

SAP2000® Version 21.2.0 Release Notes

© Copyright Computers and Structures, Inc., 2019

Notice Date: 2019-11-13

This file lists all changes made to SAP2000 since the previous version. **Most changes do not affect most users.** Items marked with an asterisk (*) in the first column of the tables below are more significant.

The reference number for each change below is now the development Ticket rather than support Incident which was used in previous Release Notes. Emails sent when an Incident is released will now indicate this Ticket number as well.

Changes from v21.1.0 (Released 2019-08-22)

Analysis

Enhancements Implemented

*	Ticket #	Description
*	2760	An enhancement has been made to reduce the amount of memory used for large framed structures loaded with open structure wind loading, enabling larger models and more loaded members.
	2972	An enhancement has been implemented to allow cancelling of a single load case when the "Run Load Cases in Parallel" option is turned on. The new Cancel button on the Analysis Monitor form is for terminating an individual load case whose details are currently being displayed on the monitor, while the Cancel button on the Main form is for terminating all running & scheduled load cases.
*	3097	An enhancement has been implemented to more clearly display and save messages generated during analysis runs. These are the same messages that appear in the analysis LOG file, but provided in a more concise format. Analysis messages are categorized as information, warnings, and errors. After the analysis is run, messages are automatically displayed as text if they are any warnings or errors in the latest run. This same text display is available any time using the command Analysis > Messages. The messages are also available in tabular format under table Analysis Results > Run Information > Analysis Messages. Messages are cumulative with subsequent runs until the model is unlocked, at which time the messages are deleted. Each message includes its type, message text, associated load case or stiffness case, operation being performed, date-time stamp, parallel run tag, run serial number, and machine name. The run serial number counts subsequent runs before the model is unlocked. The parallel run tag indicates which thread was used when analyses are run in parallel during the same analysis run.

API

Enhancements Implemented

*	Ticket #	Description
	2365	An enhancement was implemented in the Application Programming Interface (API) to provide new functions for handling rebar size definitions.

Database Tables

Enhancements Implemented

*	Ticket #	Description
	2875	An enhancement has been implemented to allow the tables for design preferences and design overwrites to be modified using the interactive database editor when the model is locked.

Design – Concrete Frame

Enhancements Implemented

*	Ticket #	Description
	2709	An enhancement has been made to concrete column design based on Eurocode 2-2004. The program now performs nine (9) permutations including analysis moments, analysis moments plus imperfection moments (4 permutations), analysis moments plus imperfection and second-order moments (4 permutations.) This will handle the case of transverse lateral load applied in mid-span of the columns. The design also supports section designer sections with unsymmetrical section and reinforcement, and non-prismatic elements.
*	2986	An enhancement has been made to concrete column design based on Italian NTC 2008. The program now performs nine (9) permutations including analysis moments, analysis moments plus imperfection moments (4 permutations), analysis moments plus imperfection and second-order moments (4 permutations.) The second-order analysis methods including the Nominal Curvature (specified by NTC 2008) and Nominal Stiffness (as an additional choice for users) have been updated to fully calculate all parameters properly according to the NTC 2008 design code. The improvement will also handle the case of transverse lateral load applied in mid-span of the columns. The design also supports section designer sections with unsymmetrical section and reinforcement, and non-prismatic elements.
	3106	An enhancement has been made to concrete frame design per the Russian SP 63.13330.2012 code in which the program now allows an additional input parameter for the reliability factor, Gamma_n, in the preferences form. The reliability factor is a factor used to obtain the design seismic force depending on the functional use of the structure. It is considered the level of responsibility for the structure. For essential structures, it is more than 1.0, for temporary structures it may be less than 1.0. Its default value is taken as 1.0. However, the user can change it to any positive value they want. By increasing the criticality safety factor Gamma_n, the design force values are practically increased by multiplying by that factor.
	3107	An enhancement has been made to concrete frame design per the Russian code SP 63.13330.2012 in which the program now allows input of two additional parameters for seismic factor in the preferences: m_tr,flexure and m_tr,shear. The parameter m_tr,flexure is applied to enhance the steel and concrete design strengths Rb and Rs when the design load combination contains an earthquake load case. It is typically equal to 1.2 per Table 6 of SP 14.13330.2014 and used for column NMM (PMM) design, beam flexure design, and beam torsion design. The parameter m_tr,shear is applied to enhance the steel and concrete design strengths Rb and Rs when the design load combination contains an earthquake load case. It is typically equal to 1.0 per Table 6 of SP 14.13330.2014 and used for column shear design and beam shear design. In addition, the values of the relative neutral axis depth, XiR, is reduced for beams by a reduction factor based on seismicity when a seismic load is present in the load combination. The reduction factor is equal to 0.85 for Seismic Intensity 7, 0.7 for Seismic Intensity 8, and 0.5 for Seismic Intensity 9. It is not reduced even if the load combination is seismic when the structure is declared Non-Seismic.
	3176	An enhancement has been made to the Russian concrete frame design code SP 63.13330.2012 in which Rbt,long and Rbt,short are no longer modified by the factors Gamma_b2, Gamma_b3, Gamma_b4, and Gamma_b5. However, Rb,long and Rb,short are still modified by the factors Gamma_b2, Gamma_b3, Gamma_b4, and Gamma_b5 as before. This change affects the shear design.

* Ticket #	Description
3177	An enhancement has been made to the Russian concrete frame design code SP 63.13330.2012 in which the default values of the "Live Load Duration Factor" and "Snow Load Duration Factor" are now set to 0.35 and 0.5, respectively. Previously they were both set to 1.0. This change affects the definition of load combinations involving live loads and snow loads as long term and short-term loading, which affects the value of Gamma_b1, and eventually affects the allowable concrete compression strengths.

Design – Steel Frame Enhancements Implemented

* Ticket #	Description
1446	An enhancement has been made to the Eurocode 3-2005, Italian NTC 2008, and Italian NTC 2018 steel frame design codes, in which the effective moment of inertia of Class 4 singly-symmetric I-shape sections is now calculated.
3109	An enhancement has been made to the steel frame design code "Chinese 2018" in which the gamma_RE(S) is now taken as 0.8 when the beam is designated as a Flexo-Compression member. It is taken as 1.0 when the beam is not designated as a Flexo-Compression member. Its default value was 1.0 irrespective of whether the beam's designation was a Flexo-Compression member or not.

Installation and Licensing Enhancements Implemented

* Ticket #	Description
2755	The version number has been changed to v21.2.0 for a new intermediate release.

Results Display and Output Enhancements Implemented

* Ticket #	Description
2468	An enhancement has been made to display load combination results on a step-by-step basis for Linear Add load combinations of load cases that may contain one or more stepped load cases. Both on-screen display and tabulated results are now available. Max/min results were always available. Design is not affected as it already had the option for step-by-step design.
2687	An enhancement has been made to output nonlinear energy components (nonlinear hysteretic damping and nonlinear viscous damping) per group for nonlinear load cases. The nonlinear energy is reported separately for model objects (internal element energy) and support elements (support element energy). The energy output is available in the "Element Nonlinear Energy By Group" table and "Energy by Group" type function in using the Display > Plot Functions command.

**Analysis
Incidents Resolved**

*	Ticket #	Description
	2748	<p>An incident was resolved that addressed several minor issues related to running load cases in parallel:</p> <ol style="list-style-type: none"> 1. Load cases inadvertently continued from/used the stiffness of load cases that were "not finished" (e.g. failed to converge) within the same analysis session. 2. Certain analysis files could be left behind after unlocking the model. 3. Occasionally some of the scheduled moving load cases failed to start. A second attempt at running them was usually successful. 4. In Equation Solver Options form, when switching from serial back to parallel mode, the form always defaulted to Fixed Number of Cores option regardless of the previous setting. 5. In Analysis Monitor form, monitor tabs from a previous parallel run session remained even after unlocking and re-running the analysis, or after switching to serial mode. <p>None of these issues affected the results obtained from analysis.</p>
*	2791	<p>An incident was resolved where reversing frame connectivity did not immediately update frame local axes based on the new configuration. Results were affected only if the analysis was run immediately after the reverse connectivity operation. Closing and opening the model, or any other edit or assign operation on the affected frame objects caused frame local axes to be correctly recalculated.</p>
*	2945	<p>An incident was resolved where, for nonlinear static, staged-construction, direct-integration load cases, and sequences of such load cases, the frame member forces and stresses used for display and design could have been incorrect for a specific frame member when the following conditions were met:</p> <ol style="list-style-type: none"> 1. The load case (or sequence of load cases) contained more than one load pattern that applies loads directly to the frame member, including at least one load pattern with self-weight loads. 2. A load pattern containing self-weight load (A) was applied after another load pattern containing frame loads (B); in other words, A was listed after B in the list of applied loads in the load case definition (or sequence). 3. The load patterns A and B had different scale factors. <p>Frame members that did not meet the above conditions, such as those without frame loads assigned to the load patterns used in the load case sequence, were not affected. Load case sequences without self-weight applied were not affected. Load case sequences having only one self-weight load pattern applied and with that load pattern being listed first in the load case definition were not affected. Note that this error did not affect how the frame loads were transferred to the structure, and therefore all other analysis results were correct (displacements, reactions, forces and stresses in other objects). Nonlinear behavior, including frame hinges and P-delta, was not affected. Only the reported forces and stresses within the affected frame members themselves were in error, including the forces used for frame design of those members. Because self-weight is most commonly applied first, most models were not affected by this error. Linear load cases were not affected, even if they used the stiffness from a nonlinear load case. This error affected SAP2000 versions 20.2.0 to 21.1.0.</p>

* Ticket #	Description
* 3042	<p>An incident was resolved where, for nonlinear static, staged-construction, direct-integration load cases, the frame member forces and stresses used for display and design could have been incorrect for a specific frame member when the following conditions were met:</p> <ol style="list-style-type: none"> 1. Results were requested for multiple load cases at the same time, including situations where one or more requested load combinations referenced multiple load cases. 2. Among all the requested load cases, only one of these was a nonlinear static, staged-construction, or direct-integration time-history load case, and only a single step was requested from that load case. 3. Among all the requested load cases, at least two of these were linear load cases (including modal or response-spectrum), at least two of these load cases used the stiffnesses from different nonlinear load cases (or zero initial conditions), and at least one of these was a linear static, linear multistep static, or modal time-history load case. 4. The affected frame object had loads assigned to it as part of the single nonlinear load case. This could include self-weight. <p>This was not common. When this error occurred, the frame response reported in the affected object could be incorrect at all stations along the length of the object except at the start (I end). For frame objects that were discretized into multiple elements for analysis, the results would be correct at the start of each element and deviate along the length of the individual elements. This deviation in response, when present, was due to using the wrong element load for equilibrium calculations. This error would be most likely to affect table results when multiple load cases or load combinations were requested, and frame design when the load combinations used satisfy the conditions listed above. Note that this error did not affect how the frame loads were applied to the structure, and therefore all other analysis results were correct (displacements, reactions, forces and stresses in other objects).</p>

API

Incidents Resolved

* Ticket #	Description
2904	An incident was resolved for the Application Programming Interface (API) where the functions cLineElm.GetLoadDeformation and cLineElm.GetLoadDistributed did not return results when using the cross-product API CSIAPIv1.dll. They worked correctly for the product-specific API SAP2000v1.dll.

Data Files

Incidents Resolved

* Ticket #	Description
3058	An incident was resolved where opening an old model while another model with results was already open would cause the results of the current model to be deleted if the prompt to save current changes was accepted.

Database Tables

Incidents Resolved

* Ticket #	Description
2876	An incident was resolved where the table "Frame Hinge Assigns 09 - Hinge Overwrites" was showing the value of "LinNegStiff" equal to 10 if no hinge overwrites had been assigned to a frame object. The values should instead have been shown as 0.1, which is the default. This was a display issue only and did not affect analysis results. Assigned values were displayed correctly. Note that values larger than 1.0 are not allowed.
2905	An incident was resolved where the concrete load combination type Crack Width was not recognized in an import from the load combination definitions table.

* Ticket #	Description
2915	An incident was resolved where Export to Excel from the table display form accessed using the Display menu > Show Tables command failed if the Program Control table was renamed in a user defined table name file.

Design – Concrete Frame

Incidents Resolved

* Ticket #	Description
2041	An incident has been resolved in the concrete frame design code Eurocode 2-2004 in which the second order effect calculation for column design moment (MEd) was incorrect when there was an in-span load in the column. The design could have been unconservative when there is an in-span lateral load on the column. The design was okay when there was no in-span lateral load on the column.
2827	An incident has been resolved in concrete frame design per Eurocode 2-2004 which was causing an error condition for a model with both concrete and steel members. This problem was introduced in v18.1.0.
2867	An incident has been resolved in the concrete frame design code "ACI 318-14" in which the torsion design of concrete beams may be under-reinforced when T_u falls between the threshold limit and the cracking value ($T_{th} < T_u < T_{cr}$) as in this case the required torsion rebar (A_t/s and A_l) was not calculated per section 22.7.6.1 and only the minimum rebar was provided per section 9.6.4.2. The calculation of rebar areas A_t/s and A_l was correct for $T_u < T_{th}$ and for $T_u > T_{cr}$.
3053	An incident has been resolved for steel frame design in which the design of an EBF type of frame could not be performed in SAP2000 v21 and later.
3116	An incident was resolved for the TS 500-2000 concrete frame design code where Zone 1 was not available in the design preferences.

Design – Steel Frame

Incidents Resolved

* Ticket #	Description
1977	An incident was resolved in the Italian NTC 2008 and NTC 2018 steel frame design codes in which the average moment M_{m_Ed} was calculated incorrectly. The moment at the design station was being used instead of the average moment along the length of the member. M_{m_Ed} is used in NTC Eq. C4.2.32 to check the demand/capacity ratio. Previously, the results tended to be overconservative if the moment at the design station was larger than the average but could be underconservative if the maximum moments were not considered.
2944	An incident was resolved in the Canadian steel frame design codes CSA S16-09 and CSA S16-14, in which the section classification for tee, angle, and double-angle sections had an error in the slenderness for the stem (tee) and the vertical leg (angle and double-angle). Previously the flange or horizontal leg b/t ratio was being used while checking for the stem or vertical leg slenderness of these sections.
3105	An incident has been resolved in steel frame design codes Eurocode 3-2005 and Italian NTC 2018 where the right-click design detail information was not able to be displayed for equal-legged angle shapes. No results were affected. The error only affected v21.1.0.
3110	An incident has been resolved in the Chinese 2018 steel frame design code in which the ϕ_b factors were not matching perfectly for members which had a linear moment diagram. The β_b factors were not overwritten by the equation given in Item 10 of GB50017-2017 App C Table C.0.1 correctly.

* Ticket #	Description
3111	An incident has been resolved in the Chinese 2018 steel frame design code where the K factors were not matching perfectly for columns which were connected to beams which either had a fixed far-end or a hinged far-end or to a column which had a fixed or hinged support condition. Both sway and non-sway moment frames are affected. Similarly, the K factors were affected for sway frames of all codes for columns which were connected to beams which either had a fixed far-end or a hinged far-end or to a column which had a fixed or hinged support condition.

Documentation Incidents Resolved

* Ticket #	Description
3187	An incident was resolved where Equation 4-7 of the Material Time-Dependent Properties Technical Notes was incorrect. The equation has been corrected in the documentation. This was a documentation issue and did not affect the results.

External Import and Export Incidents Resolved

* Ticket #	Description
1244	An incident was resolved to correctly process IGES character literals that were too long to fit on a single IGES line. In addition, error messages pointing out syntax errors have been enhanced when an IGES file is imported.

Graphics Incidents Resolved

* Ticket #	Description
2311	An incident was resolved where the display of area Uniform to Frame loading values when in DirectX graphics mode was not showing the values on each area object. Classical graphics mode worked as expected. This was a graphics issue only and did not affect results.
2314	An incident was resolved that addressed several issues with the graphical display in DirectX mode: <ol style="list-style-type: none"> 1.) Solid face colors were not being displayed. 2.) Frame labels were not centered. 3.) Extruded shells with transparency were sometimes displaying internal triangle edges. 4.) Point and area selection colors sometimes changed when zooming. 5.) Bubble text colors could be incorrect. 6.) The cut line was not displayed while drawing section cuts. 7.) Area local-3 axis was sometimes reversed when drawing in plan and elevation. The local-3 axis should point toward the user when drawing counter-clockwise or using the draw-rectangle option in 2-D. 8.) The window zoom sometimes produced unexpected results when the grid-bubble size was excessively large. 9.) When scrolling for values within the frame force/moment/stress diagram in the right-click "Diagrams for Frame Object" form, the red dot showing the corresponding location on the structural model did not move correspondingly. 10.) The deformed shape display of shell objects was incorrect when the View Type was set to Offset or Extruded in the View Options form. 11.) The option to display area colors by section property in the View Options form did not fill all objects. 12.) The Print Graphics command did not show joint reaction values when they were displayed on the screen. 13.) Area joint offsets were not being considered when displaying extruded view. 14.) Joint pattern values were not being displayed. 15.) The display of frame distributed loads didn't show loading applied in two different

* Ticket #	Description
	<p>directions for a single load pattern. Only loading in one of the directions was displayed.</p> <p>16.) When displaying frame hinge assignments in a 2-D view, the text labels were not shown at the correct locations.</p> <p>17.) The multi-select option was not properly activated by using CTRL + left-mouse-click.</p> <p>18.) Snapping to a midpoint of another object when drawing an area object in a 3-D view could sometimes create the joint at an unexpected location.</p> <p>19.) Contour plots of area-object response (forces/moments/stresses) were sometimes incorrect when the joint-averaging option was used. If the value at three corners of an element were nearly equal, the value at the fourth corner would be plotted the same even if it should be different. The error was generally small, and no other results were affected.</p>
2773	An incident was resolved where animation of the deformed shape did not work in DirectX graphics mode on Windows 7 and Windows 8.1 machines after recent Windows updates, or on some Windows 10 machines with limited graphics memory. Now, when this issue is able to be detected, the software will generate a "static" animation that displays correctly but cannot be rotated, panned, or zoomed without regenerating the animation. This is similar to the animated deformed shape display in Classical Graphics (GDI+) mode. On capable Windows 10 machines, a "dynamic" animation will be generated in DirectX mode that can be rotated, panned, and zoomed without stopping to regenerate the animation. In some cases the issue may not be detected by the software, and the animation will only be available using Classical Graphics mode.
3029	An incident was resolved where drafting in DirectX mode did not snap exactly to gridlines when working in certain units. The error was extremely small, affecting accuracy in the sixth or seventh digit of the coordinate value.
3104	An incident was resolved where the display of loads on the analysis (element) model did not work when in DirectX graphics mode. Minor corrections were also made for the display of loads on the object model in DirectX. The display of loads on both the element and object model when using Classical Graphics mode was not affected.

Loading Incidents Resolved

* Ticket #	Description
2703	An incident was resolved for the Eurocode 8-2004 response spectrum function for the Singapore National Annex where selecting the Ground Type "S1" was causing a "Bad Function Data" error during analysis.
2706	An incident was resolved where having auto wind loads based on the Chinese code together with Response Spectrum loads with added eccentricity loads would cause an error condition in the program.
3032	An incident was resolved for the TSC-2018 auto seismic load pattern and response spectrum function where the Site Class and for Long-Period Transition Period values would revert to default values when the model was imported from a text file.
3141	An incident was resolved for the TSC-2018 auto seismic load pattern where the computed time period was limited to $1.0 * T_{pA}$ instead of $1.4 * T_{pA}$.

Results Display and Output Incidents Resolved

* Ticket #	Description
1116	An incident was resolved where the contour range bar was missing from the print graphics display when displaying soil pressure results with the classical graphics mode. This was a display issue only and did not affect results.
* 2781	An incident was resolved where right-click design details were unavailable on Windows 8.1 and Windows 10 after installing the latest Microsoft Windows updates.

* Ticket #	Description
* 2797	An incident was resolved where hinge states could not be retrieved for hinges in frame objects containing one fiber hinge and another hinge of any type. This was a display issue and the actual analysis results were not affected. Manually dividing the frame object to contain either only one fiber hinge or any number of non-fiber hinges prevented the issue.
2823	An incident was resolved where the hinge states reported for fiber hinges in the table "Frame Fiber Hinge States 01 - Overall Hinge" (Display > Show Tables > Analysis Results > Element Output > Frame Output) could be inconsistent with the states of the individual fibers contained in the hinge, and these hinge states could be changed incorrectly to "AtoB" if the model was unlocked and rerun. Additionally, the hinge states reported for each step in the table "Pushover Capacity Curve" (Display > Show Static Pushover Curve, then File > Display Table in the Pushover Curve form) could similarly be incorrect for the fiber hinges in the model. This issue only affected the Hinge State output and did not affect the force and displacement results. Non-fiber hinges were not affected. Note that the hinge state for a fiber hinge is expected to be determined from the contained fiber having the most extreme state (furthest from point A on its stress-strain curve) and any output time or load step.
* 3083	An incident was resolved where tabulated analysis results may have been displayed as zero for load combinations that contained one or more Linear Add type load combinations. When this issue occurred, the tabulated results for a Linear Add type load combination that contained another Linear Add type load combination could have been displayed as zero. This issue only occurred when the following conditions were met: 1. The model included multiple load combinations that each contained the same Linear Add type load combination (say "A") in its definition. 2. In the Choose Tables for Display form (Display menu > Show Tables), more than one of the containing load combinations were selected for output. 3. At least one of the selected load combinations containing combination "A" must have been of type Linear Add, and at least one must have been of a different type (e.g. Envelope). Any load combination containing affected load combinations could also be affected. This issue only affected the tables under the Analysis Results section. This issue did not affect visually displayed results or design results. This issue did not affect load combinations that were not nested. This issue was only present in SAP2000 version 21.1.0.
* 3163	An incident was resolved where the plotted deformed shape for mode-based load cases was incorrect at joints having local coordinate systems different from the default (global axes). The displacement values plotted and shown in local coordinates (U1, U2, ..., R3) were actually the values in global coordinates (UX, UY, ... RZ). This affected the plotted shape, the values displayed when the mouse was moved over a joint, the values shown when right-clicking on a joint, and videos made of the deformed shape. Similarly, the joint displacements reported in the two database tables "Joint Displacements" and "Joint Displacements - Absolute" were always the displacements in global coordinates rather than the expected joint local coordinates. Plot functions were not affected. This error affected modal, response-spectrum, and linear and nonlinear (FNA) modal time-history load cases, as well as load combinations containing these load cases. No other types of load cases were affected. All other response quantities (displacements, forces, stresses) and design results throughout the model were correct and unaffected. This reporting error only affected version 21.1.0 of SAP2000.

**Section Designer
Incidents Resolved**

* Ticket #	Description
2654	An incident was resolved where auto hinge definitions assigned to concrete columns with section designer sections would get corrupted when converted from FEMA 356 hinges to ASCE 41-13 hinges. The conversion would happen automatically when opening a model created in older versions of the program to the newer version that does not support FEMA 356 definitions. The corruption would cause an error condition and results were unavailable.

Structural Model
Incidents Resolved

*	Ticket #	Description
	2239 2985	An incident was resolved where in certain instances the replication of models that included link objects with advanced local axes could result in an error. When this occurred some of the link objects may not have been replicated.

User Interface
Incidents Resolved

*	Ticket #	Description
	2275	An incident was resolved where custom menu shortcut keys defined by the user were reset when the user interface language was changed.
	2775	An incident was resolved where changing the units in the linear link property definition form when the model was locked could cause values to not be shown. This was a user interface issue only and did not affect results.
	2897	An incident was resolved where the definition form for a nonlinear direct integration time history load case with collapse considered was not synchronized correctly when it was initially opened. The checkbox to display objects to remove was checked but that data was not shown. Now the form opens with the check box to display objects to remove unchecked.
	2899	An incident was resolved where sections inside a frame section auto select list could not be deleted from the model using the Frame Properties form until the auto select section was deleted and the model was saved.
	2914	An incident was resolved where an abnormal error occurred displaying individual fiber data for hinge results. This was a user interface issue only and did not affect analysis results.