SAP2000® Version 19.2.0
Release Notes

Notice Date: 2017-08-21

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

Changes from v19.1.1 (Released 2017-05-10)

User Interface
Enhancements Implemented

*	Incident	Description
	200411	Additional checks have been added when adding or modifying grid systems to ensure the name is not 'Local' or 'Global' which are reserved names in SAP2000.

Graphics
Enhancements Implemented

*	Incident	Description
	102579	An enhancement has been made to display auto generated loads on the analysis element model and in the right button click dialog.

Modeling
Enhancements Implemented

*	Incident	Description
*	201058	A new link property has been implemented to represent high-damping rubber bearings (isolators). These bearings represent coupled two-dimensional hysteretic damping in the shear degrees of freedom of the link, which is independent of axial force and rate of deformation. Parameters include control strains and strengths for one or more hysteresis loops, as well as a degradable elastic stiffness. These values can be determined from experiment or from the device manufacturer, such as Bridgestone or others. The mathematical formulation is based on N. Masaki, T. Mori, N. Murota, K. Kasai, "Validation of Hysteresis Model of Deformation-History Integral Type for High Damping Rubber Bearings," Paper 4583, *Proceedings of the 16th World Conference on Earthquake Engineering*, Santiago, Chile, 2017, and private communication with the authors.
*	201147	The assignment of insertion points (joint offsets) to cable objects has been removed. These offsets were of little practical use and could easily create instability in a model where the joints were pinned. A warning will be provided when opening a model from a previous version that has cable insertion points assigned, and they will be removed from the model. The command Assign > Cable > Insertion Point is removed. The associated database table Cable Insertion Point Assignments has been removed. The API (Application Programming Interface) functions SapModel.CableObj.SetInsertionPoint and GetInsertionPoint have been deprecated. SetInsertionPoint will have no effect on the model, and GetInsertionPoint will always return zeroes.

Section Designer

Enhancements Implemented

*	Incident	Description
	101396	Section Designer has been enhanced to allow further editing of the edges and points of polygon shapes. Polygon edges can be divided by right-clicking on an edge while using the Reshape tool. Two options are provided, similar to those available for line objects in the main graphical user interface: (1) Divide into Specified Number of Segments, and (2) Divide at Specified Distance from I-end of Edge. In addition, the form "Change Coordinates" that previously appeared when right-clicking on a polygon point has been renamed "Edit Selected Point", and an option to delete the point has been added.

Loading

Enhancements Implemented

*	Incident	Description
*	95479	An enhancement was implemented to add automated wind loading according to the Indian code IS:875 (Part 3) 2015 for Wind Loads.
*	202410	A new operation, Add Guide Structure, has been implemented for staged-construction analysis to improve deflection reporting for certain types of structures. Using this operation, frame, tendon, and homogeneous shell objects can be added as "guides" at the beginning of a staged construction analysis, to be replaced with the actual objects as the construction proceeds. A guide has the same geometry and connectivity as the actual object, but with reduced stiffness and no mass, weight, loading, or time-dependent behavior. Guide objects deflect with the portion of the structure that is already present, so that the reported deflections for a newly added object will include the deflections of the guide as caused by the previously existing portion of the structure. While the use of guides affects deflection reporting, it normally does not affect forces, moments, stresses, or design results except when large-displacement geometric nonlinearity is considered. Typical uses for guide structures include cantilever construction, tall and unsymmetrical buildings, and composite bridges where the concrete slab is cast-in-place on previously erected girders.

Analysis

Enhancements Implemented

*	Incident	Description
	102953	An enhancement was made to improve the speed of creating the analysis model when a large number of tendons were present.
*	103383	An enhancement has been made to allow additional modal damping in linear and nonlinear direct integration time history load cases. This feature uses the mode shapes and periods from a specified modal load case to calculate a modal damping matrix. This matrix is restricted to the shape of the stiffness matrix, meaning that modal damping does not couple elements that are not connected. The associated modal case must use the same mass source as the direct-integration load case, and must be run before the direct-integration load case that uses it. Modal damping parameters allow the damping ratio to be constant for all modes, interpolated by period or frequency, or determined based on a mass and stiffness proportional coefficient. Any modal damping specified as additional material damping will also be included in linear and nonlinear direct-integration time history load cases. When modal damping is specified in a nonlinear direct integration time history load case, more iterations may be necessary to reach equilibrium. Modal damping is in addition to any proportional damping that may be specified for the direct-integration load case. A small amount of stiffness proportional damping is recommended to control higher modes. The CSI Analysis Reference Manual has been updated for this topic.

*	Incident	Description
*	200664	The displacement control option for nonlinear static load cases has been enhanced to allow additional controlled displacements, either as joint degrees of freedom or generalized displacements, to be specified in the load case definition. When additional controlled displacements are defined, the most significant of the monitored displacement and the additional displacements will be used to determine the displacement in a load step. Only the monitored displacement will be used when plotting the Static Pushover Curve results. Using multiple controlled displacements may improve convergence behavior for models with degrading strength, localized deformation, or snap-back behavior.
	202197	Convergence behavior for nonlinear static displacement-control analysis has been improved for Newton-Raphson iteration. For affected models, this produces fewer iterations and/or less sub-stepping, resulting in faster run-times and/or fewer convergence failures. For some models, analysis results for nonlinear static displacement-control load cases may change from previous versions, particularly for models with poor convergence behavior or large step sizes. Such changes in results are expected to be within the specified convergence tolerance. Verification examples 1-029, 2-018 and 2-019 were updated to reflect the effect of this change. The validity of these verification examples was not affected.

Frame Design

Enhancements Implemented

*	Incident	Description
	78845	An enhancement has been made in steel frame design codes AISC 360-05 and AISC 360-10 in which the SRSS procedure was performed on the moment ratios, but SRSS procedure was not performed on the shear ratios when calculating the interaction equation per AISC Eqn. H3-6 for pipes. Now the SRSS procedure is performed on both the moment and shear ratios before the PMM ratio is calculated based on using AISC Eqn. H3-6 for pipes. This only affects the PMM ratio slightly and only if the member is subjected to torsion, shear, and flexure together.
	96827	An enhancement has been implemented for the AISC 360-10 code in which the following changes have been made: (a) The limit state of lateral-torsional buckling (LTB) for tees and double-angle sections about the axis of symmetry (2-2) is not normally considered, as typically the beam is not loaded about that axis. However, for certain cases, this axis becomes the major principal axis. Previously LTB was not considered about this axis. Now LTB for these two types of sections are considered about the axis of symmetry based on the recommendation given in the last paragraph of commentary section AISC C-F9. In this case the equation for Me as stated in Eqn C-F9-3 and the expressions for Mn in equations F10-2 and F10-3 are used. Previously this limit state was not considered. Note that the LTB about the nominal major axis bending (3-3) is always considered. (b) The limit state of flange local buckling (FLB) for bending about the symmetrical axis (2-2) of double-angle sections was based on Eqns. F6-2, F6-3, and F6-4 which is correct. However, the limiting values of b/t were used as 0.3*Sqrt(E/Fy) and 0.45*Sqrt (E/Fy) instead of 0.54*Sqrt(E/Fy) and 0.91*Sqrt (E/Fy) as Lambda_p and Lambda_r, respectively. This has been fixed. (c) The limit state of web local buckling (WLB) for bending about the major axis (3-3) was not considered. Now equations F9-8 through F9-11 are used to consider the WLB. The documentation has been updated in this regard.
	101767	An enhancement has been implemented in the concrete frame design code NZS 3101-06 in which the user can now overwrite the maximum aggregate size. This affects the shear rebar design by modifying the factor k_a. Previously the program assumed that the maximum aggregate size was 25mm for which k_a was equal to 1.0.
	101771	An enhancement has been made in the concrete frame design codes “BS 8110-97”, "Hong Kong CP 2013", and "Singapore CP 65:99" where the flexural rebar of a T-beam now uses basic principles of ultimate strength design with the rectangular stress block. The previous version used the code specified approximate equation as given in BS 8110-97 section 3.4.4.5. This was done to use the same basic rules for all design codes and all beam section shapes.

*	Incident	Description
	201572	An enhancement was made for BS 8110-1997 concrete frame design where frame shear design now accounts for the rebar shear strength fyv equal to 500 MPa as published in the Amendments to BS 8110-1997. Previously the shear rebar strength was limited to 460 MPa.

Results Display and Output

Enhancements Implemented

*	Incident	Description
	91142	An enhancement has been implemented to provide database tables for beam, column, and brace design forces for the design load combinations. The results can differ from regular frame forces as they account for other factors required during design. These results are also available via the API.
	103109	An enhancement has been implemented to add the section properties J and Cw to the CSA S16-09 and CSA S16-14 steel frame design detailed output.
	200228	An enhancement has been made to add displacement contours on faces of solid elements.

External Import/Export

Enhancements Implemented

*	Incident	Description
	100046	An enhancement has been made to the import of walls from Autodesk Revit projects. Some planar walls with complex outlines were previously not imported. All planar walls are now imported. This was previously implemented in v19.1.0 but inadvertently missed from the release notes.

Documentation

Enhancements Implemented

*	Incident	Description
	100974	A technical note was added for material time-dependent properties. This technical note discusses the concrete and steel tendon time-dependent behavior for the time dependent types currently available.

Application Programming Interface

Enhancements Implemented

*	Incident	Description
	100528	The API has been enhanced to allow MATLAB client applications to attach to a running instance of SAP2000. The documented MATLAB API example has been updated to demonstrate this new feature.

Miscellaneous

Enhancements Implemented

*	Incident	Description
	101822	The version number has been changed to v19.2.0 for a new intermediate update.

User Interface and Display

Incidents Resolved

*	Incident	Description
	100348	An incident was resolved where loads were not being displayed when using the command Display > Show Object Load Assignments > Load Case when the number of load patterns was large. In particular, the error occurred when the number of load patterns, n, in the model and the number of load patterns, m, assigned in any load case or stage was such that n^(m-1) > 2^32. This was a display issue only and no results were affected.
	100401	An incident was resolved where load patterns of type notional would not save the auto lateral load pattern option if it was set to None. The value always reverted to Auto.
	102425	An incident was resolved where the frame object length was not immediately updated when entering reshape mode, selecting a frame object, and then selecting and moving a point object at the end of the frame object.
	103574	An incident was resolved where an unconfined Mander stress-strain curve did not correctly drop to zero at the ultimate unconfined strain capacity and also the units were not correctly converted for the stress in the table shown with the form that displays the Mander stress-strain curve.
	103648	An incident was resolved to identify material stress-strain curves used in fiber hinges where the strains were not monotonically increasing. In such cases an error message was given during analysis. The software now forces the curves to have monotonically increasing strains.
	200410	An incident was resolved where copying/pasting data in the grid definition form would create an error condition if an empty string was pasted to a cell.
	201203	An incident was resolved where the multivalued options were disabled when a load combination was selected on the Display Solid Stresses form.
	202082	An incident was resolved where resizing the frame design details form would cause the section image to grow in size.

Graphics

Incidents Resolved

*	Incident	Description
	70381	An incident was resolved where the N-Patch Tessalation option under Options > Graphics Mode > Advanced Parameters for DirectX display has been removed as it was no longer applicable to the newer version.
	80955	An incident was resolved in which springs assigned to lines and areas were not shown when displaying the deformed shape of the model. They were shown when displaying the undeformed shape of the analysis model. This was a display issue only and did not affect results.
	85576	An incident was resolved where extra lines were sometimes visible in extruded views in custom coordinate systems if triangular area objects were present.
	89390	An incident was resolved where the display of the animated deformed shape in DirectX Graphics mode did not animate the symbols representing link objects. This was a graphical effect only and results were not affected.
	90518	An incident was resolved where the display of models with a large number of link properties would make screen operations very slow. This has been significantly improved.
	95713	An incident was resolved where a magenta colored dot was shown onscreen when displaying spring supports that were not coupled. The magenta colored dot showed the presence of coupled springs. This was due to a very tight tolerance being used to check for coupling of assembled springs at a joint. This has been relaxed.
	96331	An incident was resolved in which the display window would freeze if the deformed shape was animated and then a different display, such as undeformed shape, was requested without first stopping the animation. This was a display issue only and did not affect the results.

*	Incident	Description
	96593	An incident was resolved where viewing area loads to frame showed the loads in the wrong direction if the view was set to show frames in the offset location. They were shown correctly otherwise. This was a display issue only and no results were affected.
	96594	An incident was resolved in which the DirectX color contour legend was inverted when displaying solid object faces by color.
	98534	An incident was resolved where the display of extruded area objects was not correct in non-global coordinate systems. This was a display issue only and no results were affected.
	101687	An incident was resolved where the line springs and area springs would not show on the analysis model display. This was a display issue only and was inadvertently introduced in v19.1.0. No results were affected.
	102116	An incident was resolved where the extruded shape of a frame was not located correctly if the insertion point of the section was at the shear center and the shear center differed from the centroid. This was a display issue only. No results were affected.
	200748	An incident was resolved where the on-screen display of assembled distributed loads on frames would in some rare cases be incorrect. This was due to very tight tolerances being used in assembling the load for display. This has been improved. No results were affected.
	201204	An incident was resolved where the values shown for solid element stresses when hovering over the element were not for the exact location and did not correspond with the contour color displayed. This issue was only present in DirectX display mode and not in the Standard display mode. This only affected the value shown when hovering and did not affect any results or the contour colors displayed.

Modeling

Incidents Resolved

*	Incident	Description
	98791	An incident was resolved where the axial-force or shear-force values used to generate auto nonlinear hinges were not being fully updated after the analysis was run. This only affected hinges that were automatically generated and were based on ASCE 41-13 Tables 9-6, 10-7, 10-8 or Caltrans hinges and for which the axial or shear force was specified to be updated based on a load case or a load combination. Now, the hinge properties will be updated for the next time the model is unlocked and run again.
	98794	An incident was resolved for asolid elements where the following incorrect behavior was noticed: (1) The asolid property exposed the Property Modifier button whereas the Property Modifiers should have been restricted to "Shell" type properties only. (2) The self-weight of the asolid element was not being calculated correctly. (3) The material list did not include asolid type properties. (4) The asolid elements were not being plotted when deformed shape plots were showing. This affected v19.1.0 and v19.1.1 only.
	102930	An incident was resolved where the auto-generated hinges using ASCE 41 for concrete sections was determining the generated hinge rotation based on an incorrect V value. This issue occurred only if the database units of the model were different than lbs and inches. In cases where this issue occurred, analysis results were consistent with the generated hinge properties viewable in Define > Section Properties > Hinge Properties.

Section Designer

Incidents Resolved

*	Incident	Description
	202070	An incident was resolved in section designer where an error condition would occur after clicking on the "Show Interaction Surface" button when the concrete frame design code was set to SP 63.13330.2012.

Loading
Incidents Resolved

*	Incident	Description
*	202054	An incident was resolved where strain, temperature, or surface pressure area loads applied on a layered shell element during nonlinear analysis were not applied. This issue occurred in the case where strain, temperature, or surface pressure loads were not included in the first load pattern in the load case but were included in subsequent load patterns applied in the load case. This issue was not present if the load case contained only one load pattern and did not affect other types of area loads (e.g. gravity or uniform loading). This issue did not affect plane or asolid elements and did not affect shell element types other than the layered shell.

Analysis
Incidents Resolved

*	Incident	Description
	74529 101003	An incident was resolved where element loads on a shell, solid, or planar element and specified to act in a fixed coordinate direction (such as gravity) would rotate with the element after being applied during an analysis with large-displacement geometric nonlinearity. This issue only affected nonlinear static and nonlinear direct-integration time history load cases with the "Geometric Nonlinear Parameters" set to "P-Delta plus Large Displacements". When this issue occurred, the computed response was in equilibrium with the rotated load and the issue was reflected in the reported forces and base reactions. Loads specified to act in an element-local coordinate direction are expected to rotate with the element under large-displacements, and that has not changed. Frame and link elements were not affected. Asolid elements do not support large-displacement effects.
*	100793	An incident was resolved where shell (area) objects that are not subject to a Change Section operation in a staged-construction load case could be affected by a Change Section operation in another staged-construction load case that was analyzed earlier in the same run, even if the affected load case did not continue from the prior-run load case. This did not occur if the two load cases were analyzed in separate runs.
*	101220 101993	An incident was resolved where staged-construction load cases starting from zero initial conditions could be affected by the “Change Section” operation in another staged-construction load case that preceded it in the same analysis run. Consequently, this would also impact any other load case that used the affected staged-construction load case for initial conditions. This did not occur if the two staged-construction load cases were analyzed in separate runs. Only the stiffness of shell elements used for the staged-construction load cases were affected - no other type of element was involved.
	101968 103316	An incident was resolved where the analysis would terminate with an error if a target-force load was applied in a nonlinear staged-construction load case in any stage after the first stage. When this issue occurred, no results were available from the stage where the target-force load was applied, and subsequent stages would not be run. This issue affected SAP2000 v19.0.0 through v19.1.1.
	102115	An incident was resolved where analysis results were not available for the unusual case where a hinge was assigned at the end of a frame object with a non-zero end offset, and the frame object was meshed such that the resulting frame elements were small enough for the element at the end to be fully contained within the end offset. When this occurred, results could not be plotted and values were displayed as "NaN" or other non-numeric values.
	102636	An incident was resolved where models with asolid (axisymmetric solid) elements were often unable to converge to a solution when large-displacement geometric nonlinearity was requested in a nonlinear load case. Large-displacement geometric nonlinearity is not consistent with the formulation of the asolid element. Now, when a load case is specified to use large-displacement geometric nonlinearity, P-delta geometric nonlinearity will be used instead for the asolid elements. All other types of elements present in the model will still consider large-displacement effects when requested.

*	Incident	Description
	200082	An incident was resolved where the analysis was unable to run for large models that also had a very large number of load patterns of type Vehicle Live. When this occurred, results were not available. Reducing the number of these load patterns enabled the model to run. This has been resolved by making more efficient use of available memory. Larger models may require use of the 64-bit installation and the presence of sufficient physical memory (RAM) on the machine.
	200582	An incident was resolved where stresses were incorrectly reported for asolid elements where the local 1-direction of the element did not coincide with the radial direction of the element. Stresses from asolid elements where the local 1-direction coincides with the radial direction are correctly reported. When this issue occurred, the S11, S22, and S33 stress results may be affected. This issue does not affect the computed displacements and joint forces.
	202146	An incident was resolved where the stiffness and self-weight of a frame object could vary with discretization if the frame was assigned both a non-prismatic section property and significant joint offsets. This issue only affected frame objects where the joint offsets and/or cardinal point changed the length of the frame object compared to the distance between the two end joints. This included the case where axial joint offsets were specified, and the case where transverse joint offsets inclined the local 1-axis with respect to the line connecting the two joints. Only frame objects that were discretized due to auto-meshing assignments or hinge overwrite lengths were affected. When this issue occurred, the differences in stiffness and self-weight were small and generally insignificant. Small changes in results may occur for models that use many non-prismatic frame section properties with significant joint offsets. Note that the affected joint offsets were those specified as part of the insertion point assignment. End offset assignments were not affected by this issue.

Frame Design

Incidents Resolved

*	Incident	Description
	68088	An incident was resolved where in some rare cases steel design deflection checks for a frame member would consider it as a cantilever when the member was actually supported on both ends. This normally happened when the support on one end was very soft. The tolerances have been adjusted.
	70317	An incident was resolved for axial compression capacity, Cr, of steel frame design using the CSA S16-09 code where the value of Cr in certain conditions (when “n” was calculated as 2.24 or overwritten as greater than 1.34) was calculated conservatively. The power "n" which could be 1.0, 1.34, 2.24, or any overwritten value was used in calculating the axial compression capacity for flexural buckling per CSA 13.3.1 based on Euler buckling stress for major and minor axes bending (Fex and Fey). That was correct. The power "n" with a value of 1.34 was used in calculating axial compression capacity for the flexural, torsional, or flexural-torsional buckling per CSA 13.3.2 based on Euler buckling stress Fe (CSA 13.3.2). In the latter case for certain conditions (for any doubly symmetric shape) Fe was taken as the minimum of its values for two flexural modes (Fex and Fey) and one flexural-torsional buckling mode (Fez) (CSA 13.3.2(a)). The Cr calculated based on section CSA 13.3.2(a) with an n=1.34 was always smaller than that calculated based on CSA 13.3.1 when the calculated “n” was equal to 2.24 or overwritten as greater than 1.34. The current version of the program does not take Fe as the minimum of Fex, Fey, and Fez for the flexural-torsional buckling mode anymore. Rather it considers this mode as strictly flexural-torsional mode and takes Fe=Fez. This implementation makes sure that flexural mode uses the calculated “n” or its overwritten value whereas the flexural-torsional buckling mode uses n=1.34. The implementation for CSA S16-14 had similar problem regarding Fe as the minimum of Fex, Fey, and Fez which is fixed. The CSA S16-14 code however always used the value of “n” as either the calculated value or the overwritten value irrespective of the mode. Now both the codes use the same value of “n” and similar value of Fe for flexural-torsional buckling mode. Both codes have been enhanced to now report n, KL/r, Lambda, Cr, and other values for all modes.

*	Incident	Description
	98365	An incident has been resolved in Italian NTC 2008, Eurocode 3-2005, and Indian IS 800:2007 codes in which the displayed PMM interaction ratio of a member immediately after design was differing from the PMM ratio in the details after the right button click. This happened when the unbraced length ratio was specified to be more than 1. The program skipped the calculation of the equivalent moment as the actual span was not really known. Now the program calculates the equivalent moment based on full span of the member even though the braced length is even bigger than the full length of the member. In addition, this also fixes reporting of several Ncr values which were showing as "Infinite" in Italian NTC 2008. The latter issue was a display error only.
	101764	An incident has been resolved in the concrete frame design code Eurocode 2-2004 in which the design longitudinal rebar did not match with hand calculations because alpha_cc was taken as 1 when it is specified as 0.85 for certain countries. The countries affected include Finland, Singapore, and the United Kingdom.
	101770	An incident has been resolved for concrete frame design codes BS 8110-97, Hong Kong CP 2013, and Singapore CP 65-99 where the design minimum flexural rebar was not being enforced for the rare case of the design moment being exactly zero.
	101924	An incident was resolved where automatically generated design loading combinations would sometimes include load cases that were indicated by user to be excluded.
	102991	An incident was resolved for steel frame design codes AISC 360-10, AISC 360-05, and KBC 2009 where the torsion capacity phi*Tn for box shapes with differing thicknesses of the webs and the flanges was calculated based on the thickness of the flange instead of the minimum thickness. The resulting torsion capacity was incorrect when the thickness of the web was smaller than the thickness of the flanges. This could have affected the stress ratios if the interaction equation AISC H3-6 governed.
	103281	An incident was resolved for steel frame design in which some superfluous default design combinations may have been created if snow loads were present.
	103282	An incident has been resolved for steel frame design using Russian code SP 16.13330.2011 where there was a problem primarily affecting the documentation for calculation of the moment coefficients "m" and "m_ef" for the different boundary conditions: Fixed-Fixed, Hinged-Hinged, Fixed-Hinged, or Fixed-Free. The documentation has been corrected for members loaded with span loads and for members not loaded with span loads. The calculation of the moment coefficients "m" and "m_ef" should be different between section SP16 9.2.3 (to be used in calculation of "phi_e") and section SP16 9.2.6 (to be used in calculation of "c"). In addition, a minor change has been made in the software to more accurately determine the values of "m" for different boundary conditions. The effect of this change is generally very small.
	103489	An incident has been resolved in steel frame design codes AISC 360-05 and AISC 360-10 in which there was a sudden jump in the stress ratio when the member axial force switched between axial tension and axial compression by a tiny magnitude. This happens when the governing combination is AISC H1.3b, Eqn. H1-2 in compression for doubly symmetric sections with very small minor axis bending moment. In all cases the result was not changed between the failure and safe. A stress ratio less than one was always less than 1 irrespective of the stress ratio.
	103597	An incident was resolved for steel frame design using code AISC 360-10 in which the interaction equation AISC H1-2 was not implemented accurately. The value Mcx was taken as the available flexural strength considering all limit states, including lateral-torsional buckling, while considering the actual Cb value. However, it is now taken as the available lateral-torsional strength for strong-axis flexure determined in accordance with Chapter F using Cb = 1.0. In Equation H1-2, Cb Mcx may be larger than φb Mpx in LRFD or Mpx /Ωb in ASD. The yielding resistance of the beam-column is not imposed on Cb*Mcx separately. This happens when the governing interaction equation as given in AISC H1.3b, Eqn. H1-2 in compression for doubly symmetric sections with very small minor axis bending moment. The previous design results were conservative.

*	Incident	Description
	202315	An incident was resolved for Indian IS 456:2000 concrete frame design code where the minimum eccentricity moment in the minor direction could have been incorrect when the unbraced length of the column was different in the major and the minor directions. The unbraced length in the major direction was used for computing minimum eccentricity moment in both major and minor directions.

Results Display and Output

Incidents Resolved

*	Incident	Description
	80222 83198	An incident was resolved where display of text on virtual work displays was not correctly located in some cases. This happened when the virtual work display was requested after a deformed shape display.
	80927	An incident was resolved in which printing the plot function graphics would sometimes result in unexpected black boxes drawn onto the output.
	83919	An incident was resolved in which the steel frame design ratios were not displayed as colors when running the design of a model imported from a STAAD file. This was a display issue only and the design results were correct.
	99102	An incident was resolved in which the graphical display of link force values could be incorrect if the results were first displayed in classical graphics and then the display was switched to DirectX graphics mode or a zoom and refresh were performed in DirectX mode. Requesting the display of link forces again when already in DirectX graphics mode would correct the values. This was a display issue only.
	103022 200752	An incident was resolved for plane elements where deformed shape and other results plots were not showing. This affected v19.1.0 and v19.1.1 only.

Database Tables

Incidents Resolved

*	Incident	Description
	103358	An incident was resolved where warnings could be generated when reading a table names XML file that was generated with default values. This did not affect results.

Data Files

Incidents Resolved

*	Incident	Description
	200545	An incident was resolved where the starting frame section for an auto-select list would not be correctly initialized as the median section in the list if the model was imported from a text file. This was not a problem if the auto-select list was defined in the GUI or through the API.
	201472	An incident was resolved where importing a model from *.s2k, *.xls, or *.mdb that contained an IBC 2006, 2009, or 2012 response spectrum function would default the S1 spectral acceleration value to 0.869 instead of using the value from the file being imported. When this happened the analysis results would be based on the response spectrum function using the default S1 value.

Application Programming Interface

Incidents Resolved

*	Incident	Description
	101381	An incident has been resolved in which the cHelper.CreateObjectProgID() API function failed to start the 64bit program when called from a 32bit API client running on a 64bit OS. All other combinations (64bit program & 64bit API client, 64bit OS; 32bit program & 64bit API client, 64bit OS; 32bit program & 32bit API client, 64bit OS; 32bit program & 32bit API client, 32bit OS) worked as expected.

External Import/Export

Incidents Resolved

*	Incident	Description
	99177 99872 101454 102616 103615 202023 202066	An incident was resolved wherein attempting to export models to Perform could result in a "Run-time error ". This issue was limited to v19.0.0 through v19.1.1.

Documentation

Incidents Resolved

*	Incident	Description
	98850	The concrete frame design manuals have been updated to remove the Concrete Design Preferences, Concrete Frame Overwrites, and Error messages and Warnings appendices (where applicable), as these provided redundant information. This information is already available within the software Concrete Frame Design Preferences form, Concrete Frame Design Overwrites form, and the various design reports.