

ETABS® Version 18.0.1 Release Notes

© Copyright Computers and Structures, Inc., 2019

Notice Date: 2019-06-27

This file lists all changes made to ETABS 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 v18.0.0 (Released 2019-06-17)

Licensing and Installation Enhancements Implemented

*	Incident	Description
*	241938	The version number has been changed to v18.0.1 for a new minor release.

Frame Design Enhancements Implemented

*	Incident	Description
	240624	An enhancement has been made to expand the design details reporting for concrete frame design according to the Russian SP 63.13330.2012 code. This includes the following: removal of long-term allowable strain limits, updating the subscripts of the epsilon values, identifying the loading type as long-term or short-term, removing the duplicate minimum moments from flexural details of columns, and adding the tension-side longitudinal rebar area as a basic shear design basis data.
	242091	The following enhancements have been made to the Russian SP 16.13330.2011 steel frame design: (1) The program has added a new parameter "Allow Plastic Strain" as a preference item and also as an overwrite item which can take a value "Yes" or "No". This parameter controls which interaction equation is to be checked. (2) The program now determines whether the bending-buckling check needs to be performed based on sections SP 16.13330.2011 8.4.4 and 8.4.6 and SP 16.13330.2011 Table 11. (3) The program checks the interaction equation Eq.10 of section SP 16.13330.2011 7.1.5 for only channel sections. This equation was previously checked for all sections. (4) The program no longer checks the interaction equation Eq.109 of section SP 16.13330.2011 9.2.2 for box sections. This is still used for all other sections. (5) The program no longer checks the interaction equation Eq.107 of section SP 16.13330.2011 9.1.3 for members with tensile force. This is still used for all sections with compression force. (6) While checking the interaction equation Eq.43 of section SP 16.13330.2011 8.2.1, the program now uses $W_{n,min}$. For single-symmetric I-shapes, T-shaped and double angles, $W_{n,min}$ is the smaller value of W_n between the tension side and compression side. For doubly symmetric sections, this change has no effect. (7) The shear areas Av_{Major} and Av_{Minor} are not recalculated based on the web and flange areas for I, channel, box, double channel, and T sections. Previously the areas were slightly different based on the corner radius. (8) The program now reports the values of τ . This is used in the determination of shear-stress ratio (τ/R_s) which is used in calculating beta and β_m factors per section 8.2.3 and in the calculation of cm factors per SP 16.13330.2011 section 8.2.3 and Eq. 53. (9) The program now reports the slenderness (l_{ef}/I , or Kl/r) limit related parameters in the database, exports, and imports. These were missing in the earlier versions.

Analysis Incidents Resolved

*	Incident	Description
	240384	An incident was resolved where the potential energy may have been incorrectly reported for models where the analysis model for nonlinear hinges was set to "Model Hinges Within Elements" and the hinges had significant nonlinear behavior. When this issue occurred, large energy errors were reported. This was a reporting error and did not affect other results.
*	241286 241431 241484 241626 241645 241692 241693 241704 241805 241837 241881 241895	An incident was resolved where analysis results were unavailable when using a Nonlinear license. Other license levels (Plus and Ultimate) were not affected. This only affected v18.0.0.
	241896	An incident was resolved where an access violation was reported while getting shell modal response results at the same time as results for other load cases.

Results Display and Output Incidents Resolved

*	Incident	Description
	229844	An incident was resolved where section cuts, story forces, and shell force/stress results at the bottom of a wall with a P-M3 wall hinge assigned were incorrect if the Analysis Model for Nonlinear Hinges option was set to "Model Hinges Within Elements". This issue did not affect models where the Analysis Model for Nonlinear Hinges option was set to "Model Hinges as Separate Link Elements".
	241034	An incident was resolved where in certain displays in DirectX graphics mode the arrow heads were missing from the arrows. This was an inadvertent error introduced in v18.0.0. It did not affect standard graphics mode.

External Import/Export Incidents Resolved

*	Incident	Description
	240881	An incident was resolved which affected the export of load combinations to Autodesk Revit. The load case factors were exported to Revit with an incorrect value of 1 instead of their actual values.

Application Programming Interface (API) Incidents Resolved

*	Incident	Description
	241799	An incident was resolved where, when using the ETABSv17 API in ETABS v18.0.0, the API methods cPropMaterial.GetMaterial and cPropMaterial.GetTypeOAPI always returned a zero (0) value for parameters which were of type eMatType. This issue did not occur when using the ETABSv1 API.

ETABS® Version 18.0.0 Release Notes

© Copyright Computers and Structures, Inc., 2019

Notice Date: 2019-06-12

This file lists all changes made to ETABS 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 v17.0.1 (Released 2018-07-30)

New Product CSiDetail™

*	Description
*	<p>A new product, CSiDetail™, for creating detailed drawings of concrete and steel structures is being released in conjunction with ETABS v18.</p> <p>Features include:</p> <ul style="list-style-type: none"> • Structural model and designed reinforcement from ETABS • Creation of framing plans and schedules for steel frames • Creation of framing plans and schedules for composite floors • Reinforcing details for concrete beams and columns • Reinforcing details for concrete shear wall elevations and cross-sections • Customizable detailing rules • Customizable user reinforcement in beams and columns • Customizable drawing sheets with multiple views • Automated bill of quantities • Print or export drawings to AutoCAD • Export reinforcement details of beams and columns to Revit <p>One free license of CSiDetail will be available for each valid commercial license of ETABS v17 in effect at the time ETABS v18.0.0 is first released. For additional licenses of CSiDetail, please contact CSI Sales. Future upgrades of CSiDetail will require a separate Maintenance contract or upgrade fee.</p>

Licensing and Installation Enhancements Implemented

*	Incident	Description
*	240739	The ETABS version number has been changed to v18.0.0 for a new major release.
	238505	The software and installation have been updated to use the Microsoft .NET Framework 4.7.1. This does not affect the behavior of the software or any results.

Loading

Enhancements Implemented

*	Incident	Description
	217307 240031	An enhancement was made to remove the ability to specify multiple angles in the same load pattern for auto wind loads based on the AS/NZS 1170.2:2011 code. Older models with multiple angles specified will keep the first angle, and new load patterns should be created by the user for other angles using the appropriate parameters. The purpose of this change is to better handle torsion for multiple directions of wind loading.
*	234583	An enhancement was made to update the default load combinations for Chinese design codes based on recent changes to the National Standard for Reliability of Building Structures.

Multi-Tower

Enhancements Implemented

*	Incident	Description
	219249	An enhancement has been made so that a window selection in a model with multiple towers will only select items from towers which are tagged to be visible.

Meshing

Enhancements Implemented

*	Incident	Description
	221368	An enhancement has been made to significantly improve the speed of creating the analysis model if a very large number of previously meshed area objects are modeled that do not need to be further meshed. Area objects that are changed, or whose meshing properties have changed, will still need to be re-meshed next time the analysis model is created.

Analysis

Enhancements Implemented

*	Incident	Description
*	066306	A modification factor for stiffness-proportional viscous damping to be used in direct-integration time-history analysis can be specified in the Link Properties Definition form. This modification factor is multiplied with the stiffness-proportional damping coefficient defined in the Time-History load case to compute the net stiffness-proportional damping coefficient to be used by the link element. This can be used to reduce or eliminate stiffness-proportional damping in a link element. In addition, the reference stiffness value to be used for stiffness-proportional viscous damping in nonlinear direct-integration time-history analyses can be specified for link properties with nonlinear degrees of freedom (DOFs). The stiffness options are: the initial stiffness, the tangent stiffness, or the effective stiffness of the nonlinear DOFs. To maintain the same behavior as in previous versions of the software, set the modification factor to unity and use the initial stiffness for stiffness-proportional viscous damping. This is still the default.
*	229433	Nonlinear static analysis, including staged-construction, has been enhanced to allow the use of line search and event-stepping at the same time. Previously only one of these two options could be used for a given load case, and event-stepping took precedence. Now, when both options are selected, event stepping will be used for the first iteration, and line search will be used for subsequent iterations. By default, events and iteration without line search will be enabled for newly created load cases, which is equivalent to the previous behavior. When opening existing models from a previous version, load cases that had both options enabled will have the line search turned off to reproduce the previous behavior. Note that line search is never used for load cases that use event-stepping only, i.e., that have iteration turned off. Note also that line search is never

*	Incident	Description
		used for nonlinear static load cases under displacement control, only for load cases under force control. Staged construction load cases always use force control.
*	018263	Rotational degrees of freedom will no longer be generated for frame (line) objects that represent truss members. Previously these zero-stiffness rotational degrees of freedom could generate instability warnings during analysis, even though the accuracy of the solution was not affected unless moment loads or rotational moments of inertia were applied at these degrees of freedom. A frame object is now automatically determined to be a truss member if (1.) End releases are assigned for moments M2 and M3 at both ends and for torsion at either end, (2.) The end offsets are zero at both ends, and (3.) The insertion points at both ends are at the centroid with no joint offsets. Note that a truss member may still exhibit internal bending moments due to transverse loading, including self-weight, but no moments will be transferred to the structure at the two ends of the member.
	223578	The size of the saved analysis results files has been reduced for multi-step nonlinear static and nonlinear direct-integration time-history load cases using the None or P-Delta geometric nonlinear options. This will reduce the amount of disk space required for these types of load cases in models containing frame, shell, and link elements. This may also result in some speed increase when running the analysis and displaying results, particularly for load cases with many steps. Results are not affected.
	223655	A new option has been added when specifying viscous proportional damping for time-history and response-spectrum load cases where the mass- and stiffness-proportional coefficients can be calculated during analysis to provide specific modal damping ratios at frequencies or periods that are relative to a given mode of the structure. Viscous proportional damping is always used for direct-integration time-history, and can be used as an option for modal time-history and response-spectrum load cases.
*	240736	The stiffness to be used for the nonlinear degrees of freedom (DOF) of link elements when running linear load cases can now be specified with more control. Previously the linear effective stiffness was always used for linear load cases starting from zero initial conditions, and the actual nonlinear stiffness existing at the end of a nonlinear load case was always used for linear load cases continuing from that load case. Now the following stiffness options are available for each nonlinear link property: (1.) "Effective stiffness", (2.) "Nonlinear stiffness", or (3.) "Effective stiffness from zero, else Nonlinear". The first option is most suitable for isolators where mode shapes and damping are to be calculated based on a specified secant stiffness, regardless of any preceding load case. The second or third options are more appropriate for gaps and other link properties where previous conditions do affect mode shapes, damping, and other linear behavior. When "Nonlinear stiffness" is chosen, the initial nonlinear stiffness is used rather than the effective stiffness for linear load cases starting from zero. This value is taken as zero for viscous dampers. To maintain the same behavior as previous versions of the software, use "Effective stiffness from zero, else Nonlinear", which is still the default. Note that geometric nonlinearity effects (P-delta and large deflections) are always included from a preceding nonlinear load case regardless of the option chosen. Linear link properties and linear DOF of nonlinear link properties are not affected by this enhancement.
*	229428	Convergence behavior of the tension-compression friction-pendulum isolator link element has been improved, particularly to deal with large variations in the axial force, which can cause alternating slip-stick behavior during lateral loading. Models that exhibited slow convergence behavior in previous versions should be re-run in the new version to verify the results. The new results will be more accurate in cases where a significant difference is observed between the old and new results. Furthermore, the friction model has been changed from the previous Wen formulation that exhibited a gradual transition between sticking and slipping to a bilinear model that exhibits a sudden transition. Some difference in results can be expected due to the new formulation, particularly for models where the initial stiffness specified for the isolator was small. Similar changes were made to the formulations of the friction-pendulum and triple-pendulum

*	Incident	Description
		isolator link elements in previous releases of the software. Finally, the new formulation supports event stepping, which can be used as part of the iteration and stepping strategy specified for nonlinear static, staged-construction, and nonlinear direct-integration time-history load cases.

Frame Design Enhancements Implemented

*	Incident	Description
	212000	An enhancement has been made to the concrete frame design code “NZS 3101-06” in which the torsion design of beams has been incorporated based on NZS 3101-06 Amendment A3.
	215319	An enhancement was made to include slightly inclined columns when calculating the frame shears used to calculate Frame Shear Ratios in Dual Frame-Shearwall systems. This affects frame modifiers when designing for seismic loads based on the Chinese code. Earlier only vertical frame members were considered for this shear calculation. Now any member that satisfies the definition of column will be included in the calculation. This is controlled by the command Options > Tolerances > Maximum Inclination from Vertical for Columns (degrees). The frame shear will use the horizontal component of the force carried by the inclined column, which may include a small fraction of the axial load.
	234084	An enhancement was made to improve the Design Details window for concrete frame design to show only those tabs that are applicable to the particular design code. Previously all available tabs were visible, even though some of the tabs did not display results for certain codes. This was a display issue only. No results were affected. The Design Details window is accessed by right-clicking on a frame member while viewing concrete frame design results on the model, then clicking the Details button on the resulting form.
	220215	An enhancement was made to speed up frame and wall design for all design codes and materials when considering many load combinations containing load cases with multi-stepped results and when step-by-step design was requested. Examples of such cases include linear and nonlinear static load cases with multiple wind or seismic load directions, and time-history load cases. No results are affected.
*	226392	An enhancement was implemented to add the new Turkish TS 500-2000(R2018) concrete frame design code, including seismic design requirements.
*	226393	An enhancement has been implemented to add concrete frame design for the Mexican Building Code (Mexico RCDF 2017).
*	226394	An enhancement has been implemented to add the new Australian AS 3600-2018 concrete frame design code.
*	226395	An enhancement has been implemented to add the new Korean KBC 2016 concrete frame design code, including seismic design requirements.
*	226396	An enhancement has been implemented to add the new Korean KBC 2016 steel frame design code, including seismic design requirements.
*	101845 102237	An enhancement has been made to the Russian concrete frame design code “SP 63.13330.2012” to include crack-width analysis for concrete beams.
*	228114	An enhancement has been made to the Russian steel frame design code SP 16.13330.2011 in which the interaction equation as stated in equation (44) of section 8.2.1 is now being checked as a PMM interaction equation. This involves the determination of elastic normal and shear stresses, calculation of the von-Mises stress, and then checking the von-Mises stress with $(R_y * \Gamma_C) / 0.87$. The results are now reported in the details window as a separate page for “Equivalent Stress Check Details”.

*	Incident	Description
	228115	An enhancement has been made to expand the design details reporting for concrete frame design according to the Russian SP 63.13330.2012 code. This includes the following: removal of long-term allowable strain limits, updating the subscripts of the epsilon values, identifying the loading type as long-term or short-term, removing the duplicate minimum moments from flexural details of columns, and adding the tension-side longitudinal rebar area as a basic shear design basis data.
	228117	An enhancement has been made to steel frame design per the Russian code SP 16.13330.2011 in which the program now allows additional input parameters for the slenderness limit check for compression and tension members per SP 16.13330.2011 section 10.4.1 and Tables 32 and 33. For compression members, the limit for slenderness ratio KL/i (or $\Lambda = l_{ef}/i$) is taken as follows: $\Lambda < K_{LoverRLimitC} - K_{LoverRLimitSlope} * \text{Alpha}$, where $\text{Alpha} = \max \{N/(\Phi * A * R_y * \Gamma_{c}), N/(\Phi_e * A * R_y * \Gamma_{c}), 0.5\}$. The program allows input for $K_{LoverRLimitC}$ and $K_{LoverRLimitSlope}$ in the overwrites. By default, the program takes the value of $K_{LoverRLimitC}$ as 180 for columns, 210 for braces, and 200 for beams; and takes the value of $K_{LoverRLimitSlope}$ as 60 for columns, 60 for braces, and 0 for beams. For tension members, the limit for slenderness ratio l/i (or Λ) can be assigned one value in the overwrites. The default value for the limit for slenderness ratio l/i is taken as 300 for columns, 400 for braces, and 300 for beams.
	228118	A change was made in concrete frame design per the Russian "SP 63.13330.2012" code on how load combinations are to be tagged (identified) as being for long-term loading. Previously all load combinations that had lateral loads (seismic or wind) were tagged as short-term, and all the remaining combinations were tagged as long-term. Now those load combinations that contain only sustained load are identified as long-term load combinations. All dead load and superimposed dead load cases are considered sustained. All wind load and earthquake dead load cases are considered short-term. All live load cases will be considered as sustained if the "Live Load Duration Factor" is 1.0. If the "Live Load Duration Factor" is less than 1.0, any combination involving live load will not be considered as long-term. Similarly, all snow load cases will be considered as sustained if the "Snow Load Duration Factor" is 1.0. If the "Snow Load Duration Factor" is less than 1.0, any combination involving snow load will not be considered as long-term. These two parameters can be modified in the Design Preferences. Tagging a load combination as short-term or long-term affects the factor Γ_{b1} , which in turn affects the design values of the compressive and tensile resistance of concrete, R_b and R_{bt} . In addition to this change, the "Gamma_b1 Short Term" parameter is removed from the Design Preferences values as this value must always be 1.0 (unity). The value of Γ_{b1} is now reported in the design details output.
	228121	An enhancement has been made to steel frame design per the Russian code SP 16.13330.2011 in which the program now considers an additional parameter for seismic factor, m_{tr} , per Table 6 of SP 14.13330.2014 when the design load combination contains an earthquake load case. For strength design/check, its value is taken as 1.3. For buckling design/check, it is taken as 1.0 for steel frame members with slenderness more than 100, 1.2 for steel frame members with slenderness less than 20, and interpolated between 1.2 and 1.0 for steel frame members with slenderness ratio between 20 and 100. This value cannot be modified by the user using the preferences or overwrites.
	228123	An enhancement has been made to steel frame design per the Russian code SP 16.13330.2011 in which the program now allows an additional input parameter for the reliability factor, Γ_n , in the Preferences form. 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 Γ_n , the design force values are practically increased. The PMM interaction ratios are directly related to this factor.

*	Incident	Description
	228125	An enhancement has been made to the Russian steel frame design code SP 16.13330.2011 in which several preference and overwrite items have been removed from the Preferences and Overwrites forms. These include "Framing Type", "Section Class", and "Live Load Limit, L/" from the Steel Frame Design Preferences form. These also include "Framing Type", "Section Class", "Live Load Limit, L/", "Live Load Limit, Abs", "Effective Length Factor Braced (K1) Major", and "Effective Length Factor Braced (K1) Minor" from the Steel Frame Design Overwrites form. All of these parameters were not used in the design. The corresponding items have been removed from the API and Database.
*	234426	<p>A change has been made to remove certain older design and loading codes. The old loading codes were inconsistent with the newer design codes which assume ultimate-level wind and seismic loading. Specifically:</p> <p>(1.) Superseded code CSA A23.3-04 for concrete frame, shear wall, and slab design is no longer supported, and this will be replaced by the CSA A23.3-14 code if existing models are opened and re-run in the new version.</p> <p>(2.) Superseded code CSA S16-09 for steel frame and composite beam design is no longer supported, and this will be replaced by the CSA S16-14 code if existing models are opened and re-run in the new version.</p> <p>(3.) Superseded auto seismic loads for BOCA 1996, UBC 94, UBC 97 and UBC 97 Isolated are no longer supported, and these will be replaced by the ASCE 7-16 load if existing models are opened and re-run in the new version.</p> <p>(4.) Superseded response spectrum functions for BOCA 1996, UBC 94 and UBC 97 are no longer supported, and these will be replaced by the ASCE 7-16 response spectrum function if existing models are opened and re-run in the new version.</p> <p>(5.) Superseded auto wind loads for BOCA 1996, UBC 94 and UBC 97 are no longer supported, and these will be replaced by the ASCE 7-16 load if existing models are opened and re-run in the new version.</p> <p>Users should check that the loading and design parameters are correct for their purposes for any converted codes.</p>
	236676	An enhancement has been made to concrete frame design for those codes where column design requires a beam-column capacity check for seismic design. The column moment is now checked for capacity in one direction (major or minor) at a time, using the axial load from each considered design combination, and while taking the moment in the other direction as zero. In addition, this check is only performed for moment directions which have significant moment for that design combination. Previously it was being checked for both major and minor direction moments simultaneously for each design combination axial load, which was overly conservative. Affected codes are "ACI 318-14", "ACI 318-11", "ACI 318-08", "AS 3600-2018", "AS 3600-09", "CSA A23.3-14", "Eurocode 2-2004", "IS 456:2000", "Italian NTC 2008", "KBC 2016", "KBC 2009", "Mexican RCDF 2017", "Mexican RCDF 2004", "TS 500-2000", and "TS 500-2000(R2018)".

Composite Beam Design Enhancements Implemented

*	Incident	Description
*	055121	Composite beam design per the AISC 360-16 code has been enhanced so that axial forces can be taken into account for design. This new option can be activated on a model-wide basis in the Composite Beam Preferences, or on a beam-by-beam basis in the Composite Beam Overwrites. The option is turned off by default, meaning that existing composite beam designs are not affected. Additional information on this option is available in the revised Composite Beam Design Manual for AISC 360-16.

*	Incident	Description
	221196	<p>Three enhancements have been made to the design reports generated for composite beams that feature web openings:</p> <p>(1.) If the user provides reinforcing plates at an opening, the aspect ratio of these plates is reported as well as the maximum value of that ratio satisfying the plates' compactness requirements.</p> <p>(2.) If reinforcing plates at an opening are not compact, an explicit message referring to the plates is now provided instead of a generic message referring to the overall beam section.</p> <p>(3.) If the beam is designed assuming composite action, the interaction ratios for each opening are reported for construction conditions and service conditions in both the interactive design form and in the output. Previously, a single interaction ratio, the most critical one of these two, was reported at each web opening.</p>
	221321	<p>Several enhancements have been made to composite beam design per Eurocode 4-2004, referred to as EC4 in the following:</p> <p>(1.) In accordance with EC4 Clause 5.4.2.1(5), the effective widths of composite beams are now limited to a fraction of their lengths L_e between points of zero bending moment. In previous versions of ETABS, the effective widths were limited to a fraction of the beam spans. Note that in the case of simply supported beams, the most common case, the two lengths are the same.</p> <p>(2.) In accordance with EC4 Clause 5.5.2(3), ETABS computes the design resistance moment of sections that have webs in class 3 and flanges in class 1 or 2 as effective cross-sections in class 2, with their effective web areas reduced per EC3-1 Clause 6.2.2.4.</p> <p>(3.) In accordance with EC4 Clause 6.2.1, the design resistance moment of composite beams is now reduced when the grade of steel is S420 or higher.</p> <p>(4.) The minimum percentage of composite action is now the lowest of the percentages specified in EC4 Clause 6.6.1.2 and in SCI NCCI: Modified Limitations on Partial Shear Connection in Beams for Building PN002a-GB - referred to as PN002a in the following.</p> <p>(5.) In accordance with EC4 Clause 6.6.1.3, ETABS displays warning messages in the Interactive Composite Beam Design form and in the composite beam design output when the plastic resistance moment of the composite section exceeds two and a half times the plastic resistance moment of the steel section alone.</p> <p>(6.) In accordance with EC4 Clause 5.4.2.2 (11), in order to account for the effects of concrete creep and shrinkage, the value of the transformed moment of inertia of the composite section is now computed using $E_c = 0.5 E_{cm}$, where E_{cm} is the modulus of elasticity specified in the concrete properties. The value of the 0.5 factor can be changed in the composite beam design preferences and beam overwrites.</p> <p>(7.) ETABS now ignores the effect of partial composite action on beam deflections when the conditions for ignoring it set forth in EC4 Clause 7.3.1(4) are satisfied, and computes beam deflections per PN002a otherwise.</p> <p>(8.) In accordance with EC4 Clause 7.3.1, ETABS now takes into account the deflection caused by concrete shrinkage when applicable.</p> <p>(9.) In accordance with SCI Publication 359: Composite Design of Steel Frame Buildings, ETABS checks that the flexural stresses remain in the elastic range under un-factored service loads.</p> <p>(10.) When a beam frames into a column or wall and is not coped, ETABS computes its shear capacity at the ends (which is the reported shear capacity) per Eurocode 3 Clause 6.2.6(3)a. Previously, it computed the shear capacity as the product of its section depth by its web thickness which was more restrictive than Eurocode 3 Clause 6.2.6(3)a.</p>

* Incident	Description
225308	Composite Beam Design has been enhanced to now consider two new types of steel sections: castellated sections and cellular sections. While castellated and cellular sections can be assigned to any frame object, sections of these types can only be designed if: (1) the selected composite beam design code is the "AISC 360-16" code, and (2) the design procedure assigned to the beams is the Composite Beam Design procedure. Castellated and cellular beams are then designed per the Provisions of AISC Design Guide 31: Castellated and Cellular Beam Design, as documented in the ETABS Composite Beam Design Manual AISC 360-16. Four Verification Examples are provided based on the design examples published in the Design Guide 31.
235872	An enhancement was made to composite beam design for all design codes: When the user selects construction stress ratios, final stress ratios, live-load deflection ratios, or total load deflection ratios in the Display Composite Beam Design Info form, the colors of the beams in the current display window now reflect the values of the selected ratio. If instead the user selects "Labels" or "Design Data" in the Display Composite Beam Design Info form, the colors of the beams reflect the values of the beams' most critical ratios. Previously the color of the beams was always based on the critical ratio regardless of the selected option. In addition, the Display Composite Beam Design Info form now features three new options: an option for displaying bottom flange stress ratios, an option for displaying vibration check ratios, and an option for displaying shear-stud crowding ratios. When one of these options is selected, the colors of the beams reflect the values of the corresponding ratio.

Shear Wall Design

Enhancements Implemented

* Incident	Description
088315	An enhancement was made to change design defaults for shear wall design to be step-by-step for load combinations that include a multi-response load case. The old default was to use enveloped forces for design. This new default makes shear wall design consistent with concrete frame design.
100052	An enhancement was made to add shear wall design based on the Russian SP 63.13330.2012 code.
229660	An enhancement was made to change the method of tracing wall legs in 3D walls. Now co-planar wall legs are no longer broken at intermediate cross walls. If the user wants to break them at the cross walls, this can be done by giving the walls on either side of the cross wall different pier assignments.
236964	Shear wall design has been implemented for the new Australian AS 3600-2018 code.
236965	Shear wall design has been implemented for the new Korean KBC 2016 code, including seismic design requirements.
236966	Shear wall design has been implemented for the new Mexican Building code (Mexico RCDF 2017), including seismic design requirements.
236967	Shear wall design has been implemented for the new Turkish TS 500-2000(R2018) code, including seismic design requirements.
225857	An enhancement has been made for concrete shear wall design per code "Chinese 2010" where the design details for spandrels has been reorganized for clarity. No results are affected. An enhancement was also made to update the overwrites for spandrels to include the clear cover at the left and right for both top and bottom.

Slab Design

Enhancements Implemented

* Incident	Description
217324 218900	Concrete slab design is now available for the Russian SP 63.13330.2012 code.

Results Display and Output Enhancements Implemented

*	Incident	Description
*	101228	An enhancement has been made to add deformation, shear, and rotation gauges. Line and quad gauges can be defined and acceptance criteria specified. Tabulated results can be obtained for the strain gauges for different linear and nonlinear load cases. This option is only available at the ETABS Ultimate level.
	220832	The command File > Create Video has been enhanced to check if the selected compression option (codec) for creating AVI files will work on the user's machine, and inform the user if it will not.
	222052 226388 228796	An enhancement has been made to add auto seismic loading and corresponding auto seismic load calculations to the Project Report for the ASCE 7-16, NBCC 2015, KBC 2016, and TSC-2018 codes.
*	224289 230486	An enhancement has been made to display load combination results on a step-by-step basis for linear 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.
*	012706 082048	An enhancement has been implemented to the response output for Generalized Displacements. Absolute and relative displacements, velocities, and accelerations will be available for all Generalized Displacements and results presented in the tables.
*	234806	The display of results from modal time-history load cases, including FNA, has been made faster for large models. This improvement will most notably affect modal time-history load cases using a large number of modes for models with a large number of joints. Speed will be improved for the graphical display of displacements, forces and stresses, section cuts, and other response quantities; for creating videos; for producing tabular output; and for running design. The speed up will be most noticeable for individual steps. The effect for envelopes will be less pronounced.

Graphics Enhancements Implemented

*	Incident	Description
	079632	An enhancement was made so that the command View > Restore Full View will now scale and center the view to include only the visible grid systems, in addition to the structure itself. Previously the full view included hidden grid systems, even if they were far from the structure.
	233487 233981	An enhancement has been made to display floor loading as contours. This display is available in plan view and will display the cumulative uniform and non-uniform distributed load applied in a load pattern.

Database Tables Enhancements Implemented

*	Incident	Description
*	053244 218309 219138	An enhancement has been made to add a new set of database tables that allow significantly better interactive editing capability (similar to other CSI programs, such as SAP2000 and SAFE). Export and import of models using tables in Excel, Access, Text, or XML format is also now supported. Some table names and field names have been changed to provide more consistency. Named Sets of tables can be defined for repeated selection of tables used for editing, display, or export. The existing text model file (.E2K, .SET) has been retained.
	220244	An enhancement was made to add the minimum rebar required by code to the slab strip design summary table. This was already available in the detailed table on a station-by-station basis and now has been added to the summary table.

**External Import/Export
Enhancements Implemented**

*	Incident	Description
	080613	A minor enhancement was made to export displayed tabulated reactions to CAD (DXF/DWG) or PDF formats as tables when using the command File > Capture Picture. This is a formatting change only; the exported data has not changed. This option to display tabulated reactions is only available in Standard Graphics mode, so the export from DirectX mode has not changed.
	227829	When creating a shell object from a .DXF file using the command Draw > Draw Floor/Wall Objects > Import User Mesh as Shell Object, warning messages are now provided when one or more of the imported mesh elements has an illegal shape and has been discarded. Illegal shapes include elements whose corners are coincident, edges are colinear or cross each other, or whose area is too small. Such illegal elements were previously discarded without warning. Legal elements are retained as before, but the resulting shell object may have holes in it, and it should be checked for suitability before being used for analysis and design.

**Application Programming Interface (API)
Enhancements Implemented**

*	Incident	Description
	203537 232255	The Application Program Interface (API) was enhanced for speed for the following functions in SapModel.cAnalysisResult used to retrieve analysis results: AreaStressShellLayered, AreaStrainShellLayered, AssembledJointMass, FrameForce, FrameJointForce, GeneralizedDispl, LinkDeformation, LinkForce, LinkJointForce, PanelZoneDeformation, PanelZoneForce, SectionCutAnalysis, and SectionCutDesign. A minor correction was also made for the programming example provided in the Help file for the AreaStressShellLayered function.
*	221919	The Application Programming Interface (API) has been updated from ETABSV17 to ETABSV1. The new API is designed to be compatible with future versions of ETABS (v19, etc.) and cross-compatible with other CSI products. The new cross-product interface, called CSiAPIv1, will work with ETABS, SAP2000, and CSiBridge, although not all functions will be implemented for every product. These interfaces will maintain compatibility with older versions of the product until a major breaking change is made to the entire cross-product system. Any client programmed against ETABSV1 will also work with CSiAPIv1. Clients previously programmed against the ETABSV17 interface should work with ETABSV1 after simply recompiling to reference the new interface.
*	238396	An enhancement was made to the Application Programming Interface (API) so that all database tables are now available for output. Functions are provided to request the contents of any table, with options to control which columns are output. Load patterns, load cases, and load combinations to be included in the tables can be controlled, as well as the type of output results (Envelope, Last Step, etc.). Available output formats include an array of values, a CSV file or string, or an XML string. An additional function is provided to display a set of tables in Excel, provided that Excel is installed on the same machine as ETABS. These functions are available in the new API class cDatabaseTables. All tables are presented in the consistent units set as the current units using the API.
*	238396	An enhancement was made to the Application Programming Interface (API) so that all importable database tables are now available for editing. Functions are provided to request the contents of any set of editable tables, with each table formatted as an array of values, a CSV file, or a CSV string. Additional functions are provided to update the model from these tables after modification. This feature behaves similarly to interactive database editing that is available in the graphical user interface. These functions are available in the new API class cDatabaseTables.

**Documentation
Enhancements Implemented**

*	Incident	Description
	221014	The composite beam design manual for the CSA S16-14 code has been revised for clarity, and is now consistent with the composite beam design manuals for the AISC codes. Composite beam design results are not affected by this change except as may be documented under other incidents.
	221322	The composite beam design manuals for the Eurocode 4-2004 code has been revised for clarity, and are now consistent with the composite beam design manuals for the AISC codes. Composite beam design results are not affected by this change except as may be documented under other incidents.

Drafting Incidents Resolved

*	Incident	Description
	100098	An incident was resolved where extrusion of lines to areas would not work correctly in certain cases. When this happened the error was obvious. Results agreed with the model as generated.
	201241	An incident was resolved where the Drawing Control Type, once set to an applicable value for drawing one type of object (say frames), sometimes remained set to that value when drawing a different type of object (say floors), even if the value was not applicable. This could make drawing the second type of object difficult, but could be resolved by returning to drawing the first type of object and setting the Drawing Control Type to "None".
	201288	An incident was resolved where certain editing options for walls, including on-screen reshaping, would not work correctly after their local-3 axes had been reversed using the command Edit > Edit Shells > Reverse Wall Local 3 Axis. Results agreed with the model as edited.
	215440	An incident was resolved where moving joints in one part of the structure was causing tendon profiles to be lost in a model even though the joints were not connected to the tendons. This happened when the tendon profiles were being recalculated, but the original strips referenced for drawing the tendons had been deleted. The tendon profiles are now no longer recalculated if the joints moved are not connected to them.
	216037	An incident was resolved where the option to have pier names automatically assigned did not work when creating walls from an architectural plan.
	217327	An incident was resolved where objects drawn in a 2D view (plan or elevation) did not always appear in 3D views until the 3D views were manually refreshed. This issue only affected DirectX graphics mode.
	228994	An incident was resolved where an abnormal termination would occur when trying to add a template model to an existing model with no gridlines in the grid system. Adding from a template requires at least two gridlines in each direction to work. The condition is now detected and an error message is provided to the user so that it can be corrected before allowing the operation to proceed. The affected menu command is Edit > Add to Model from Template.
	225133	An incident was resolved where, in certain particular cases, the Replicate command was adding duplicated joints. This was a very rare occurrence and typically occurred before saving the model after starting with a blank model.
	233510	An incident was resolved where the option to add grids via the menu command Edit > Edit Stories and Grid Systems ... "Add from dxf/dwg File" was only intended to work from dxf files. The command name has been corrected to "Add from dxf File" to avoid confusion.
	233603	An incident was resolved where architectural layers used as reference for building the model would not be correctly adjusted when a story was deleted, causing the architectural layers to disappear or to be associated with the wrong stories. No results were affected.

Modeling Incidents Resolved

*	Incident	Description
	088801	An incident was resolved where the centroid was not correct for hat-shaped, cold-formed steel frame sections. For such sections, analysis and design results could be incorrect, but agreed with the location of the centroid as shown.
	202780 212302 214809 218283	An incident was resolved where nonlinear hinges could not be assigned to wall objects after their local-3 axes had been reversed using the command Edit > Edit Shells > Reverse Wall Local 3 Axis. When this occurred, an error message was given and the hinge assignment did not take effect.

*	Incident	Description
	219900	An incident was resolved where a frame member that was assigned a frame section property of type buckling-restrained brace (BRB), including CoreBrace BRB properties, was not using the BRB features if the member was classified as a beam rather than a brace. This would be the case if the angle of the member with respect to the horizontal plane was less than the brace tolerance, typically 20 degrees. When this happened, the member used the underlying section property and ignored the additional BRB features. In addition to the effect on the behavior of the model, this could also generate an error message when displaying the table of frame members assigned CoreBrace BRB section properties. Now, all inclined members assigned BRB properties will use the full BRB features. However, horizontal members will still ignore the additional BRB features. The tolerance for horizontality is that the sine of the angle of inclination be 0.01 or less.
	229424	An incident was resolved where, in rare cases, exporting and then importing a model using the text file (.E2K, .SET) would cause user-defined notional sizes of concrete members used for time-dependent creep and shrinkage calculations to not be applied correctly. When this occurred, results agreed with the model as imported.
	238127	An incident was resolved where the yield moment for nonlinear hinges in a concrete frame member may not have been calculated correctly if the section was originally a T section, but was later modified in the Frame Section Property Data form to become a rectangular beam. In such a case, the width of the beam was not properly used to calculate the hinge properties. This did not affect rectangular frame sections or T sections that were not converted to rectangular.

Section Designer Incidents Resolved

*	Incident	Description
	221770	An incident was resolved where an error message was generated when saving the text file if the model contained a Section Designer section that had a circular rebar pattern where only the tie bar information was given, but no longitudinal rebar size or area was given.
	231293	An incident was resolved where in some instances the CG Offsets shown in the Frame Section Properties form were not correct for Section Designer sections. No results were affected, and this was only a display issue in that form. This would occur when the geometric center of the section bounding box did not coincide with the origin of the x-y coordinate system in Section Designer and the 2-3 axes of the section were rotated with respect to the x-y axes. The CG Offsets are measured from the geometric center of the bounding box to the centroid. No other section property values were affected.

Loading Incidents Resolved

*	Incident	Description
	207741	An incident was resolved where the joint pattern information shown when right-clicking on area (shell) objects did not provide the option to select "Zero positive values" from the drop-down list. Instead the option "Zero negative values" was presented twice. Results were not affected.
	218412	An incident was resolved for the NBCC 2015 auto seismic load case where parameter Mv was not saved when changed on the form and instead it defaulted back to its original value. The error was evident from the auto seismic table and the seismic report, and results were consistent with the default value.
	220945 222148 228158	An incident was resolved where, for certain models, an error condition would occur when calculating torsional loads due to accidental eccentricity for a response spectrum analysis. This happened when there were one or more stories in the model that had no joints. Now such stories will be ignored without error.

*	Incident	Description
	222347	An incident was resolved for the ASCE 7-16 response spectrum function where coefficients Fa and Fv were not allowed to be overwritten by the user for Site Class E in cases where they should be site specific.
	223867	An incident was resolved where in the ASCE 7-05 Auto-Seismic Loading form the options for selecting coefficients "Ct" and "x" for Time Period options "Approximate" and "Program Calculated" were always being reset to the first option "0.028, 0.8" regardless of user choice. This form is accessed using the menu command Define > Load Patterns > Modify Lateral Load for load patterns of type Seismic and with Auto Lateral Load set to ASCE 7-05. Results were consistent with the values of "Ct", "x" = "0.028, 0.8". No other codes were affected.
	228400	An incident was resolved where time-history functions of the type "Matched to Response Spectrum" would generate an error when later viewed or edited if that function referenced a user-defined response-spectrum or time-history function, and either of those functions was subsequently deleted. Now user-defined response-spectrum or time-history functions that are referenced by a "Matched to Response Spectrum" function are protected from deletion.
	236042	An incident was resolved where time-history functions defined to match to a response-spectrum curve could not be generated when the source response-spectrum function had more than 400 points. The upper limit on the number of points has been reduced to 200 for reasons of efficiency, but now response-spectrum functions exceeding that limit will be simplified by interpolation before being used to generate matched time-history functions. Practical response-spectrum curves with a reasonably smooth shape will not be affected by this process.
	225639	An incident was resolved where the PhiZ parameter in Chinese Wind Load calculations was not correctly interpolated from the mode shape, when it was specified to be taken from mode shapes, if the Bottom story specified for exposure was not at zero elevation.

Multi-Tower Incidents Resolved

*	Incident	Description
	058268	An incident was resolved where Quick-Draw wall objects for multi-tower models were always being assigned to Tower 1 if the auto-tower option was selected. Now the walls will be assigned to the correct tower based on the grids used for the Quick Draw operation.
	059250	An incident was resolved where using the arrow buttons to move up and down the stories when displaying a plan view would sometimes cause an error condition to occur in multi-tower models. Results were not affected.
	062048	An incident was resolved where in a multi-tower model, some frame elements were not being auto-meshed at intermediate joints, leading to disconnected models. This particularly could affect the connection of secondary beams to primary members.
	066787	An incident was resolved for multi-tower models where the replicate-by-story command between towers did not always work correctly, and subsequently exporting and then re-importing the model using the text file (.E2K, .SET) could result in an error condition.
	067034	<p>An incident was resolved where joints and objects when drawn for multi-tower models would be assigned to incorrect towers in some cases when the tower was not explicitly specified and the "Auto" option was used. The following process is now used to determine the tower assignment when the "Auto" option is used:</p> <ol style="list-style-type: none"> (1.) When using Quick Draw options, the tower assigned is based on the tower associated with the grids that are clicked. If grids overlap, the tower defined first is assigned. The Similar-Story option is then applied only to the stories of this tower. (2.) When drawing on-screen without using Quick-Draw tools, the tower associated with the current Plan view, Elevation view, or Working Plane in a 3D view is assigned. (3.) When drawing in 3D without setting a working plane, the tower associated with the first object snapped-to when starting to draw the object is assigned.

*	Incident	Description
	067292	An incident was resolved where it was possible to have duplicated elements in multi-tower models. The duplicated elements were at the same location but belonged to different towers. This is now detected and prevented, and previous models are corrected as needed when opened.
	067962	An incident was resolved where, in certain multi-tower models, the option to merge towers or delete towers did not work correctly. This could corrupt the model and later cause an error condition when working with the model.
	089694 093956 097301 103835	An incident was resolved where, when selecting columns in a plan view using the window selection for a model with multiple towers, columns would be selected both below and above the story level. Only columns below the story level are supposed to be selected in plan views.
	093182	An incident was resolved for multi-tower models where the joints created when drawing frame, shell, and link objects in one tower would be assigned to the default tower. This could cause connectivity problems when the model was exported to text file (.E2K, .SET) and then re-imported. In addition, this could cause wall objects to be drawn with the incorrect geometry if the story levels were different between the tower where drawn and the default tower. When this occurred, the effect was obvious and results agreed with the model as drawn.
	095065	An incident was resolved where selecting an object in a plan view when multiple towers are being modeled would sometimes select more items than actually clicked on.
	062082 065813 093398 096955	An incident was resolved where, for multi-tower models, exporting and re-importing the model using the text file (.E2K, .SET) did not always bring in all members. This caused an error condition later when generating joint tables.
	201537	An incident was resolved for multi-tower models where selecting columns while the "All Stories" option was active did not always select all the columns on that same column line.
	202501	An incident was resolved for multi-tower models where the story limits selected under the command View > Set Building View Limits did not always work correctly, causing more or less of the building to be displayed than expected. No results were affected.
	101747	An incident was resolved where, when importing a DXF file into a multi-tower model, the form used to choose the vertical elevation for import did not always display the correct story names.
	103423 214217 233364	An incident was resolved for multi-tower models where in some cases the data written out to the text file (.E2K, .SET) was not sufficient to re-import the model from the text file. The binary model file (.EDB) was not affected.
	100272	An incident was resolved where for multi-tower models the "Quick Draw Walls" option and the replicate option for walls on another story did not always work correctly.

Meshing Incidents Resolved

*	Incident	Description
	100336	An incident was resolved where the automatic Rectangular mesh option for floors did not recognize the end points of wall objects from the story above, and hence the floor mesh did not always connect fully to the wall. This issue did not affect the floor above the wall, and hence also did not affect the floor below if the wall was continuous between stories. Results agreed with the model as meshed. Now the end points of the wall will be explicitly included in the rectangular mesh of the floor below. Intermediate wall points will connect to the floor below using edge constraints, as they did before. This issue did not affect the automatic General mesh option for floors, which fully meshes to walls above and below the floor.
	200892	An incident was resolved where the meshing of shells was not correct at visible grids for a particular model. The issue was due to a tolerance problem when determining the intersection of two nearly parallel lines. The behavior has been improved.

*	Incident	Description
	220061 220966	An incident was resolved where, in certain cases, the general meshing would not work correctly near openings. When this occurred, the error was obvious, and manually adding some meshing lines to connect the opening to the nearest column or wall support would resolve the problem.
	221514	An incident was resolved where an error condition occurred during the creation of the analysis model if a wall was assigned a hinge after the model had been run earlier with the wall being assigned any type of meshing, which then stored an internal mesh for future runs. Meshing of wall objects with hinge assignments is not allowed. This is now being detected and the stored mesh is deleted after a hinge is assigned.
	221714 221833	An incident was resolved where general meshing on an inclined floor was, in some cases, creating a mismatched mesh that could cause the analysis to experience convergence problems for nonlinear load cases. The general meshing feature has been improved to create better connected meshes. Models from previous versions will have their automatic floor meshes deleted when opened in the new version so that an improved mesh will be regenerated next time the analysis is run.
	221130 222082 222221 222750	An incident was resolved where the intersection of nearly parallel lines and/or edges was causing errors in the mesh created by the general mesher. This happened when lines/edges that were supposed to be in the same location were slightly off. Tolerances have been modified and new checks added to minimize this problem.
	228681	An incident was resolved where, during meshing of a floor, a drop panel near to a wall was sometimes meshed as an opening. This was a rare case and the error was obvious when it occurred.
	228333	An incident was resolved where the General mesher would not pick the nodes of a brace member as joints to be included for floor meshing unless the joints were themselves tagged as mesh points. Now they are treated the same as the nodes for a column and automatically included.
*	227177	An incident was resolved where general meshing of certain floor objects, such as very acute triangles, could create illegally shaped elements that prevented the analysis from running. The general meshing feature has been improved to create valid meshes in such cases. Models from previous versions will have their automatic floor meshes deleted when opened in the new version so that an improved mesh will be regenerated the next time the analysis is run.
	232231	An incident was resolved where the saved floor mesh was not being updated for analysis if the floor-meshing type or mesh size was not changed, but other options were changed, such as where to mesh (at grids, etc.). Now the saved mesh will be regenerated at analysis time if any meshing option is changed that applies to the floor object.
	232435	An incident was resolved where joint offsets were sometimes not properly interpolated to shell elements when a shell object (e.g., wall or floor) was meshed. This error only affected the following case where three conditions were met: (1.) the shell object had more than four joints, and some of the joints were co-linear or nearly so along a skewed line; (2.) the shell objects had explicit joint offsets defined beyond the implicit offsets that are created for the top-of-slab cardinal point; and (3.) the object was assigned that the stiffness was to be transformed for the effect of joint offsets or a tendon was present in the model. When this error occurred, some joints of the shell elements generated by meshing the shell object might not have been properly offset, resulting in elements that were inclined. If the resulting element was excessively distorted the analysis produced an error message in the .LOG file and the analysis was stopped. Otherwise, results for the affected elements may have been inaccurate.

Analysis Incidents Resolved

*	Incident	Description
	220150	An incident was resolved where the analysis would become unresponsive if the model file name or folder had characters that were inconsistent with the system locale settings. When this occurred, no results were available. Now the software will give an error message in this case, so that the user can change the system locale settings to be consistent with file and folder names.

*	Incident	Description
	221098	An incident was resolved where the stiffness transformation when the frame object connectivity was not specified at its centroid could be slightly off. This happened for unsymmetrical sections when the "transform stiffness" flag was "On". The stiffness transformation was off by the distance between the centroid and the geometric center of the section bounding box.
*	222053	An incident was resolved where mass-proportional damping specified in the material property definitions was not being considered for shell elements in nonlinear direct-integration time-history load cases. Additionally, this issue sometimes required extra iteration to reach convergence. Mass-proportional damping specified in the load case itself was not affected. Models without shell elements or without mass-proportional damping in materials used by shell elements were not affected.
	221172	An incident was resolved where nonlinear analysis of models containing triple-pendulum isolators could fail to converge if the isolators went into axial tension. This occurred because the locations of the internal components of the device become undefined when there is no compression to keep the multiple surfaces in contact. Now a small, fictional internal transverse stiffness is assumed to provide definiteness in the presence of tension so that convergence can be more readily achieved. However, analysis results can still be numerically sensitive when the isolators go into tension, depending upon the stiffness characteristics of the isolators themselves and the rest of the structure. Engineering judgment is required to determine if tension is acceptable for these devices.
	228871	An incident was resolved where a column fiber hinge that contained a fiber assigned using a concrete "Mander" curve would, in some cases, create a stress-strain curve that was missing the compression side. When this occurred, an error was generated that prevented the analysis from running, and no results were available.
*	230613	An incident was resolved where the creep and shrinkage strains for frame objects calculated during time-dependent staged construction load cases could be incorrect when all of the following conditions were met: 1. The frame objects were assigned a non-zero value of rebar ratio for creep and shrinkage (menu command Assign > Frame > Rebar Ratio for Creep Analysis). 2. More than one time-dependent concrete material property was used in the frame section properties for the model. 3. The analysis was run in a multi-threaded environment. This would normally be the case on any machine with more than 2 cores, unless the environment variable SAPFIRE_NUM_THREADS was set to 1 (unity). Correct results were able to be obtained by setting the environment variable SAPFIRE_NUM_THREADS = 1 in Windows Control Panel before starting ETABS.

Frame Design
Incidents Resolved

*	Incident	Description
	087610	An incident was resolved for concrete frame column design where Beam/Column capacity ratios were incorrectly reported in the detailed design report. The reported capacities were correct and only the reported ratios were switched between those obtained from clockwise moments and counter-clockwise moments. This was a reporting issue only, and all concrete frame design with seismic provisions were affected.
	100265	An incident has been resolved for steel frame design per the "Chinese 2010" code in which brace members were sometimes incorrectly identified with the message "Section is seismically slender." This message was over-conservative. No other results were affected.

*	Incident	Description
	101901	An incident was resolved for concrete frame design using the Chinese 2010 code in which the calculation of λ (lambda) for columns was not correct for two reasons: (1.) In the calculation of λ , the full depth of column was used instead of the effective depth h_0 . λ should be equal to $M/(V \cdot h_0)$. (2.) The program used the values of M and V after modification with magnification factors SMF and MMFs instead of using the values of M and V before modification with magnification factors SMF and MMFs. The calculated λ values should not change with the overwrites of magnification factors for columns. This issue was only present for columns. The lambda values for beams are calculated based on the actual presence of a point load, if any. If there is no point load on the beam, then λ is taken as 1.5. The calculation of λ for beams was not changed. Note that this incident was erroneously reported as being resolved in the Release Notes for ETABS v17.0.0, which was not the case.
	101908	An incident was resolved for concrete frame design per the Chinese 2010 code in which the calculation of the minimum rebar for columns was not correct. This affected only the concrete column design. This did not affect column checking or beam design.
	215273 215524	An incident was resolved for concrete frame design where the moments for negative bending were sometimes reported as zeroes in the table "Concrete Beam Flexure Envelope", based on whether the tension-side required rebar (top) for negative moment was less than the compression-side required rebar (top) for positive moment, and vice-versa. This was a reporting issue only. The design procedure and all other results were not affected. The design procedure considers all the steps within a multi-step load case or multi-valued load combination, so that the net enveloping rebar requirements were reported correctly.
	217361	An incident has been resolved for steel frame design per the Indian "IS 800:2007" code where the design preference form showed zones as 0 to 4 but these should instead be from Zone II to Zone V. Previous zones 1 to 4 correspond, respectively, to zones II, III, IV and V, and previous zone 0 is now treated as zone II. After considering the updated numbering, results were otherwise not affected.
	219905 220967 221298 240226	An incident was resolved where concrete frame design would produce an error condition when there was an inconsistency between the analysis section of a member and its design section in the specific case where one was of type prismatic and the other was of type non-prismatic. When this occurred, design results were not available for that member. Analysis results were not affected and were consistent with the analysis section.
	221686	An incident was resolved for steel frame design using codes "Eurocode 3-2005", "Italian NTC 2008", and "Indian IS 800:2007" in which the PMM interaction ratio for the section-capacity check of rectangular hollow sections became extremely large when the axial force ratio n , ($n = N_{ed}/N_{pl,Rd}$), became slightly larger than 0.9407 ($n > 1/\sqrt{1.13}$) but still less than 1.0. In this case the denominator of the expressions for Alpha and Beta became a very small negative number, causing Alpha and Beta to be very large negative numbers. These, in turn, produced extremely large values for $[M_y, E_d/M_{n,y}, E_d]$ and $[M_z, E_d/M_{n,z}, E_d]$. Now, n is limited to 0.7375 when calculating Alpha and Beta so that they do not exceed the code-specified limit of 6.0 or become negative. This issue was sometimes causing problems when creating certain database tables because of the large numbers.
	222713 223862 227924	An incident was resolved for concrete frame design per the ACI 318-14, ACI 318-11 and ACI 318-08 codes where no warning was reported when the δ_{ns} factor was greater than 1.4. Now this is checked and a warning message is provided in the design report and design tables when the δ_{ns} factor exceeds the 1.4 limit.

*	Incident	Description
	228113	<p>An incident was resolved for steel frame design per the Russian code SP 16.13330.2011 affecting some of the interaction equations used to determine the D/C ratio of pipe sections:</p> <p>(1.) The interaction equations of section p9.2.2 were calculating the parameter ϕ_e based on the major- and minor-axis bending moments separately. Now the parameter ϕ_e is based on m_{ef} for the resultant bending moment $M = \sqrt{M2^2 + M3^2}$. The previous results were under-conservative regarding this.</p> <p>(2.) Equations 120 and 121 of section p9.2.10 were being used to determine the D/C ratio of pipe sections. This was not necessary, and this interaction equation is no longer used. The previous results were slightly over-conservative regarding this.</p>
	228119	<p>An incident was resolved for concrete frame design per the Russian code SP 63.13330.2012 in which the limiting strain ϵ_{b2} was not being reported correctly. The short-term limit concrete strain was determined using p.6.1.20 SP63 and was correct for both concrete beams and columns. This was a reporting issue only. Design results were not affected.</p>
*	234629	<p>An incident was resolved for steel frame design per the “Eurocode 3-2005” code where the demand-capacity ratio (D/C) computed for closed pipe and tube (box) sections could be unconservative in the presence of significant torsion. For the design of pipe sections, the shear stress (τ) due to torsion was being calculated using $c = t/2$ instead of $c = d/2$ in the formula $\tau = T*c/J$. A similar error occurred for tube sections, where the computed stress was proportional to the wall thickness rather than to the exterior dimensions of the section. This error affected the calculations of the strength reduction factor, reduced shear capacity for the presence of torsion, and reduced flexural capacity for the presence of torsion. The resulting D/C ratios were smaller than their correct values in the presence of torsion. Only pipe and tube sections were affected. In the absence of torsion, the D/C ratios were correct.</p>
	225637	<p>An incident was resolved for concrete frame design code “Chinese 2010” in which the Beam Gravity Neg Moment Reduction Factor parameter set in the design preferences was not being used. Members used a default value of 1.0 unless a design overwrite assigned a different value. Now the default value of the Beam Gravity Neg Moment Reduction Factor for members without overwrites will be taken from the design preferences. The default value in the design preferences has been changed from 1.0 to 0.85, but this can be changed by the user.</p>
	225688	<p>An enhancement was made to update the design details output for concrete frame design code “Chinese 2010”. Previously the values for N, M2, and M3 were reported only after all modifications. Now the design reports those forces that directly come from the load combinations, and all manipulated values are reported later in the design output. No results are affected.</p>
	239399	<p>An incident was resolved for steel frame design per the Russian code “SP 16.13330.2011” which corrected the following issues:</p> <p>(1.) The sign of axial force N is now taken as negative for compression for the calculation of delta in equation (108) of section 9.1.3 and in equation (122) of section 9.2.10. Previously the factor delta was always 1.0 (unity) for compression. Now it is always greater than 1.0 for compression. The factor delta was and still is taken as 1.0 for positive N. This factor is not used when the member is in tension. The previous results were over-conservative.</p> <p>(2.) The equation $\eta = (0.75 + 0.05*m) - 0.01*(5.0 - m)*\lambda_{bar}$ is now changed to equation $\eta = (0.75 + 0.05*m) + 0.01*(5.0 - m)*\lambda_{bar}$ by changing the subtraction to addition for minor axis bending of I-shapes (Section type 8) per Annex E, Table E.2, Section type 8 for $A_f/A_w = 0.25$. The net effect is that eta for this case will have an increased value, the factor m_{ef} will have an increased value per equation (110) of section 9.2.2, the stability factor ϕ_e will have a reduced value per Table E.3, 9.2.2, and the PMM ratio will have a slightly increased value. Previously the design was slightly unconservative. However, the overall effect was usually small since the minor-axis bending moment for I-shaped members is normally small; for larger moments in minor direction, the effect was null since the affected equation does not apply in that case.</p> <p>(3.) For the stability check of axial compression with flexure for box (tube) and pipe sections, only Section 9.2.10 Equations (120) and (121) are now being checked. The following interaction equations</p>

*	Incident	Description
		<p>are no longer checked for this case: Section 9.2.4 Eqn. (111), Section 8.2.8 Eqn. (59) and (60), Section 9.1.1 Eqn. (106), and Section 9.1.3 Eqn. (107a). The previous results were over-conservative.</p> <p>(4.) When the Eqn. (107) of section 9.1.3 governed, the reporting of this equation and the ratios was not correct. In some cases, M_y was being reported instead of M_x. In addition, for singly-symmetric I-sections, Double-Angles, and T-sections, the minor axis bending is now ignored. For Double-Angles and T-sections, this interaction equation is now only checked if the major moment is positive. In this case, the maximum tension occurs at the bottom tip. For Channel sections, both the minor and major moments are considered as both affect the tensile stress at the tips.</p> <p>(5.) For some T-sections, there was a discrepancy between the reported Λ_{bar} from its hand calculated value even though the $\Lambda (KL/i)$ matches exactly. This was caused by inconsistency of radius of gyration and $\sqrt{I/A}$ where all of these three quantities (I, A, and i) are taken from the section-property database file. Now the radius of gyration i from the section database is used to calculate KL/i rather than using $\sqrt{I/A}$.</p>
	240117	<p>An incident was resolved for steel frame design per the AISC 360-16 code where the compression-flange local buckling capacity was underestimated for I, channel, and double-channel sections with a slender web and non-compact flanges.</p>

Composite Beam Design Incidents Resolved

*	Incident	Description
	061846	<p>An incident was resolved for composite beam design that affected beams skewed in plan. When a beam supported on its right side with other beams framing into it at an angle other than 90 degrees, the effective width automatically computed for the right side of the beam supporting the others was zero. When this occurred, the designs produced were slightly conservative, and the effective width on the right side of the beam was reported as being zero in the beam composite design output. The user had the option of specifying a positive effective width in the beam composite design overwrites, which avoided this problem.</p>
	215321	<p>An incident that affected composite beam design per Eurocode 4-2004 was resolved. Previously, when a class 3 section was chosen for the design of a composite beam and that section did not have an adequate elastic positive-bending capacity, the member was reported as having failed, but the reported bending-capacity check was incorrectly based on plastic bending capacity and appeared to be adequate. However, this issue is now moot due to the further enhancements to composite beam design per Eurocode 4-2004 as described for Incident 221321 in these Release Notes: ETABS now implements EC4 Clause 5.5.2(3) and computes the design resistance moment of sections that have webs in class 3 and flanges in class 1 or 2 as effective cross-sections in class 2, with their effective web areas reduced per EC3-1 Clause 6.2.2.4.</p>
	217222	<p>An incident has been resolved for composite beam design which affected the reports generated for the case when the deck adjacent to the beams was of type solid slab as opposed to filled or unfilled deck. For solid slabs, the deck width and the concrete moduli of elasticity used to check strength and deflection were mislabeled, the concrete modulus of elasticity used to check vibrations was not reported, and the shear stud strength was not reported. This was a reporting error only for the input values. The design results themselves were correct. This affected all composite beam design codes in all versions of ETABS from v13.2.1 to v17.0.1.</p>
	217420	<p>An incident was resolved that addressed several issues with composite beam design related to web openings:</p> <p>(1.) When designing a composite beam in interactive mode, if the user specified opening reinforcing plates that did not satisfy the AISC compact-section requirements in the Beam Web Penetration form and then clicked OK, the user could not return again to the Beam Web Penetration form. The only way then to modify the openings for the beam was to export the</p>

*	Incident	Description
		<p>model to an .e2k text file, change the dimensions of the plates in a text editor, and re-import the model from the .e2k file.</p> <p>(2.) In the Interactive Composite Beam Design form, the ratios displayed next to each design in the list of alternate designs were the largest of the ratios computed for the beam overall strength, stiffness, and vibration checks, and did not take into account the interaction ratios at the various web openings along the beam. However, the web opening ratios were correctly updated and displayed in the Strength Checks section of the form when the user selected a particular design.</p> <p>(3.) When viewing composite beam design results, the beam colors depicted the largest of the ratios computed for each beam overall strength, stiffness, and vibration checks, but did not depict the interaction ratios at any of the beam web openings.</p> <p>(4.) When the shear studs placed on a beam to satisfy maximum stud-spacing requirements provided more than 25% composite action, but no composite action was needed to satisfy the overall beam strength and stiffness design requirements, any web openings on the beam were checked as if the beam was non-composite. This could lead ETABS to incorrectly reject some viable beam designs. Setting the beam type as Force Composite in the Shear Studs tab of the Beam Overwrites solved the issue.</p> <p>(5.) When a beam was designed as non-composite but had shear studs on it anyway, any web openings were checked to assume no composite action, except that the value of the phi factor was 0.85 instead of 0.9. This was slightly conservative.</p>
	219576	<p>An incident was resolved that affected composite beam design per the CSA Canadian design codes. When checking the negative bending capacity of a composite beam per the CSA Canadian design code with a section classification other than Class 1, (Class 2, Class 3 or Class 4), ETABS did not take into account the unbraced length of the beam. When this occurred, the error was obvious in the interactive design form and in the output as the corrective negative bending capacity of the beam was displayed on the form and reported in the output. All versions of ETABS that design composite beams per the Canadian Code, versions 9.7.0 to 17.0.1, were affected.</p>
	220034 220885 220988 221955	<p>An incident was resolved that affected composite beam design per the AISC 360-05 and AISC 360-10 codes in ETABS v17.0.1. The compactness of the webs of trial design sections was incorrectly checked and was essentially always classified as compact when the database length unit was feet, inches, meters or centimeters, and was always classified as non-compact when the database length unit was millimeters. Only ETABS 17.0.1 was affected by this incident. Note that per AISC, all AISC rolled ASTM A6 W, S, and HP shape sections have compact webs at $F_y \leq 65$ ksi, and that when the database length unit was millimeters, no designs were generated. The compactness of the webs of trial section was otherwise properly checked when the design code was AISC 360-16.</p>
*	223907	<p>An incident has been resolved that addresses two issues for composite beam design per the Eurocode 4 code:</p> <p>(1.) When checking the construction bending capacity or the negative bending capacity of a composite beam per Eurocode 4, ETABS did not take into account the unbraced length of the beam.</p> <p>(2.) The design resistance moment of composite beams was not being reduced per Eurocode 4 Clause 6.2.2.4 when the shear-design forces exceeded 50% of the beam shear-design resistance. All versions of ETABS that design composite beams per Eurocode 4 (v13.0.0 to v17.0.1) were affected by these two issues.</p>

*	Incident	Description
	216713	An incident was resolved for composite beam design that affected beams adjacent to floor objects having Slab section properties. When designing such beams, the design incorrectly assumed composite action with the slab and used incorrect concrete strength and slab thickness. The strength used was that of the first concrete material defined in the model, and the deck was assumed to have ribs of depth 3 in. or 75 mm, and a concrete cover of thickness 3.5 in. or 87.5 mm. Composite beams adjacent to floor objects with slab properties are now designed as non-composite, as stated in the manual. Floor objects that are intended to provide composite action should be defined as deck sections, which include an option to model solid slabs. This error affected ETABS Versions 13.0.0 to v17.0.1. Note that this incident did not affect the more common cases of beams adjacent to floor area objects with Deck section properties, even when these Deck section properties were of type "Solid Slab".
	233179	An incident was resolved which affected composite beam design per Eurocode 4-2004 when the Option "Use Multi-processing for Design/Results" was turned on and the model database length unit was not millimeters: (1.) Some of the beam sections were randomly and incorrectly classified as non-compact, resulting in excessive stress ratios being reported. (2.) The shear-stud capacity was randomly under-evaluated for some beams, resulting in these beams being deemed inadequate due to shear-stud crowding. These errors were obvious when they occurred, and they were conservative. These errors did not occur when designing beams interactively, or when the Option "Use Multi-processing for Design/Results" was turned off. Only versions 17.0.0 and 17.0.1 had this option and could be affected. Models initially created or imported in metric units were not affected, while models initiated in U.S. Customary units were affected.

Shear-Wall Design Incidents Resolved

*	Incident	Description
	220080	An incident was resolved where opening a model in ETABS v17 that was created in an earlier version would reset the option from "check" shear wall with uniform reinforcing to "design" shear wall with uniform reinforcing. Resetting the option to the desired value in v17 would correctly retain the value. This affected versions v17.0.0 to v17.0.1.
*	218753 220962 222586 222864	An incident was resolved for shear wall design based on the General Pier Section (i.e., a Section Designer section) where design was unable to calculate the interaction diagram when the section designer section included a rectangular rebar layout for the edge zone. The issue only affected ETABS v17. No other results were affected.
*	213978 226014 226527	An incident was resolved for shear wall design of piers based on the Eurocode where right-clicking on a wall pier to obtain details would give results inconsistent with what is shown on the model display. This was related to an initialization issue and affected piers where additional moments due to slenderness were required to be added for design.

Slab Design Incidents Resolved

*	Incident	Description
	207438	An incident was resolved for slab design per the ACI 318-14 code where one-way shear design was enforcing the minimum value A_{vmin} when $V_u > 0.5 \cdot \phi \cdot V_c$ instead of using the correct condition $V_u > \phi \cdot V_c$.
	209917	An incident was resolved for slab punching check where a column with insertion point offsets was causing the program to give an error message and not calculate punching shear at that column.

*	Incident	Description
	220220	An incident was resolved where the units shown when displaying Punching-Shear reinforcement results would not change when the display units were changed.

Results Display and Output

Incidents Resolved

*	Incident	Description
	215396	An incident was resolved where an abnormal termination could occur when attempting to generate a report containing auto seismic load patterns that did not have a load direction selected. Results were not affected.
	217235	An incident was resolved where the display of response values (displacements, forces, stresses) on the model at the location of the mouse cursor sometimes became inactive after zooming in when using DirectX graphics mode. Rotating the model would restore the correct display behavior until the next zoom operation. No results were affected.
*	214931	An incident was resolved where the overturning moment shown in the story response plots was incorrect when the model had no joints present at the Base story level, which is not common. Only the story response plots were affected. The overturning moment shown in the table for story shears was correct. No other results were affected.
	215146 215228	An incident was resolved where Max/Min strip forces presented in the tables was only showing the Max forces as both Max and Min. Slab design was not affected by this issue. In addition, the strip-forces diagram has been enhanced to show both Max and Min forces on the same diagram similar to the way frame forces are displayed.
*	224294 225204	An incident was resolved where the user-generated report would not populate the calculation results for concrete frame and shear wall designs even though the option to generate these calculations was activated under the report options. This issue only affected versions v17.0.0 and 17.0.1.
	225106	An incident was resolved where an abnormal termination occurred when plotting hinge results for a staged-construction load case defined to produce output at the "End of Each Stage". No results were affected. This did not occur for other output options on the load case, such as "Start and End of Each Stage".
	227944	An incident was resolved where the contour range calculated for display of slab moment/stresses was very high due to the presence of stiff areas over columns and walls. Forces/stresses for stiff areas are now excluded for calculating the range giving better contour definition over the flexible portions of the slab. This was a display issue only and no design results were affected. The design already excludes stiff areas over columns and walls.
	220812	An incident was resolved where the joint reaction information displayed by hovering the mouse cursor at a joint did not correspond to the reaction at that joint. This was a rare occurrence and happened only in developed elevations and only at internally created meshing joints.
	228650 228738	An incident was resolved where tables of analysis results, such as Base Reactions and Story Forces, could produce incorrect values for some load cases or load combinations if, and only if, results were also requested for a load combination that was invalid, i.e., it was defined to contain no load cases or load combinations. This was not common. Now such load combinations will be excluded from the results tables, even if requested. Likewise, load combinations for which no results are available (i.e., one of the contained load cases had not been run) will also be excluded from the tables. This was just a reporting issue and no analysis or design results were affected.
	225636	An incident was resolved where the formulas given in the report for the calculations of seismic loads for the Chinese code were not updated to the latest version. The program calculated loads were based on the updated formulas. This was a reporting error only and no results were affected.

*	Incident	Description
	225945	An incident was resolved where an error could occur when attempting to create a report based on an XML contents file that included an entry for a table with a filter or sort specified. Results were not affected. In particular, this could affect the built-in Chinese Summary Report, which uses filtering.

Graphics

Incidents Resolved

*	Incident	Description
	082215	An incident was resolved where deselecting the option Options > Graphics Preferences > Constant Line Thickness caused the grid bubbles to not display correctly in elevation views. This was a display issue only.
	088676 100326	An incident was resolved where a request to show the analysis mesh would result in the analysis mesh being shown with black lines that could not be seen on a black background. This was a rare occurrence and would happen only after returning to an undeformed shape after previously displaying a deformed shape with contours.
	222028	An incident was resolved where trying to animate a deformed shape for a load combination would sometimes cause an error condition. This only affected version 17.0.1 and only if the load combination being displayed had an index larger than the number of load cases. No results were affected.
	222080	An incident was resolved that corrected several minor issues related to DirectX graphics. No results were affected. Specific items include: (1.) Fine-Grid snapping did not work as expected. (2.) Perpendicular snapping did not work as expected. (3.) Sometimes the Reshape Object Tool would not work as expected. (4.) Floor midpoints could not be selected when using the Reshape Object tool. (5.) Member end-moment releases were shown as “x”, which was similar to joint objects. Now they are shown as circles, like they are when using standard graphics. (6.) Section assignments for membrane, deck, and plank floor objects were always being shown, even though they were turned off for the view using the Set Display Options command. (7.) Fixed supports were not shown similar to the way they are when using standard graphics. (8.) Walls were not included in the view when showing the shell analysis mesh. (9.) The contour color range was not displayed in plan views. (10.) Slab design results by FEM were not visible in either plan or 3D views. (11.) The animation for deformed shape became distorted when contours were shown.
	225128 227002	An incident was resolved where, for certain model files, the shell mesh with a floor opening would not display correctly in DirectX graphics. This was only a display issue and no results were affected.
	231062	An incident was resolved where a wall in plan view would not show correctly if only one node of the wall was at the plan level. No other views were affected and no results were affected.
	237125	An incident was resolved where shells with curved edges or with openings may not display correctly in a plan view when filled (command View > Set Display Options > Object Fill).

User Interface

Incidents Resolved

*	Incident	Description
	086562	An incident was resolved where the menu command File > Create Video > Multi-step Animation was available when it was not applicable, causing an error condition when clicked on. No results were affected.

*	Incident	Description
	215471	An incident was resolved where the software sometimes terminated abnormally when selecting by groups. This would occur if the selection operation was performed after performing the following sequence of operations: (1) An operation that automatically added a group, such as drawing auto cladding, then (2) undoing that operation, and finally (3) repeating the same or a similar operation. When this occurred, changes to the model since the last Save could be lost.
	219927	An incident was resolved where the text boxes for showing bar spacing would sometimes be greyed out and non-editable when displaying finite-element slab-design information.
	220631	An incident was resolved where the units for the “Precompression Displacement” and “Stop Displacement” values at rotational degrees of freedom in a link property of type “Damper – Friction Spring” were shown on the property definition form as degrees rather than radians. This was a typographical error only. The values specified were being treated as radians, consistent with all rotational displacements. The text on the form has been corrected. Results will not change from previous versions.
	220826	An incident was resolved where linear and nonlinear modal time history load case definitions with the initial conditions set as “Zero Initial Conditions – Start from Unstressed State” always showed the default mass source, which was incorrect if the selected modal load case was defined to use a mass source other than the default. This was a user interface issue only and did not affect results. The mass source actually used was shown in the analysis log (.LOG) file.
	209674	An incident was resolved where nonlinear material property stress-strain data for masonry materials that was set to be user-defined was always set back to the default parametric definition after clicking OK on the Nonlinear Material Data form (menu command Define > Material). Results were consistent with the parametric stress-strain curve, not the user-defined curve. Only masonry-type materials were affected.
	215526	An incident was resolved where a frame section was not displayed correctly in the Frame Section Property Data form (where section dimensions are specified) when any of the dimensions were small compared to the Auto Merge Tolerance specified for the model. A tighter tolerance is now used for showing the section. This was a display issue only. No results were affected.
	225866	An incident was resolved where deleting a diaphragm after user-type Wind Loads had been assigned to it would cause an error condition when the file was saved.
	227777	An incident was resolved where in some rare cases changing the interaction type of an interacting frame hinge property (P-M2, P-M3, P-M2-M3) and then assigning it to a frame would result in an error message and no assignment would be made. Other hinge property types were not affected.
	240482	An incident was resolved that corrected minor functional issues with the forms in the graphical user interface used to input data. No results were affected.
	229778	An incident was resolved where the labels for the “Symmetry Condition” options on the Frame Hinge Property Data form used to define interacting P-M2-M3 hinge properties changed and became incorrect when the “Hinge Specification Type” was changed from “Moment - Rotation” to “Moment - Curvature”, and vice-versa. This was a labeling issue only. The selected options themselves were not affected. This form is obtained using the menu command Define > Section Properties > Frame/Wall Nonlinear Hinges.
	229917 233605	An incident was resolved where an abnormal termination could occur when only wall objects were selected and the Merge Shell option was used. The Merge Shell operation only applies to floor (horizontal) objects. Any selected wall objects will now be ignored, but no error will be generated.
	232481	An incident was resolved where property modifiers defined in shell (slab, deck, wall) section properties retained the values entered on the data form even if the Cancel button was clicked. The values used in the model could be seen by going back into the form or by checking the database tables. Results agreed with the property modifiers as shown.

* Incident	Description
238797 238812 238816 238847 238887 238910 238948 238961 238993	An incident was resolved in which the program could terminate when starting up and checking for the latest version. This was related to interaction with a web endpoint that was being blocked by OpenDNS, an outside vendor.
240629	An incident was resolved where the software could terminate abnormally when trying to modify the rebar for a layered shell property. This could occur when the "Size and Spacing are the Same for All Rebar" option was selected and the number of rebar layers was 1.

Database Tables

Incidents Resolved

* Incident	Description
206640	An incident was resolved where the export of analysis results to Access format failed for a particular model containing multiple towers.
220183 221529 221555 222128 226255	An incident was resolved where the database table "Assembled Joint Masses" was incorrectly reporting the masses for three of the joints in the table. For these joints, the total mass of the structure was being reported, with each joint reporting the total mass in a different direction. The affected joints could be in any order in the table, and were easily identifiable as the joints having the largest masses in the table. For the most part, this was a reporting error affecting this table only. However, the incorrect mass values were also being used to compute the seismic mass that is then used to determine the performance point when plotting static pushover curves. The net effect was to nearly double the seismic mass. Only the performance point was incorrect. No other results were affected.
216801 222212	An incident was resolved where an error message was generated when trying to display or export the table for concrete material properties whenever time-dependent properties (creep, shrinkage, stiffness) had been defined using the AS3600-2009 code. No results were affected.
209797	An incident was resolved where, in certain cases, requesting a table of results for a load combination of type algebraic that contained other load combinations within its definition would result in an error condition. When this occurred, the requested results were not available.
214136 232766	An incident was resolved where the weight of the non-prismatic frames reported in the material list table may be incorrect when the non-prismatic section included absolute-length segments. The reported weight was correct if the non-prismatic section included only variable-length segments. This was an issue with the weights shown in the material list tables only and did not affect the weights used in analysis or design.
223523 224975	An incident was resolved where exporting a Table Set to Excel did not properly convert for units when multiple tables were present in the Table Set. This error did not occur for individual tables. In the new version, Tables Sets have been replaced with Named Table Sets, and the unit-conversion issue has been resolved. Units for export to Access, Text, and XML are consistent and requested at the time of export. Units for Excel may be consistent or inconsistent and are based on Display Units currently set for the model.

Data Files

Incidents Resolved

*	Incident	Description
	219755 219949 222075	An incident was resolved where opening a file saved in ETABS versions 17.0.0 or 17.0.1 would generate an error message "Exception Type: Unknown version encountered" for some models. This error was limited to files where composite column design overwrites were present.
	220205	An incident was resolved where an unsymmetrical steel shape used with a "Mirror about..." option selected in a Section Designer section would not keep that option once exported and imported through the model text file (.e2k or .Set).
	233859	An incident was resolved where the shear area for pipe sections imported from .XML files was incorrect for the following section libraries: AISC13, AISC13M, AISC14, AISC14M, AISC15, AISC15M, AISCLRFD2, APCLRFD3, ASTM A1085, AusNZ8, CISC9, CISC10, Euro, Nordic. Analysis results were based on the shear areas shown in the frame section definitions.
	239470	An incident was resolved where the import of Section Designer sections from text files (.E2K, .SET) was not working properly. This only affected creation of fibers for fiber hinges. No other results were affected.

External Import/Export

Incidents Resolved

*	Incident	Description
	090432	An incident was resolved where importing a DXF file as an architectural layer into a multi-tower model could generate error messages during the import process, and further error messages could be generated when later changing views to 3D or moving up and down story levels in a plan view. No results were affected.
	217920	An incident was resolved which affected the export from ETABS to Revit of line objects that had joint offsets specified in the line-object local coordinate system. These line objects were exported to Revit as if the coordinates of the joint offset had been specified in the global coordinate system. When this occurred, the discrepancy with the ETABS model was visually obvious in the Revit project. This occurred with all versions of ETABS capable of exporting models to Revit. Line objects whose joint offsets were specified in the global coordinate system were correctly exported.
	220038 221427 222526 223074 224851	An incident was resolved where response-spectrum load cases exported from ETABS v17 to SAFE reported zero base shear in SAFE because the scale factors for modal mass and modal stiffness were not being exported from ETABS v17. Only versions v17.0.0 and v17.0.1 of ETABS were affected.

Application Programming Interface (API)

Incidents Resolved

*	Incident	Description
	204541	An incident was resolved for the Application Programming Interface (API) where the functions cFrameObj.SetLoadPoint, cFrameObj.SetLoadDistributed and cFrameObj.SetLoadTemperature always added load rather than replacing the existing load regardless of whether the Replace argument was set to True or False.
	208487 233700	An incident was resolved for the Application Programming Interface (API) where the function cAnalysisResults.AreaStressShell was only extracting a number of results for each load case or load combination requested equal to the number of points in the selected area elements even though more results were available (such as for Max/Min or stepped results).

*	Incident	Description
	216502 221460	An incident was resolved for the Application Programming Interface (API) where the wrong unique name could be returned for certain objects in the following cases: (1.) Function cAreaObj.GetNameFromLabel could return the wrong unique name when the model contained multiple types of area objects (floors, walls, null areas). (2.) Function cAreaObj.GetNameFromLabel could return "null" for the name of wall objects if the model contained only wall and null-area objects (no floors or slabs). (3.) Function cFrameObj.GetNameFromLabel could return the wrong unique name when the model contained multiple types of frame objects (beams, columns, braces).
	219227	An incident was resolved for the Application Programming Interface (API) where the functions LoadCases.*.GetInitialCase would return "None" for linear, nonlinear static, staged construction, or direct-integration time-history load cases that continued from a nonlinear staged construction load case. Also the function LoadCases.StaticNonlinearStaged.GetStageData_2 was unable to return the correct ObjectType and ObjectName values when the Object Type defined in the stage operation was "Story".
	219803	An incident was resolved in which the API functions cCombo.GetCaseList, cPropFrame.GetAllFrameProperties, cPropFrame.GetPropFileNameList, and cConstraint.GetDiaphragm would generate an error when the API script referenced ETABS2016.dll and was used with the ETABS v17 program. This issue is corrected in the new release, but not for the corresponding case where API scripts reference ETABSv17.dll from ETABS v18.
	222392	An incident was resolved for the Application Programming Interface (API) where the function SapModel.File.OpenFile would produce error and warning messages when opening some .e2k and .Set text files.
	226841	An incident was resolved to correct an error in the documentation for the Application Programming Interface (API) where the sample code provided for the following four functions was incorrect and would generate an error if run: cPropArea.GetShellLayer Method, cPropArea.GetShellLayer_1 Method, cPropArea.SetShellLayer Method, and cPropArea.SetShellLayer_1 Method. The functions SetShell and AddQuick in the examples have now been replaced by "SetWall" and "AddMaterial" instead. This was a documentation error only. No results were affected.
	226876	An incident was resolved for the Application Programming Interface (API) where the functions cAnalysisResults.JointDrifts and cAnalysisResults.StoryDrifts were returning drift results that were converted for length units, when in fact the returned results should be dimensionless. Results for these values presented in the tables were correct, and no other results were affected.
	229109	An incident was resolved for the Application Programming Interface (API) where the input arguments dx, dy, dz for the function cEditGeneral.Move were not being converted for the present units as specified through the API, but instead were always applied as being given in database units.
	239662	An incident was resolved for the Application Programming Interface (API) in which the function GetLateralBracing would always fail and return a value of 1.
	239653	An incident was resolved for the Application Programming Interface (API) where the function cPropRebar.GetNameListWithData did not convert the diameter and area parameters to present units. Instead the values were returned in the database units used when the model was first created or imported.
	239872	An incident was resolved for the Application Programming Interface (API) where the function cJointObj.GetPanelZone always returned an error code of 1 for joints that had a panel zone assigned to them.

**Documentation
Incidents Resolved**

*	Incident	Description
	217119 224671	A documentation error has been fixed for ACI 318-14 shear wall boundary zone design where the limit specified in ACI 318-14 equation 18.10.6.2(a) should be greater than 0.005 while previously it was documented as less than 0.005. Similarly, ACI 318-11 equation ACI 21.9.6.2(a) should be greater than 0.007 while previously it was documented as less than 0.007. These were only documentation errors and design results were not affected.
	220507	An incident was resolved where the code examples in the API (Application Programming Interface) documentation for functions <code>cDesignConcrete.GetSummaryResultsBeam</code> and <code>GetSummaryResultsColumn</code> used incorrect syntax and hence could not be run when implemented in a client application. The examples have been updated to run correctly. The behavior of the software has not been changed.
	219067	An incident was resolved where the EC3 2005 Steel Frame Design manual was missing Chapter 10, which contained all the seismic provisions. This has been corrected. This was a documentation error only, and no results were affected.
	221187	The documentation for API (Application Programming Interface) function <code>cFrameObj.SetLoadDistributed</code> has been corrected for the definition of argument <code>Dist2</code> . The correct definition should be the distance from the I-End of the frame object to the end of the distributed load. Previously it referred to the distance from the J-End of the frame object to the start of the distributed load. The behavior of the function was correct and consistent with the now revised description.
	208934	A documentation error has been fixed for Section 2.2 of the concrete frame design manuals for ACI 318-14, ACI 318-11 and ACI 318-08 codes. Section 2.2 now clarifies that parameters for Special Seismic Load Effects are specified in the concrete frame design preferences instead of the Define menu. This was a documentation update only.
	217089	Documentation for superseded design codes AS 360-2001, CSA A23.3-04, and Hong Kong CP 2004 have been removed from the RC and PT slab design manuals.
	216872	The Help topic "Line Springs (Form: Frame Assignment - Line Springs)" was updated to indicate that compression-only line springs apply to horizontal and vertical frame objects, not just vertical frame objects as previously stated.