu > Edit Points > Create Points at Line-Area Intersections. When one or more line objects and 3- or 4-point area objects are selected, this tool will create points objects at the intersections of the selected lines and areas. |
| | 11840 | A modeling geometry tool feature has been added as a new menu command: Edit menu > Edit Lines > Create Lines at Area-Area Intersections. When multiple 4-point area objects are selected, this tool will create line objects at the intersections of the selected areas. If planar area objects with more than 4 points are included in the selection, the tool will create line objects at their intersection with the selected 4-point area objects. |

## External Import and Export

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| * | 11261 | An enhancement has been added to import OBJ files into SAP2000 (File menu > Import > OBJ. obj File). When importing, the program creates groups and materials based on the names defined in the OBJ file. Spline and Bezier surfaces are not currently supported. |
| * | 11678 | An enhancement has been added to export a SAP2000 model as an OBJ file (File menu > Export > OBJ. obj File). The exported OBJ file will contain the Point, Line and Area objects from the model with their material and section assignments. |
| | 11787 | An enhancement in DXF file import in SAP2000 Section Designer now supports importing polygons and polylines that are not initially closed by using close command. The feature ignores the last point if it is identical or nearly coincident to the first point. |

## Graphics

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| | 11842 | The DirectX graphics (View menu > DirectX View Settings) has been enhanced with options for shell edge detection and edge shading. |

## Installation and Licensing

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| * | 11513 | The version number has been changed to v27.0.0 for a new major release. |
| * | 11644 | The standalone, network, and cloud key licensing options have been removed from SAP2000 v27. All licenses will now use cloud sign-in licensing which is more flexible and secure. |

## Loading

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| | 11525 | An incident has been resolved that addresses drifts in the velocity and displacement time histories, obtained from time history analyses, when a time history matched to a response spectrum was used for the time history analyses. Base line correction was applied to the matched time histories as part of the matching process to prevent the drifts. |

## Structural Model

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| | 11337 | A change was made to allow seismic weight used in auto-lateral seismic load patterns to be calculated based on the default mass source when no load case uses the load pattern. This may be useful when the auto-lateral seismic load is used only for scaling the response spectrum or other dynamic seismic load cases. Earlier the reported seismic weight was calculated from the element material mass specified and any applied mass and not based on any mass source specifications. |
| * | 11342 | An enhancement has been made to the solid element to include the option of nonlinear material behavior, specified using the "Use Nonlinear Behavior" checkbox in the Solid Property Data form (Define menu > Section Properties > Solid Properties) for the Standard and Legacy Standard formulation types. The solid nonlinear behavior uses the Coupled Nonlinear Data from the material property (Define menu > Material Properties) that is available for isotropic materials. Currently, only the Von Mises Plasticity is available for use in solids and can be applied to steel and other non-concrete materials. Additional nonlinear 3D material models will be added as future enhancements. |
| | 11568 | An enhancement has been implemented that now allows the conversion of load combinations to nonlinear load cases to include large displacements as an option. Previously the generated load cases were always generated with just the P-delta option. |
| | 11898 | An enhancement has been made to the General Divide Tool (General Meshing) for area divide (Edit menu > Edit Areas > Divide Areas) and auto-meshing (Assign menu > Area > Automatic Area Mesh) to allow the existing quadrilateral-only meshing routine to proceed for more complex cases. In cases where the quadrilateral-only meshing routine cannot process the mesh, the program will automatically switch to an alternative meshing routine which allows triangular elements. |

## User Interface

### *Enhancements Implemented*

| * | Ticket | Description |
|---|---|---|
| | 9424 | An enhancement has been made in the user interface to sort lists within the user interface. The first phase of this work includes lists for load patterns, load cases, load combinations, material properties, frame section properties, area section properties, solid properties, tendon properties, cable properties, hinge properties, link properties, and groups. Other lists will be handled in a future release. |
| | 11730 | An issue has been resolved where deleting objects in large models could sometimes take an unusually long time when the Model Explorer was active. The Model Explorer update process has been optimized to address this performance problem. Additionally, an unrelated issue has been fixed in which object labels within the Model Explorer were not updating when modified through the right-click “Information” form in the GUI. This issue did not occur when labels were changed in the information form using the “Modify/Show” option accessed by right-clicking the object label directly in the Model Explorer. Both of these issues were user interface related and did not affect analysis results in any way. |
| | 11745 | An incident has been resolved where text elements in the application did not properly resize when the Windows 11 system text size (Accessibility → Text Size) was set above the default value. In affected versions, increasing the system text size caused labels, buttons, and other UI controls in some of the forms to display clipped or truncated text, and in some cases controls did not resize to accommodate larger font metrics. Majority of the forms in the application were not affected by issue. The update improves the readability and accessibility for users who rely on larger text settings. |

## Analysis

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11536 | An incident was resolved where individual tendons and cables modeled with straight frame elements that were added in a Guide Structure operation of a staged-construction load case incorrectly contributed their full stiffness to the structure, instead of a negligible stiffness as intended. This did not affect those objects added as a part of a group. Cables modeled by catenary cable elements always contribute full stiffness and should not be used as guide elements. |
| * | 11722 | An incident was resolved where load patterns of type seastate applied via the 'Load Objects If Added' operation in a staged construction load case acted on all objects that were added so far rather than objects that were added in that stage. |

## API

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11601 | API documentation has been updated to explain that, with the conversion of the API to a .NET Standard 2.0 AnyCPU dynamic link library (DLL), certain late binding VBA calls that may have worked in earlier versions of the API will no longer work. |
| | 11901 | An incident was resolved wherein frame section assignments, when set through the API, could be reset when the analysis was rerun. This only happened if the new assigned section was different from the design section. The design section was overwriting the analysis section. |
| | 11904 | An incident was resolved where calling cAnalyze.RunAnalysis() API function immediately after calling cOAPI.ApplicationStart() API function failed to run the analysis and the model remained unlocked. Adding a delay of several seconds (enough for the license to be acquired) after calling cOAPI.ApplicationStart() prevented this issue. |

## Data Files

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11560 | An incident was resolved where importing a model from text file that contained a NBCC 2020 wind load pattern definition would generate an error and incorrectly refer to NBCC 2015 instead. |

## Database Tables

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11303 | An incident has been resolved concerning the display of concrete frame design database tables, where the database failed to report rebar areas for variable beam sections. This was a display issue only and the design remained unaffected. |
| | 11594 | An incident was resolved where the automatic table output that is generated after running the analysis was incorrectly handling filenames that contained a period (.). In this case the output was still generated, but the filenames were not always as requested by the user. |
| | 11639 | An incident has been resolved for the NTC 2008 and NTC 2018 steel frame designs in which exporting and re-importing the model text file mistakenly showed warning messages regarding Psi(C1), C2, & C3 parameters. The design results were not affected. |
| | 11865 | An incident was resolved for the ASCE 7-22 auto wind load pattern where Case 4 (i.e. "4x(+e1, +e2)", "4x(-e1, -e2)", "4y(+e1, +e2)", "4y(-e1, -e2)") was not recognized when the ASCECase value was changed using interactive database editing. |

| * | Ticket | Description |
|---|---|---|
| | 11881 | An incident was resolved where the Case - Static 1 - Load Assignments database table was unable to be populated when the model contained a response spectrum load case with a diaphragm eccentricity specified. This was only an issue with filling the data in the table and did not affect results. |
| | 11896 | An incident was resolved in the interactive database when editing the "Case - Modal 2 - Load Assignments - Eigen" and/or "Case - Modal 3 - Load Assignments - Ritz" tables. If the model contained both Eigen and Ritz modal cases, when the table for one was edited, the load assignments in the other would be cleared out upon applying changes. |

## Design – Cold Formed Frame

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11771 | An incident has been resolved for the AISI-16 cold-formed steel design in which the pattern live load factor was always taken as 0.75 and used in the design. It is now available in the design preferences. |

## Design – Concrete Frame

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11618 | An incident has been resolved in the concrete frame design codes "Eurocode 2-2004" and "Italian NTC 2008" in which torsion design of concrete beams per Eurocode EN 1992-1-1:2004 section 6.3 has been reinterpreted and re-implemented based on the procedure as described in the updated manual. The main difference between the new implementation and the old one is in the application of factor 2 and subtraction of Tcon from TEd as described in the equation of Vt on page 3-43 of the previous manual. The new method is consistent with section 6.4 of the book "Designer's Guide to EN1992-1-1 and EN1992-1-2--- Eurocode 2: Design of Concrete Structures. General rules and rules for buildings and structural fire design" by R. S. Narayan and A Beeby. This new implementation will lead to a change in results in the stirrup areas. |
| | 11668 | An incident has been resolved in the concrete frame design codes "Eurocode 2-2004" with German National Annex, in which the minimum shear rebar area as calculated using DIN EN 1992-1-1/NA:2011-01 section NDP Zu 9.2.2(5) was improperly implemented. Previously, the expression used for Asw,min/s was Asw,min/s = 0.16 sqrt(fctm) / fyk, which was wrong and which is corrected as Asw,min/s = 0.16 (fctm / fyk). The program and documentation are updated. |
| | 11669 | An incident has been resolved in the concrete frame design codes "Eurocode 2-2004" and "Italian NTC 2008" in which design shear rebar area and the minimum shear rebar area were calculated for concrete beams per Eurocode EN 1992-1-1:2004 sections EC2 6.2.3(3) and EC2 9.2.2(5). The minimum was enforced when VEd < VRd,c. However, the program did not enforce the minimum required shear rebar area when VEd > VRdc. The program now enforces the minimum required shear rebar areas for both cases. |
| | 11710 | An incident has been resolved for Eurocode 2-2004, EN 1992-1-1:2023, and NTC 2008 concrete frame design in which the unit [mm] for the crack width limit overwrite was there unnecessarily. The value of the crack width limit overwrite has also been corrected in the design. |
| | 11822 | An incident has been resolved for Chinese 2018 concrete frame design, in which the program sometimes failed to pick the maximum span-moment when the member was framed with other members at multiple intermediate points, braced in the major direction at those framing points, and the maximum moment fell on one of those intermediate brace points. This was a tolerance issue and could potentially affect the calculation of the beta_m and beta_t factors. Previously, the end moments and shears of the braced segments were obtained from the nodal values. If the point is missed because of a tolerance issue, the peak might have been missed. Now those are obtained using interpolation. |

## Design – Concrete Shell

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11537 | An incident has been resolved for concrete shell design in which the design results were shown incorrectly in the contour plot and envelope table when multiple design requests were defined. The right-click design report was correct and unaffected. |

## Design – Steel Frame

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11350 | An incident has been resolved in the steel frame design code "Chinese 2018" in which the program was checking the stability equation GB50017 8.2.1-4, even when the Direct Analysis method was used. When N exceeded 1.25*Ney, the program reported an error message stating that "1.25 N / Ney > 1.0: Column is unstable (GB50017 8.2.1-4)". This check does not apply to the Direct Analysis Method. The program no longer checks this equation when the Analysis Method is the Direct Analysis Method. The program was conservative. |
| | 11515 | An incident has been resolved for the design of pipe and circular sections using EN 1993-1-1:2022 steel frame design in which the combined D/C ratio previously was not calculated using the square root of sum of squares (SRSS) of the bending moment components. The SRSS of the moment components is now applied for pipe and circular sections. |
| | 11564 | An incident has been resolved for the CSA S16-19 steel frame design in which the effective area of the angle section was determined incorrectly. Previously, the element width for angle section was calculated using the maximum width-to-thickness ratio of 200/sqrt(Fy). It is now corrected to 250/sqrt(Fy). |
| | 11566 | An incident has been resolved for EN 1993-1-1:2022 in which section designer sections were incorrectly classified as Class 4. They are now classified as Class 3. |
| | 11580 | An incident has been resolved in the AISC 360-22 steel frame design in which the compactness limits Lambda_s (slender), Lambda_r (non-compact), and Lambda_p (compact) were correct, whereas the limits Lambda_md (moderately ductile) and Lambda_hd (highly ductile) were not accurate. The latter two limits were taken from ANSI/AISC 341-16, rather than ANSI/AISC 341-22. The documentation has been updated. |
| | 11728 | An incident has been resolved for the EN 1993-1-1:2022 steel frame design where an error occurred when designing slender sections. These sections are now not designed and a message is shown in the design report. The design manual is also updated to indicate that the design of slender sections is not currently implemented. |
| | 11843 | An incident has been resolved for EN 1993-1-1:2022 steel frame design in which the classification of the section for bending about Z-Z axis in combination with axial force was incorrect. |
| | 11887 | An incident has been resolved in AISC 360-22 code, in which the automatically generated load combination had the wrong load factor for the roof live load when both the roof live and notional load, related to the roof live load were in the same combination. In this case, the roof live load factor was using 1.0 instead of 1.6. The load factor for the associated notional load, 1.6, was used, which was correct. |

## Documentation

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11736 | An incident has been resolved for the AISI-16 cold-formed steel design in which the design code reference for members with holes was corrected in the design manual. The changes include equations for Fcre in Section 3.4.3.1.2.2, sigma_ex and sigma_ey in Section 3.4.3.1.2.6, sigma_ey and sigma_t in Section 3.4.4.1.1, Fcre in Section 3.4.4.1.2, Fcre and sigma_ey in Section 3.4.4.1.4, Fcre in Section 3.4.4.1.5. |

## Graphics

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11523 | An incident was resolved, ensuring the frame is displayed correctly when showing deformed shape with extrusion using DirectX graphics. |
| | 11584 | An incident was resolved where transparency specification for frame member deformed shape plots was not correctly imposed. This was a display issue only and no results were affected. |
| | 11621 | An incident was resolved to ensure area loads are displayed as arrows in the correct direction when using a coordinate system different from the global system. |
| | 11821 | The incident was resolved, enabling the use of black as a fill color in force diagrams for DirectX graphics. |

## Loading

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11404 | An incident was resolved where the frame deflected shape shown on the Diagram Form obtained by right clicking on the frame may not have been correct when the joints associated with the frame had non-default local axes specified. No other results were affected and this was only a display issue. |
| | 11719 | An incident was resolved for ASCE 7-22 auto seismic load case where seismic load results were not available when method for computing seismic coefficient was based on Method 1 and building Period > Time Period, Tmax. |

## Results Display and Output

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 11567 | An incident was resolved where displaying the Shell Stresses or Strains (Display menu > Show Forces/Stresses > Shells) in a model with nonlinear layered shell elements which have hinge layers may result in the program experiencing an abnormal termination error. This issue was a display issue only and did not affect analysis results. |
| | 11588 | An incident was resolved where the joint reactions displayed with the tabulated option were always absoluted. This only affected this display in v26.3.0. The results in the tables were correct. |
| | 11779 | An incident was resolved where a force-controlled hinge that lost load carrying capacity after exceeding the maximum allowed force was displayed in the deformed shape with a dot color corresponding to the "B to C" hinge state (green) instead of the ">E" hinge state (red). The issue only occurred for the displaced shape display and did not affect the hinge plots or other results. |

## User Interface

### *Incidents Resolved*

| * | Ticket | Description |
|---|---|---|
| | 9176 | An incident was resolved to correct the following two user interface issues on the Mander Concrete Stress-Strain Data form: (1) When switching the Mander Data Source option, the confinement bar inputs on the right side of the form could sometimes be for a different Mander Type than the one selected. (2) When Mander Type was Confined Rectangular Core and the Confinement Bar Property option was used to choose a rebar material, the computed stress-strain curve plot was using the previous user input Confinement Bar Yield Stress value instead of the fy value from the selected confinement bar material property. |
| | 11551 | An incident was resolved where the tabulated joint reactions in the MX, MY, and MZ directions were not correct in the graphical view when the units were something other than the default value. |

| * | Ticket | Description |
|---|---|---|
| | 11640 | An incident was resolved where an error condition could occur when showing the analysis results with solid objects that were further meshed. Also, the Show Only command on a solid property in the model explorer did not work as expected. Theses were user interface issues only. |
| | 11648 | An incident was resolved in the Frequency Dependent Link Property Data form where the unit label displayed for the damping component of the translation/rotation degrees of freedom (DOFs) was incorrect. The values displayed and used in analysis were correct. |
| | 11724 | An incident was resolved where an error could occur when modifying certain definitions or properties from the model explorer tree. Accessing the definition or property via the menu worked as expected. |
| | 11885 | An incident has been resolved on the ASCE 7-22 auto seismic load pattern definition form where the labels for S1 and Sd1 were not quite correct. This was a minor user interface issue only and did not affect loading or results. |