ETABS v19.0.0 Release Notes

Notice Date: 26-October-2020

This document lists changes made to ETABS since v18.1.1, released 11-February-2020. Items marked with an asterisk (*) in the first column are more significant.

Analysis

Enhancements Implemented

*	Ticket	Description
*	1930	An enhancement has been made to add short/long-term cracked-deflection calculations for concrete floors. The nonlinear static load case now allows for this option to be triggered. The rebar used for the cracking calculation is based on either provided rebar, the last slab and beam design, or a quick rebar overwrite provided for this purpose. A new assignment is available to specify which floor objects are considered for cracked deflection analysis. Floors not so assigned are assumed to be uncracked during a cracking analysis. For long-term deflections, the creep and shrinkage coefficients can be taken from the material properties and from the time since load application, or the user can give overwriting coefficients for the particular load case. This option is only available in ETABS Ultimate Level.
*	4154	The number of processors that can run load cases in parallel in a given session is now limited to eight. It is still recommended to limit the number of parallel load cases to the number of physical cores on the machine if less than eight. Load cases for a given model can still be run on multiple machines and combined into a single set of results on one machine.
	4172	Optimizations have been made to speed up the plotting of deformed shapes and generation of .AVI files for multi-stepped load cases. This will primarily affect direct-integration and staged construction load cases in models with a large number of elements.
*	4350	A change was made to clarify the behavior of a link element when the effective stiffness is used for stiffness-proportional damping - this occurs when "Stiffness Used for Stiffness-proportional Viscous Damping" in the Link Property Data (Define menu > Section Properties > Link/Support Properties) is set to “Effective Stiffness (KE)" or if a linear direct-integration time-history load case starts from the unstressed state (zero initial conditions). When this setting is used, the effective stiffness will be used for stiffness-proportional damping at all times, even if the link has zero nonlinear stiffness and zero nonlinear force. Previously, nonlinear direct-integration time-history load cases set the damping due to effective stiffness to zero when both the stiffness and force in the link was zero, and linear direct-integration time-history load cases set the damping due to effective stiffness to zero if the same condition was met in the initial condition (either the initial state or the end of a nonlinear load case that the linear load case continues from). This change does not affect nonlinear direct-integration time-history load cases where the link has "Stiffness Used for Stiffness-proportional Viscous Damping" set to "Initial Stiffness" or "Tangent Stiffness".
	4359	The initial rigid stiffness of parametric PMM hinges was improved and is now based on the stiffness of the frame object containing the hinge. This change is intended to prevent excessive stiffnesses which can adversely affect the convergence behavior of the model and will not significantly affect analysis results for well-conditioned models.
*	4519	An enhancement has been made to speed-up the recovery of analysis results, with particular improvement for hinge response. In addition, new options are now provided to control advanced parameters affecting analysis, design, and response recovery. For the command Analyze > Advanced SAPFire Options, the new parameters are: • Number of (Internal) Threads for Analysis. Using more threads (up to number of physical cores) will speed-up analysis, but also uses more RAM (memory) to be efficient.

*	Ticket	Description
		• Maximum Size for Response Files (MB). This may affect certain backup or archival systems. For the command Analyze > Advanced Design and Response Recovery Options: • Number of Threads for Design. Using more threads (up to number of physical cores) will speed-up design and response recovery, but also uses more RAM (memory) to be efficient. • Number of Threads for Response Recovery. Using more threads (up to number of physical cores) will speed-up design and response recovery, but also uses more RAM (memory) to be efficient. • Use Memory-mapped Files for Design and Recovery. This can speed-up retrieval of saved analysis results. Systems with Solid State Drives (SSDs) and adequate RAM (memory) will benefit the most. • Allow Model Differences when Merging Results. Normally this is not recommended, but it can be permitted, when merging results from models run on different machines.
*	4906	An enhancement has been made to speed-up repeated solving of the stiffness matrix when using the Multi-threaded Solver option. This should significantly increase the speed of nonlinear static, nonlinear multi-step static, and nonlinear direct-integration time-history load cases, especially when using Newton-Raphson iteration and/or event-to-event stepping. Staged-construction load cases will similarly benefit within stages and between stages where the structure does not change. Eigen modal load cases will benefit when many frequency shifts are performed, and linear direct-integration time-history loads cases will benefit when the time-stepping is not constant. Larger models will show more benefit. This change does not affect the default Advanced Solver or the Standard Solver. Use the command Analyze > Advanced SAPFire Options to select the Multi-threaded Solver.
	4925	An enhancement was made to the Nonlinear Parameters form (Define menu > Load Cases) to provide a clearer selection of the solution scheme to be used for a nonlinear static, staged-construction, or nonlinear direct-integration time-history load case. The previous options "Use Iteration" and "Use Event-to-Event Stepping" have been merged into one parameter "Solution Scheme", which specifies the overall solution method used in the analysis. The new Solution Scheme options are: (1.) Iterative Only, equivalent to “Use Event-to-event stepping” = No and “Use Iteration” = Yes (2.) Iterative Event-to-Event , equivalent to “Use Event-to-event stepping” = Yes and “Use Iteration” = Yes (3.) Event-to-Event Only , equivalent to “Use Event-to-event stepping” = Yes and “Use Iteration” = No Only the nonlinear parameters relevant to the selected solution scheme will be shown in the form. It is suggested to select the solution scheme first before specifying the nonlinear parameters, since the nonlinear parameters may be reset when the solution scheme is changed. See also Ticket 5110 for additional parameters and changed behavior for the Event-to-Event Only solution scheme.
*	5110	An enhancement was made to nonlinear static and nonlinear direct-integration load cases that use the "Event-to-Event Only" solution method to better handle how equilibrium unbalances are passed to each subsequent step. This change may affect results for non-iterative event-to-event analyses where the relative unbalance reported in the .LOG files are large, but the change is expected to reduce the relative unbalance in the analyses. Additionally, four parameters are added to the Nonlinear Parameters form (Define menu > Load Cases) to allow more user control for the Event-to-Event Only method: (1.) Stop Analysis when Maximum Events per Step is Exceeded. Selecting Yes for this option enables a new feature where the analysis will terminate if the specified Maximum Events per Step is reached in an analysis step. (2.) Minimum Event Step Size. The minimum event size that is allowed during analysis. When the Maximum Null Events per Step is exceeded, events that are smaller than this value are increased and counted as Null Steps.

*	Ticket	Description
		(3.) Maximum Null Events per Step. If this is set to a non-zero value, this is the number of events smaller than the specified Minimum Event Step Size allowed in an analysis step. If set to zero, the Minimum Event Step Size will not be enforced. (4.) Use Correction Step for Large Unbalance. The options are "None", "Every Step", and "Last Step". When "Every Step" is specified, an additional correction step with no applied load will be taken at the end of any step where the relative unbalance is larger than 1.0e-4 to resolve unbalances in the model before moving to the next step. If "Last Step" is specified, this correction step will be taken at the end of a Nonlinear Static load case or at the end of every stage for a Staged Construction load case. This parameter is only applicable to Nonlinear Static load cases. Event-to-Event Only analysis can be defined by setting the "Solution Scheme" option to "Event-to-Event Only" in the Nonlinear Parameters form. See also Ticket 4925 for changes to the Nonlinear Parameters form.
*	5224	The biaxial friction-pendulum isolator link element now uses tangent stiffness for iteration when running nonlinear static, staged-construction, and nonlinear direct-integration time-history load cases. Previously this isolator used the initial stiffness for iteration. Using the tangent stiffness is generally more efficient, although convergence behavior is always problem dependent. This change may cause some differences in analysis results from previous versions. These will usually be within the convergence tolerance except for numerically sensitive or ill-conditioned models. Results for Verification Example 6-011, which can vary with different processors, are slightly affected by this change. The tension-compression (T/C) friction-pendulum isolator already was using the tangent stiffness for iteration, while the triple friction-pendulum isolator will continue to use initial stiffness for iteration. Note that the use of tangent stiffness for the biaxial and tension-compression friction-pendulum isolators also makes them suitable for pure event-to-event solution methods, but the same is not true for the triple friction-pendulum isolator using initial stiffness.

API

Enhancements Implemented

*	Ticket	Description
	2153	The Application Programming Interface (API) functions cFrameObj.SetPier, cFrameObj.SetSpandrel, cAreaObj.SetPier and cAreaObj.SetSpandrel have been enhanced so that they can be used to assign pier/spandrel labels while the model is locked. Additionally, the methods in the cPierLabel and cSpandrelLabel interface can be used to add, remove, or modify pier and spandrel labels while the model is locked. Note that relabeling the piers and spandrels when the model is locked will delete the design results, which is now the same behavior as when relabeling them using the graphical user interface.
	3898	The examples included in the documentation for the Application Programming Interface (API) now have updated instructions for attaching to a running instance of ETABS. In addition, the C++ example has also been modified to provide clearer guidance on creating a forward-compatible client.
	4071	An enhancement was made to the Application Programming Interface (API) to add the functions SapModel.PropArea.GetShellLayer_2 and SetShellLayer_2, which get and set the definitions of all the layers in a layered-shell property. These new functions include the option to get and set the material behavior type, either Directional or Coupled, which was not available in the now superseded functions GetShellLayer_1 and SetShellLayer_1. If the function SetShellLayer_1 is used, the material behavior type defaults to Directional for all layers. Note that the Coupled option only applies to concrete materials, and is ignored without error for other materials.
*	5190	The ETABS API can now be used to start and/or connect to a running instance of ETABS on a Remote Computer that is running the new API Service for distributed computing. Simultaneous runs can be started on multiple Remote Computers using an API script or plug-in, and results can be merged to the Main Computer programmatically, without user

*	Ticket	Description
		intervention, as they become available. Possible applications that can be sped up by distributed computing include parameter studies, Monte Carlo simulations, or performance-based design requiring a large number of load cases to be run.

Database Tables

Enhancements Implemented

*	Ticket	Description
	1316	An enhancement was made whereby the tables for general pier Section-Designer sections, defined through the shear wall design module, can now be generated. The data for these sections can now also be edited interactively.
*	2684	An enhancement was made to enhance the Joint Coordinates and object connectivity tables and to expose them for interactive database editing.
*	3425	An enhancement was performed to allow the name of the database file that is automatically saved after the analysis is run to include a path. Also, multiple files can now be specified to be automatically saved after the analysis is run.
	3919	An enhancement was made to add the database table "Analysis Options - Analysis Model for Nonlinear Hinges" under Model Definitions > Options and Preferences > Analysis Options. This database table reflects the options available in the Analysis Model for Nonlinear Hinges form, which is accessible by going to the Analyze menu > Analysis Model for Nonlinear Hinges.
	5312	The Interactive Database (Edit menu>Interactive Database) has been enhanced so the tables "Pier Label Definitions", "Spandrel Label Definitions", "Frame Assigns - Pier Labels", "Frame Assigns - Spandrel Labels", "Area Assigns - Pier Labels", and "Area Assigns - Spandrel Labels" can be edited while the model is locked.

Design – Composite Beam

Enhancements Implemented

*	Ticket	Description
	4213	Composite beam design has been enhanced for the following items: (1.) ETABS does not take into account floor objects with sections of type Slab when computing beam composite section properties, and when this condition affects the design of a beam, ETABS will post a warning message to that effect in the Interactive Design form and in the calculations output. However, ETABS previously would keep posting that message even after the sections of the involved floor objects had been changed to type Deck. This had no effect on the design of the beams. The message is now posted only when relevant. (2.) In the Interactive Composite Beam Design and Review form, the percentage of composite action corresponding to the maximum number of shear studs that fit on the selected beam section is now always one of the available choices. (3.) An enhancement was made to user reports that feature composite beam calculations. When some of the beam designs fail, the calculations reported for the affected beams are based on the percentage of composite action corresponding to the maximum number of shear studs that will fit. This is consistent with the stress ratios displayed when the Start Design/Check command is run and with the contents of user reports featuring composite beam summary results.
	5205	An enhancement was made to the design of composite beams per the CSAS16-14 code to add a new Composite Beam Design Preference that specifies whether shear studs are located in the centers of deck ribs or off-center. When the deck is perpendicular to the beam being designed, specifying the off-center location typically increases the shear stud capacity.
	5314	An enhancement was made to composite beam design so that fewer shear studs may now be determined as required, with the controlling stress ratio being closer to 1.0 or to the limit specified in the composite beam design preferences. This change will not affect most designs, but is more efficient when it does occur.

Design – Concrete Frame

Enhancements Implemented

*	Ticket	Description
*	2667	An enhancement has been made in which the beam design can now consider prestressing tendons. This feature is only available in the Ultimate level of ETABS.
*	3159	An enhancement was added for CSA A23.3-14 concrete frame design code where torsional longitudinal rebar needed at tension and compression side is now designed when combined axial force, torsion and shear exceeds the code allowable limit. Torsional longitudinal rebar is reported at top and bottom separately when tension or/and compression caused by combined axial, shear and torsion are more than flexural longitudinal rebar.
*	3486	An enhancement has been made to add the new design code “ACI 318-19” for designing concrete frames.
	4997	An enhancement has been made for steel frame design code "Chinese 2018" where now the stability check equation information will be displayed with most of the terms as zeros and with the calculated phiB factor when the design preferences or overwrites specify "Ignore beam phiB" as "Yes". Previously the stability check information was shown in this case for the phiB factor set to 1.0. The new implementation for the "Chinese 2018" code is similar to that of "Chinese 2010".
*	5318	An enhancement has been implemented to add composite column design according to AISC 360-16.

Design – Shear Wall

Enhancements Implemented

*	Ticket	Description
	2561	An enhancement was made to AS 3600-2018 shear wall design to account for the ductility requirements in shear walls. For ductile shear walls, boundary zone design conditions are now checked and pier shear design forces are amplified.
*	3488	An enhancement has been implemented to add shear wall design according to ACI 318-19.

Design – Slab

Enhancements Implemented

*	Ticket	Description
*	2573	Concrete slab design has been added per the Australian AS 3600-2018 code for reinforced-concrete and PT slabs.
*	3489	An enhancement has been implemented to add slab design according to the ACI 318-19 code.
*	4904	The National Annexes for Bulgaria, Germany, Ireland, Poland and Portugal have been added as an option to the Eurocode 2-2004 reinforced-concrete and PT slab design.

Design – Steel Frame

Enhancements Implemented

*	Ticket	Description
	4031	An enhancement was made to the Russian steel frame design code SP 16.13330.2017 where now the maximum elastic shear-stresses and the maximum elastic combined-stresses are checked by comparing them with Rs*GammaC and Ry*GammaC/0.87, respectively, at the critical points per the code (SP 16.13330.2017 Eqn. 42, Eqn 44, Section 8.2.1.) for the following sections: Singly-symmetric I-shape, Channel, T-shape, Double Angle, and Box sections per the code (SP 16.13330.2017 Eqn. 42, Eqn 44, Section 8.2.1.). Previously, those checks were performed only for the following sections: Doubly-symmetric I-shape, Double Channels, Rectangular shape, Pipes, and Circles. These checks are not performed for any Section Designer, General, or Angle sections. The average shear checks based on Q3/Av33 and Q2/Av22 are checked by comparing them with Rs*GammaC for all shapes as a simplified measure. These simple checks are still done. The previous results were slightly unconservative for the new considered shapes.

*	Ticket	Description
	4033	An enhancement has been made in the Russian steel frame design code SP 16.13330.2017 to distinguish between the two phi_c factors which are reported: (1) phi_c from Table 21, and (2) the phi_c factor from clause 7.1.5 to be used in Eqn.10. The second one is calculated only for channel sections. This is now clarified by specifying the corresponding clauses in the header of the table. This was a reporting only issue.
	4044	An enhancement has been made to the Russian steel frame design code per the Change 2 of SP 16.13330.2017 (SP 16.13330.2017 7.1.3, Table 7), where for the buckling check of I-shaped sections, the following buckling curves are now used: (i) The buckling curve "b" is used for the major direction buckling for all rolled or welded, doubly-symmetric I-shapes except for case (ii). (ii) The buckling curve "a" is used for the major direction buckling of all rolled, doubly-symmetric I-shapes with the depth of the section along the web direction more than 500 mm. (iii) The buckling curve "c" is used for the major direction buckling for all rolled or welded, singly-symmetric I-shapes, and (iv) The buckling curve "c" is used for the minor direction buckling for all rolled or welded, singly-symmetric or doubly symmetric I-shapes. The buckling curves are used in the determination of axial compression capacity per clause SP 16.13330.2017 7.1.3.
	4995	An enhancement has been made to the steel frame design code "Chinese 2018" where now the gamma_RE(S) stability coefficient is always taken as 0.8 for the beams. Previously it was taken as 0.8 for beams treated as flexo-compression members and 1.0 for flexural-only beams. This factor is a multiplier on the stress ratio terms in the stability-based interaction equation. Previously, the design was slightly conservative.
	5272	An enhancement has been made to the steel frame design code "Chinese 2018" in which the Chinese steel grade Q345 has been replaced with Q355. The yield strengths of the grades Q355, Q390, Q420, and Q460 have also been changed. Now the calculation of epsilon_k [epsilon_k = sqrt(235/fy)] uses the yield strength fy from the name of steel grade (355, 390, 420, and 460 MPa) and is not affected by plate thickness.

External Import and Export

Enhancements Implemented

*	Ticket	Description
*	2391	An enhancement has been made to Implement bi-directional transfer of data for reinforcement between ETABS and Revit. This feature requires CSiXRevit v2020 or later.
*	3356	An enhancement was made affecting the import of IFC files that describe structures in "structural analysis view". When importing IfcStructuralPointConnection, IfcStructuralCurveMember, and IfcStructuralSurfaceMember objects whose ObjectPlacement attribute referred to an IfcLocalPlacement that in turn had a PlacementRelTo attribute, ETABS previously would not take into account that last attribute. The PlacementRelTo attribute indirectly referred to in IfcStructuralPointConnection, IfcStructuralCurveMember, and IfcStructuralSurfaceMember objects is now taken into account when these objects are imported in ETABS. Not taking that attribute into account typically shifted the entire structure from its location in the originating software, but did not affect the analysis results. The import of IFC files that describe structures in "architectural coordination view" was not affected. The previous behavior affected all versions of ETABS capable of importing IFC files that describe structures in "structural analysis view". Another related enhancement was also made. Objects imported from all IFC files are now imported in local coordinates relative to the site coordinate system, instead of in global coordinates. The location and rotation of the site coordinate system are stored in the ETABS model. If the structure is re-exported, the location and rotation of the site are included in the export, and the ObjectPlacement attribute of the exported IFC objects is exported as a compound transformation that explicitly refers to the location of the site and

*	Ticket	Description
		to the relative coordinates visible in the ETABS model.
*	4217	An enhancement was made to allow the export of ETABS model files to Perform®3D v7. Most structural objects, properties, loads, and load cases will be exported and translated. The ETABS model to be exported must meet modeling requirements of Perform®3D. No auto-meshing of either frames or shells, or edge constraints should be used. The shell objects should be well conditioned quadrilateral shapes. The material stress-strain curves should be simplified to meet Perform®3D criteria, and the same applies to hinge force-displacement and moment-rotation curves. The two products are different in the way they handle certain items. A report is available of what was translated, and users should verify and modify the exported model as necessary.
*	4961	An enhancement was made to allow the import of Perform®3D v7 model files into ETABS. Most elements, cross-section properties, mass, loads, and load cases will be imported and translated. Certain types of nonlinear components will be transferred, limited currently to moment hinges for beams, P-M2-M3 hinges for columns, and shear-wall inelastic-fiber sections. The two products are different in the way they handle certain items. Upon importing, users will be presented with the option to select story levels for the ETABS model. A report is available of what was translated, and users should verify and modify the imported model as necessary.
	5127	An enhancement to the import of DXF files containing polylines has been implemented. Polylines that are not closed are now imported as a series of frame objects. Closed polylines are still imported as area objects, like before.
	5168	An enhancement was made to the import of .EXR files from Revit using CSiXRevit. The Revit orientations of framing elements with angle, double-angle, tee, and channel sections are now imported. Note that this enhancement requires that the .EXR files be created in Revit 2021 or later.
	5218	Non-rectangular, trapezoidal walls can now be imported into ETABS when exported from Revit using CSiXRevit. Previously such walls were ignored when importing from Revit.

Graphics

Enhancements Implemented

*	Ticket	Description
*	3801	Enhancements have been made to DirectX graphics by using Direct2D in conjunction with Direct3D. Specifically: (1.) Plan and elevation (XY, XZ, YZ and developed) views are generated much faster and are clearer. (2.) Line quality for dashed lines, wide lines and selection lines during editing, drafting and display has been improved for both 2D and 3D views. (3.) Windows True Type fonts, including foreign language fonts, are available in both 2D and 3D views. Previously only Arial fonts were available and did not cover some languages.

Installation and Licensing

Enhancements Implemented

*	Ticket	Description
*	4737	ETABS now utilizes cloud licensing by default, allowing access to the license by multiple users and/or from multiple machines. The number of simultaneous users corresponds to the number of licenses owned. Cloud licensing requires connection to the internet while using the software, either directly or through a proxy. Connection to a company network or VPN is not necessary. Licenses can be checked out for a limited time period to allow use while disconnected from the internet. Legacy licensing options (Standalone and Network) are still available upon request.
*	4932	The version number has been changed to 19.0.0 for a new major release.

Loading

Enhancements Implemented

*	Ticket	Description
	1087	An enhancement was made in Eurocode 8:2004 auto seismic load patterns and response spectrum function for Portugal National Annex where S factor now incorporates the ground acceleration ag when ground acceleration is greater than 1.0 m/s^2.
*	2142	An enhancement has been implemented to add auto lateral loading per the NTC 2018 code. This includes auto wind, auto seismic, and the response-spectrum function.
	2234	An enhancement was added for AS/NZS 1170.2 auto wind load pattern where additional terrain categories 1.5 and 2.5 have been added as per Amendment 2.
	2464	A new option has been added to the Chinese wind loads to Ignore Vibration Effects. When selected, this assumes that BetaZ is 1 (unity) and turns off all calculations for determining BetaZ. This also turns off across-wind and torsional-wind effects.
*	5108	An enhancement has been implemented to add auto lateral loading per the Korean KDS 41 17 00:2019 code. This includes auto-seismic loading and the response-spectrum function.
	5154	An enhancement was made to restore the UBC 97 auto seismic and response spectrum function as these are still used in some jurisdictions. Users are cautioned when using this old loading code with the newer design codes. Loading combinations with earthquakes may have to be adjusted.

Results Display and Output

Enhancements Implemented

*	Ticket	Description
	3587	An enhancement has been made to improve the image quality of named displays used in reports.
*	4357	An enhancement was added for concrete beam rebar display where rebar envelope diagram is now available for flexural, shear, torsion design and PT stress rebar via the command Design > Concrete Frame Design > Display Beam Design.

Structural Model

Enhancements Implemented

*	Ticket	Description
*	1307	An enhancement has been made for the import of Revit projects into ETABS. When a model initially exported to Revit or initially imported from Revit is later re-imported from Revit, ETABS reports the changes from the original ETABS model that are detected while updating the model in a log file with an .EXRLog extension.
*	2435	An enhancement has been made to update the special requirements applied to steel beam-column connections that use the latest version of the proprietary SidePlate system.
*	2501	An enhancement was made to store analysis-model-generation warnings so they are available in the Analysis Message dialog from session to session.
	3436	The Korean common rebar set is now available when defining reinforcing bar sizes.
	3577	An enhancement was made to frame hinge assignments so that a hinge can be assigned at an absolute distance from either end of the clear length of an object in addition to the existing option of relative distance. Frame hinge assignments can be made in the Assign Frame Hinges form (Assign > Frames > Hinges) and through interactive database editing of the Frame Hinge Assigns tables. Correspondingly for the Application Programming Interface (API), the function cFrameObj.GetHingeAssigns has been deprecated and replaced with cFrameObj.GetHingeAssigns_1 which will extract the hinge location as either a relative distance or an offset distance based on the assignment.
*	5021	A new option has been added to frame sections definition to allow the flanges of concrete T and L sections designated as beams to be ignored from the calculation of area, weight and mass. This avoids duplication of area, weight and mass from adjoining slabs. This option does not affect steel or other non-concrete T and L sections, nor does it affect concrete T and L sections designated as column sections. Concrete beam design for these sections is

*	Ticket	Description
		not affected. Older files opened in the new version will default the option to not ignore the flange area so that results will not change.

User Interface

Enhancements Implemented

*	Ticket	Description
	1200	An enhancement was made to allow dimension lines to be added even when the model is locked after analysis.
	3052	An enhancement was made to the user interface where the tab order for controls on all forms was set so as to maximize data input efficiency for users who primarily navigate the input fields on forms by using the Tab key on their keyboard.
	4316	An enhancement was implemented where the menu command Define > Modal Cases has been removed and modal load case definitions are now combined with the other load case types using the menu command Define > Load Cases.

Analysis

Incidents Resolved

*	Ticket	Description
*	3858	An incident was resolved where the nonlinear stiffness of Damper-Exponential type links was being included in linear and modal load cases that used the stiffness from a nonlinear load case, including P-delta load cases. This issue only affected ETABS versions 18.0.0 to 18.1.1 when the option "Stiffness Used for Linear and Modal Load Cases" in the Damper-Exponential type link property was set to "Effective Stiffness from Zero, Else Nonlinear" or "Nonlinear Stiffness". This had the effect of stiffening the model for the affected load cases. Now the stiffness will be set to zero for linear and modal load cases using the stiffness from a nonlinear load case unless the option "Effective Stiffness" is chosen for Damper-Exponential type link properties.
*	4364	An Incident was resolved where a steel or concrete parametric PMM hinge with an "Elastic Perfectly Plastic" curve shape and with strength loss enabled would use axial force and moment values at points D and E that were different than the values specified in the Force-Deformation Data form. The axial force and moment values at points D and E used for analysis were equal to the force or moment at point U multiplied by the force parameter "Ratio Point D/Point B" and the force parameter "Ratio Point U/Point B", which are hidden for a "Elastic Perfectly Plastic" but have a default value of 1.5. This issue only affected the force and moment values at points D and E for Elastic Perfectly Plastic parametric PMM hinges, and it affected all nonlinear load cases.
	4510	An incident was resolved where adding a copy of a load case that had results inadvertently deleted the original load case's results.
*	4926	An issue was resolved where a Nonlinear Static load case with the Load Application Control option set to "Displacement Control" could produce results that were dependent on the scale factor of the applied load. This occurs if the model had shell objects, the Geometric Nonlinearity Parameter was set to "P-Delta", and the Nonlinear Parameters options "Use Event-to-event Stepping" and "Use Iteration" were set to "Yes" and "No", respectively. This issue only occurred for non-iterative event-to-event analysis and, generally, a larger load scale factor would result in smaller equilibrium error at each step. Very small differences may now still be exhibited for sensitive models, but they are not of engineering significance.
	4936	An incident was resolved where, when a Parametric P-M2-M3 hinge yielded purely in the axial direction, analysis results became unavailable and would show as "NaN". This issue was rare because it required the Parametric P-M2-M3 hinge to have zero moment and zero rotation in both M2 and M3 directions when yielding.
	5365	An incident was resolved where a model that had all three of the following conditions may have reported an instability and/or lost accuracy in the constraints due to coupling between link fixity constraints and rigid diaphragm constraints: (1.) The model had hinges located at the ends of frame objects (relative distance of 0 or 1) and the frame objects did not have end offsets. (2.) The option was set to model hinges as separate link elements (Analyze > Analysis Model for Nonlinear Hinges). (3.) Rigid diaphragms applied to the frame objects with hinges and affected at least one of the degrees of freedom that the hinges do not activate (e.g., the axial degree of freedom of a beam is included in the rigid diaphragm but not in an M3 moment hinge). This issue was not common because end offsets are assigned to frame objects by default, and hinges are often assigned at their own midpoint location rather than at the extreme ends of the frame object. Semi-rigid diaphragms were not affected. Now when this conflict is detected, a large stiffness is used instead of a fixity condition for the link element.

API

Incidents Resolved

*	Ticket	Description
	3597	An incident was resolved for the Application Programming Interface (API) where the function PropFrame.ImportProp would produce an error message when the function was called referencing an xml property file that was not already added to the model through the graphical user interface or by a previous call to the API function cPropFrame.GetPropFileNameList.
	3924	An incident was resolved for the Application Programming Interface (API) where the GetAllFieldsInTable function sometimes did not return all fields. This typically occurred for output tables where fields, such as StepNum, would not be populated based on the selected load cases/combinations and output options. Now the function returns all available fields for the output tables regardless of whether or not they are needed.
	4004	An incident was resolved for the Application Programming Interface (API) where the StepType and StepNum output of the function cAnalysisResults.ModelPeriod was not correctly returned. Results values themselves were not affected.
	4499	An incident was resolved for the Application Programming Interface (API) where the functions GetLabelNameList and GetLabelFromName in cPointObj, cFrameObj, and cAreaObj, as well as the function GetAllFrames in cFrameObj, could return an incorrect story name if the multiple-tower feature was enabled in a model (Options menu > Allow Multiple Towers) and the point/frame/area object was not in the active tower as set by cTower.SetActiveTower. In some cases, the affected API functions would return a value of 1 (error). This issue did not affect models that did not have the multiple-tower feature enabled.
	4865	An incident was resolved for the Application Programming Interface (API) where the function cFunction.ConvertToUser would return a runtime error when used. When the issue occurred, the function values that were converted could be corrupted and cause an error when the function viewed or used as part of a load case. This issue only affected the API function and did not affect models unless the API function cFunction.ConvertToUser was used.
	4916	An incident was resolved for the Application Programming Interface (API) where the following functions were implemented for ETABSv1.DLL , but incorrectly returned a "Not Implemented" error code when using the cross-product API (CSiAPIv1.DLL) instead: cCaseStaticNonlinearStaged.GetStageData_2, cCaseStaticNonlinearStaged.GetStageDefinitions_2, cCaseStaticNonlinearStaged.SetStageData_2, cCaseStaticNonlinearStaged.SetStageDefinitions_2, cPropFrameSDShape.GetAngle, cPropFrameSDShape.GetISection, cPropFrameSDShape.GetReinfCircle, cPropFrameSDShape.GetReinfCorner, cPropFrameSDShape.GetReinfEdge, cPropFrameSDShape.GetReinfLine, cPropFrameSDShape.GetReinfRectangular, cPropFrameSDShape.GetReinfSingle, cPropFrameSDShape.GetSolidCircle, cPropFrameSDShape.GetSolidRect, cPropFrameSDShape.GetTee, cPropFrameSDShape.GetConcreteL, cPropFrameSDShape.GetConcreteTee.
	4920	An incident was resolved for the Application Programming Interface (API) where the function Results.FrameForce returned an error when used over the entire structure by setting first parameter, "Name", to "ALL" and second parameter, "ItemTypeElm", to GroupElm (2). This issue only occurred for models that contained frame elements that were internally created from objects that were not frames, such as when edge constraints were used.
	5025	An incident was resolved for the Application Programming Interface (API) where calling the functions cPointObj.SetGroupAssign, cFrameObj.SetGroupAssign, or cLinkObj.SetGroupAssign will cause analysis results to become unavailable if the model had already been run. This behavior has been changed so that the API function SetGroupAssign for all object types will not delete the analysis results.

Data Files

Incidents Resolved

*	Ticket	Description
	3980	An incident was resolved where pier and spandrel force acceptance criteria data in a model that was exported as a .e2k text file were not successfully recovered when the model re-imported from the .e2k file. Results agreed with the model as imported.
	4994	An incident was resolved that addressed several issues related to the export and import of the text data file (.E2K, .$ET): (1.) When importing confinement data for wall hinge reinforcement assignment to shells, the "Confined" field under shear/confinement details was always being imported as "No" for all three stations even if the field was set to "Yes" for one or more stations. This has been corrected. (2.) The seismic-design grade specification has been added to text import/export for the Chinese 2010 and 2018 codes. (3.) Several missing items under the general-structure preferences for the Chinese codes have been added to text import/export. (4.) A bug was fixed where importing "Chinese 2018" code under Steel Design Preferences in the .E2K/$ET file set the code to Chinese 2010 in the Steel Frame Design Preferences form. None of these issues affected models saved and opened using the model file (.EDB).
	5362	An incident was resolved where changes made to ETABS model file (.edb) were not saved and no message was given when that file was set as "Read-only" or located in a folder that was "Read-only". Now the user will be prompted to save the model with a different name or in a different location.

Database Tables

Incidents Resolved

*	Ticket	Description
	3573	An incident was resolved where database import of additional mass in the X and Y direction only applied the mass in the Y direction. Results agreed with the model as imported.
	3585	An incident was resolved where values of a Uniform Load Set that were changed via the Interactive Database Editor were being added to the original values instead of replacing them.
	3916	An incident was resolved where database import that added to an existing model sometimes overwrote the name of the model with the name of the imported file.
	3931	An incident was resolved where the units conversion may not be correct in the quick filter option for some columns in the database tables. This could affect the behavior of the filter, but did not affect the numerical values shown in the tables. The actual filter applied was shown above the table after the filter was applied.
	3970	An incident was resolved where hinge assignments could sometimes be unexpectedly removed or duplicated when modifying hinge definitions using the Interactive Database Editor.
	4334	An incident was resolved where, when using Interactive Database editing (Edit menu > Interactive Database) to alter or add a direct-integration time-history load case, the mass- and stiffness-proportional damping coefficients specified by frequency would be changed to zero (0). This issue affected both linear and nonlinear direct-integration time-history load cases, but only affected the mass- and stiffness-proportional damping specified by frequency. When this issue occurred, the mass- and stiffness-proportional damping used for analysis reflected the damping values as shown in the Load Case Definition forms after interactive database editing was completed.
	4530	An incident was resolved where, when the Tendon Object Connectivity table was edited using Interactive Database editing on a selection of tendon objects, tendons that were not selected were incorrectly removed from the model.
	4653	An incident was resolved where the interactive database editor (Edit menu>Interactive Database) did not permit the following parameters in the Hinge Deformation tables to be

*	Ticket	Description
		set to values less than one (1), even though any positive, non-zero value should have been allowed: Parameters "Degrading Tension x1", "Degrading Tension x2", "Degrading Compression x1", and "Degrading Compression x2" for Degrading hysteresis; and parameters "BRB Tension Max Plastic Deformation Ratio", "BRB Tension Accumulated Plastic Deformation Ratio", "BRB Compression Max Plastic Deformation Ratio", and "BRB Compression Accumulated Plastic Deformation Ratio" for BRB Hardening hysteresis. This issue only affected interactive database editing. When this issue occurred, the parameters displayed in the Hinge Property Data form for the affected hysteresis types were consistent with the values used for analysis. The behavior of the database tables has been modified to be consistent with the behavior of the Hinge Property Data form.
	4864	An incident was resolved where the import of the table "Frame Section Property Definitions - Auto Select List" table did not work correctly, sometimes clearing the list of sections from the auto-select property definitions.
	5077	An incident was resolved where, when the table "Area Assignments - Diaphragms" was edited through the Interactive Database or by using the cDatabaseTables API functions, the pier labels were removed for all area objects present in the table when imported back into the model.
	5160	An incident was resolved where hinge result output tables did not show results for load combinations.
	5189	An incident was resolved where, when concrete Parametric P-M2-M3 hinges were added or modified through the interactive database editor (Edit menu > Interactive Database), the concrete Parametric P-M2-M3 hinges could not be used for analysis. When these hinges were assigned to a frame object, an error message "Error in regenerating hinges. Some hinges were deleted." appeared when running analysis and the affected hinge assignments were removed. When this issue occurred, the analysis results did not include the effect of the removed hinges.
	5209	An incident was resolved where wall spandrel overwrites table display was not available. All wall design codes were affected. This was a display issue only and no results were affected.
*	5285	An issue was resolved where, when using Interactive Database Editing (Edit menu > Interactive Database) to add or modify linear or nonlinear direct-integration time-history load cases using the "Load Case Definitions - Time History - Linear Direct Integration" and "Load Case Definitions - Time History - Nonlin Direct Integration" tables, any specified modal damping would not be used during analysis until the modal damping parameters are edited in the Load Case Data form (Define menu>Load Cases). This issue did not occur for linear or nonlinear direct-integration time-history load cases that were added through the Load Cases form (Define menu > Load Cases). When this issue occurred, the analysis log file shows that no modal damping was applied. This issue was resolved and a new field "ModalCase" has been added to the "Load Case Definitions - Time History - Linear Direct Integration" and "Load Case Definitions - Time History - Nonlin Direct Integration" tables so that the modal case used in these load cases can be viewed and edited in the database tables.
	5320	An incident was resolved where the database table "Analysis Messages" might not export to Microsoft Access and instead produced an error message when the text output to the Message field was too long.
	5355	An incident was resolved where, when frame hinge assignments in the table "Frame Assignments - Hinge - User" are modified through the interactive database (Edit menu > Interactive Database), user-defined hinges assigned to frame objects could be duplicated instead of modified. When this occurred, the additional hinges caused by the duplication was reflected in graphical user interface forms and the table "Frame Assignments - Hinge - User" when the table was re-opened. Results agreed with the model having duplicated hinges, generally exhibiting excess ductility.

Design – Composite Beam

Incidents Resolved

*	Ticket	Description
	4008	An incident has been resolved affecting composite beam design per the CSA S16-14 code of very long cantilevers in models defined in Newton and millimeter units. For such models, a warning was sometimes posted that the slab thickness was inadequate to withstand the longitudinal shear force. That warning was irrelevant since cantilevers are designed without relying on any composite action, and it could be safely ignored. The design of the cantilevers was unaffected. This warning is no longer posted for cantilevers.
	4516	An incident was resolved which affected the design of composite beams per the BS5950 code. When a beam was subject to negative bending, the negative bending design capacity was reported as 0.0 in the strength checks. The factored negative bending moment and the demand capacity ratio were correctly reported. This was a reporting issue only and the design was otherwise correct. The negative bending design capacity is now correctly reported for composite beams subject to negative bending designed per BS5950.
	5096	An incident was resolved which affected composite beam design in models where one or more load combinations had been assigned to both the construction strength and service strength checks for composite beams. When one of these load combinations controlled the construction strength check for a particular composite beam, the design was properly performed, but the numerical values reported for the construction strength check were incorrect in the Interactive Composite Beam Design and Review form and in the output. Note that whether the beam passed or failed the construction strength check was always properly reported, but the supporting numerical values were being overwritten by values from the service strength check. This issue affected ETABS versions 17.0.1 to 18.1.1.
	5258	Several composite beam design documentation changes were made. (1.) The vibration floor length and floor width were changed from their default values for the beams framing into the girder being designed to ensure ETABS would select a W21x50 section when designing those in file AISC-360-10 Example 002.edb. This change was required by changes made to the evaluation of beam vibrations documented in ETABS v17.0.0 Release Notes Incident 88436. (2.) The composite-beam design-preferences total deflection criterion was relaxed in the CSA-S16-14 Example 001.edb verification model. This change was made because, pursuant to the change documented in ETABS 18.1.0 Release Notes Incident 3189, the concrete shrinkage deflection computed by ETABS for the beam in this model had increased, and along with it, the percentage of composite action selected by ETABS in order to keep the total deflection acceptable, resulting in a mismatch between the model results and the values published in the verification manual. The vibration floor length and floor width were also changed from their default values. (3.) The joint and frame releases were changed in files AISC DG02 Example 002.edb and AISC DG02 Example 004.edb to stabilize the two models. Results were not affected as the unstable degrees of freedom were not loaded. (4.) The documented ETABS live load deflection was revised in the EC-4-2004 Example-001 manual. The change results from changes to how transformed composite beam moments of inertia are computed as documented in ETABS v18.0.0 Release Notes Incident 221321. (5.) The display units were adjusted in several models to enhance the legibility of the design results and match the numbers tabulated in the verification manuals.
	5262	An incident was resolved which affected the design of composite beams that were subject to negative bending for some, but not all, design load combinations. For such beams, and for a given shear stud distribution, the number of shear studs located between the location of the maximum positive bending moment and the nearest point of zero moment can vary across design load combinations, and so does the percentage of composite action provided by the shear studs. When designing such composite beams, ETABS would use the same smallest percentage of composite action provided by the shear studs for all the design load combinations, and would accordingly compute a larger number of shear studs than required; in the case of a beam with an auto-select section, this could results in a larger section than required. Note that when the beam being designed did not have an auto-select

*	Ticket	Description
		section and the number of shear studs computed by ETABS was enough to provide full composite action for the design load combination with the largest positive bending moment, the interactive design form and composite beam design output would correctly show a bending strength check based on the bending capacity of the fully composite beam, indicating that the flexural capacity of the beam was adequate, but would also show a message contradicting that. When that happened, the message could be ignored. This incident affected composite beams designed in all versions of ETABS that design composite beams, and it affected design per all design codes except the Eurocode. Now, the percentage of composite action provided by the shear studs varies across design load combinations and the beam-bending capacity is computed separately for each load combination being checked. The effect of this error was small and conservative.
	5339	An incident was resolved for AISC 360-10 filled composite column design (i.e. filled box/tube and pipes sections) where composite column axial compression capacity was incorrect due to an error in the computation of the EI value as per AISC equation I2-12. The value of Ic was using moment of inertia from steel section instead of from the concrete portion. This error was producing a conservative design. Composite column verification examples 1 and 3 have been updated to reflect this modification.

Design – Concrete Frame

Incidents Resolved

*	Ticket	Description
	4078	An incident has been resolved in concrete frame design codes ACI 318-14, ACI 318-11, and ACI 318-08 in which the program does not produce an error message when Pu is greater than 0.75*Pc for certain conditions. A similar problem has been fixed for the following codes too:, "Mexican RCDF 2017", "Mexican RCDF 2004", "CSA A23.3-14", "NZS 3101:2006", "SP 63.13330.2012", "TS 500-2000", and "TS 500-2000(R2018)". This error was a reporting only problem. All calculations were correct.
	4337	An incident was resolved for AS 3600-2018 concrete frame design where torsional longitudinal rebar was reported by combining the top and bottom longitudinal torsional rebar instead of reporting the additional torsional longitudinal rebar at the top and bottom separately. This was producing overly conservative results. Now, additional torsional longitudinal rebar is reported in both the design output and the tabular output for top and bottom locations separately. The reported torsional longitudinal rebar is in addition to the flexural reinforcement.
	4722	An incident was resolved where the automated triangular load applied to calculate stiffness of a building for measuring Stiffness/gravity loads in the Chinese codes did not agree with a strict interpretation of the code. Now the load is applied to each story in proportion to its height from the ground. Originally, the load was also dependent on the distribution of the mass. It's dependence on the mass has been removed.
	4724	An incident was resolved for concrete frame design per the Eurocode that addresses the following issues: (1.) A maximum concrete strain of 0.0035 was used for Eurocode for all concrete strengths when calculating PMM interaction surfaces. This limit was incorrect for concrete strengths higher than C50 where a smaller value should be used as per Eurocode (Table 3.1 of BS EN 1992-1-1:2004). (2.) The compression strain limit is now limited by the epsilon_c3 when the whole section is under compression strain, and the strain distribution is governed by the code (Figure 6.1 of BS EN 1992-1-1:2004). Specifically, the depth was previously limited by (1 - epsilon_c2/epsilon_cu2)*h, but now it is limited by (1 - epsilon_c3/epsilon_cu3)*h. This change affect will affect the PMM interaction near the top where the compression governs and where the whole section is under compression strain irrespective of the value of fck.
	4999	An incident was resolved for the concrete frame design code "Chinese 2010" in which the design output displayed using the command Design > Concrete Frame Design > Display Design Info was not of the type requested. This particularly affected the items Axial

*	Ticket	Description
		Compression Ratio N/(fc*A), General Reinforcement Details, Identify P-M failure, Identify Shear Failure, and Identify All Failures. The actual type of the results displayed was indicated in the title bar of the window and the results were correct for that type.
*	5197	An incident was resolved for concrete frame design code ACI 318-14 in which the minimum longitudinal rebar requirement for Ductile moment frames was based on clauses 9.6.1.2 and 9.6.1.3. However, the exception clause 9.6.1.3 (which allows increasing the calculated rebar areas by a factor of 4/3 and not using the minimum specified in clause 9.6.1.2) should not be used for Ductile moment frames under clause 18.6.3.1. The new implementation no longer uses the exception clause 9.6.1.3 for Ductile moment frames.

Design – Shear Wall

Incidents Resolved

*	Ticket	Description
	4848	An incident was resolved for shear-wall design using the Hong Kong CP 2013 code where running the design would often display the error messages "Error during design of Pier ... at Story ...". When this occurred, design results were not available.
*	4993	An incident was resolved for the concrete shear-wall design code "Chinese 2010" where previously the stability requirement was deemed to be satisfied if either the thickness was larger than the required thickness for stability OR if q was less than q_limit. This has been corrected so that now the stability requirement is deemed to be satisfied if both the thickness is larger than the required thickness for stability AND q is less than q_limit. Additional information is now displayed in the design details to confirm these values and the result of the stability check.
	5135	An incident was resolved for concrete shear wall design per the "ACI 318-08", "ACI 318-11", and "ACI 318-14" codes in which the design did not report values for Pu, Mu, the stress at the edge, and the stress limit for boundary elements on the side of the wall where the moment caused tension, even though the overall stress could be compressive because of the presence of compressive force Pu. Now the values of Pu, Mu, the stress at the edge, and the stress limits are always reported for both edges.
	5198	An incident was resolved that addressed two issues for shear wall design per the "ACI 319-14" code: (1.) For the case where Ln/h < 4 and the diagonal rebar was to be calculated, the design would previously report an error message when Vu > 8*phi*sqrt(fc')*Acw. This limit was too small, and has been corrected to report an error message when Vu > 10*phi*sqrt(fc')*Acw. (2.) The design correctly used the strength reduction factor phi as 0.85 for designing spandrel diagonal rebar per the code section 21.2.4.3. However, the documentation stated that the factor used was 0.75. The documentation has been corrected. This phi value is also now reported in the design details. Results were not affected.

Design – Steel Frame

Incidents Resolved

*	Ticket	Description
	2492	An incident was resolved in the New Zealand steel frame design code "NZS 3404:1997" to properly determine the effective length factor used in the calculation of the nominal axial capacity of steel members for models in which the General Second-Order Analysis is selected as the Analysis Method and the Frame Type is Moment Frame (Sway Frame). The determination of the k factors used for axial compression capacity is based on the analysis method, framing type, and the presence of seismic load. The following conditions are used:(a) k for both major and minor axes are taken as 1 (unity) if the analysis method is "General 2nd Order", (b) k for both major and minor axes are taken as 1 (unity) for "Braced Frame" if the analysis method is "Amplified 1st Order" and seismic load is present, (c) k for both major and minor axes are taken as k_eb (braced) for "Braced Frame" if the analysis method is "Amplified 1st Order" and seismic load is not present, (d) k for both major and

*	Ticket	Description
		minor axes are taken as k_es (sway) for "Sway Frame" if the analysis method is "Amplified 1st Order" irrespective of whether seismic load is present. Previously, design results were overly conservative for some conditions.
	3432	An incident was resolved for steel frame design code per the “Chinese 2018” code in which the Beta_x and Beta_y factors for circular pipe sections were calculated based on the corresponding Nex and Ney values, which in turn were based on the corresponding lambda_x and lambda_y values. Now both Beta_x and Beta_y factors for circular pipe sections are calculated based on the smaller value of Nex and Ney, which in turn is based on the larger lambda value and the appropriate values of M2/M1 per GB50017-2017 section 8.2.4. This affects calculation of Beta (Beta = Beta_x * Beta_y) per equation GB50017-2017 8.2.4-3 and calculation of the PMM interaction ratio per equation GB50017-2017 8.2.4-1.
	4034	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 in which the buckling factor phi_e for lambda_bar between 0.0 and 0.5 (0.0 < lambda_bar < 0.5) was previously determined by linear interpolation between the values of phi_e for lambda_bar = 0.0 and lambda_bar = 0.5. However, since the phi_e for lambda_bar = 0.0 is not given in the code (SP 16.13330.2017 Annex E, Table E.3), phi_e for lambda_bar = 0.0 was conservatively assumed to be 1.0 for all values of mef. Now, the phi_e for lambda_bar between 0.0 and 0.5 is calculated instead by linear extrapolation backward from the values of phi_e for lambda_bar = 0.5 and phi_e = 1.0. The previous method was conservative for large values of mef.
	4035	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 for singly-symmetric sections, especially for a singly-symmetric I-shaped section and when plastic strains are allowed, where now the different values of the section modulus W_y for the tips of the top flange and the bottom flange are used in the calculation of interaction ratios per Eqn. 105 of SP 16.13330.2017 section 9.1.1. Previously, the design was using the minimum of W_y,top and W_y,bottom for both top and bottom corner points. Now the design uses Eqn. 105 for all extreme corner points with the appropriate section modulus for each point. All singly symmetric sections (singly-symmetric I-shape, Channel, T-shape, and Double Angle) are affected.
	4039	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where, for a Channel section when the moment about the axis of symmetry is zero (major axis bending, M33=0 in ETABS terms), the power term n used in the interaction equation SP 16.13330.2017 9.1.1 is now taken as equal to 3 when M22 is zero or positive (the flange tips are in tension), and equal to 1 when M22 is negative (the flange tips are in compression) based on SP 16.13330.2017 Annex F, Table F.1 case 9. Previously, for a Channel section when the moment about the axis of symmetry was zero, n was taken as 1 whether M22 was positive, negative, or zero. However, when the major axis bending moment is nonzero or nontrivial, the power term n is taken as 1.5, which has not changed. Previously the design was slightly conservative.
	4041	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where now the calculation A_f/A_w for box tube shapes uses the ratio of the flange area bounded by the web centerlines to the web area bounded by the flange centerlines. Previously, A_f/A_w was calculated using the gross flange area and the clear web area. This affects the calculation of c_x and c_y per the code (SP 16.13330.2017 Annex F, Table F.1).
	4043	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where the calculation Af/Aw for minor axis bending of singly-symmetric I-shaped sections now takes the area of the web Aw as the sum of the two unequal flanges (SP 16.13330.2017 Annex E, Table E.2). Previously, Aw was taken as two times the top flange area. The value of Af is taken as the clear area of the web, which is unchanged. The calculation of Af/Aw for major axis bending remains unchanged. The Af/Aw ratio affects the calculation of eta, which then affects mef and phy_ey per the code (SP 16.13330.2017 Annex E, Table E.3, 9.2.2).
	4050	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where now, when the buckling check for box sections subjected to axial compression with uniaxial or biaxial flexure is performed, the design checks both Eqn. 120 and Eqn. 121 (SP

*	Ticket	Description
		16.13330.2017 section 9.2.10). Previously, this check was performed using only one of these equations, depending on the plane in which the moment was applied. The previous approach was slightly unconservative.
*	4053	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where now, when the buckling check for box sections subjected to axial compression with uniaxial or biaxial flexure is performed, the design checks both Eqn. 120 and Eqn. 121 (SP 16.13330.2017 section 9.2.10) with the replacement of phi_x for phi_ex or phi_y for phi_ey as appropriate (I_x > I_y and M_y = 0 for Eqn. 120; I_y > I_x and M_x = 0 for Eqn. 121). Previously, this check was performed using only one of these equations, depending on the plane in which the moment was applied. The previous approach was slightly unconservative.
	4056	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where the Phi buckling factor could be calculated incorrectly for the buckling curve "c" when lambda_bar was near zero due to the numerical sensitivity of the code equation at that limit. Phi should be equal to 1.0 for lambda_bar = 0.0 per the specification SP 16.13330.2017 Annex E, Table E.1. Now the value of Phi will be set to 1.0 as lambda_bar approaches zero.
	4057	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where, for a singly-symmetric I-shaped section when the moment about the major axis is zero (major axis bending, M33=0 in ETABS terms), the power term n used in the interaction equation SP 16.13330.2017 9.1.1 is now taken as equal to 3 based on SP 16.13330.2017 Annex F, Table F.1 case 6, similarly to doubly-symmetric I-shapes. Previously for this case, n was taken as 1.5. Now, for a singly-symmetric I-shaped section: (a) n=2 when M33 is significant and M22=0 (case 2) or when neither of them is significant, (b) n=3 when M22 is significant and M33=0 (case 6, similar to doubly symmetric I-shapes), and (c) n=1.5 when both moments are significant (table note*). The only change was for condition (b). Conditions (a) and (c) are unchanged. Previously condition (b) was treated as a condition (c).
	4065	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 where the shape influence coefficient (Eta) for T and double-angle sections for bending in the plane of the minor axis is now calculated assuming section type 8 per the specification SP 16.13330.2017 Annex E, Table E.2. Previously, section type 5 was assumed for this case. The Eta factor is used in clause SP 16.13330.2017 9.2.2.
	4067	An incident has been resolved in the Russian steel frame design code SP 16.13330.2017 for buckling calculations of channel sections under the combined action of axial compressive force and moment in the plane coinciding with the symmetry plane. Now the design no longer checks interaction Eqn. 70 of SP 16.13330.2017 section 8.1.4, which corresponds to the case of bending without axial force. Only Eqns. 109 and 111 of SP 16.13330.2017 section 9.2 need to be checked in the presence of compressive force, and this has not been changed. Previous results were overly conservative when also considering Eqn. 70.
	4126	An incident was resolved for the steel-frame design codes AISC 360-16 and AISC 360-10 where the design report was incorrectly showing the message "Stress Check Message - lo/i exceeds limit (GB50017-2017 7.4.6, 7.4.7)" instead of "Stress Check Message - Lb/ry > 0.19*E/RyFy (ANSI/AISC 341-16 E2.4a, D1.2a.1(c))" for the AISC 360-16 code, or instead of "Stress Check Message - Lb/ry > 0.17*E/Fy (ANSI/AISC 341-10 E2.4a, D1.2a)" for the AISC 360-10 code. No design results were affected.
	4954	An incident was resolved for Eurocode 3-2005, Indian IS 800:2007, Italian NTC 2018 and Italian NTC 2008 design codes where description of the formula used to represent D/C ratio for steel pipes was not including the SRSS combination. This was just a display issue and design results were not affected.
	5220	An incident was resolved for steel-frame design per the AISC 360-16 and AISC 360-10 codes where the capacity moment used to design a beam between the column and an RBS (reduced beam section) hinge was using a Cpr value equal to 1.1. Now this value is computed from the steel material using Fy and Fu values. AISC 360-10 joint design has also updated to use a Cpr value from the material definition instead of using Cpr equal to 1.1.

*	Ticket	Description
*	5225	An incident was resolved where steel frame design failed to complete when an automatic design iteration was performed. This only affected models where auto-select section property lists were assigned to steel frame members. Setting the maximum number of automatic iterations to one and manually iterating the design worked correctly.
	5226	An incident was resolved in steel frame design code "Eurocode 3-2005" where the program-calculated stress ratio for the torsion check was reported as infinity for the first station and first load combination.

Documentation

Incidents Resolved

*	Ticket	Description
	3513	An incident has been resolved for the Russian steel frame design manual “ETABS v18 Steel Frame Design Manual SP16.13330.2011” in which the Table of Contents indicated extra chapters that did not exist. There is no change in the software.
	3750	An incident was resolved for the Application Programming Interface (API) where the documented example code for functions cDesignStrip.GetDesignStrip and cDesignStrip.GetDesignStrip_1 was incorrect. This was an error in the documentation and did not affect the behavior of the software or the API.
	4104	A figure in the help Topic "Pier Force Acceptance Criteria" was corrected to reflect the actual behavior of the software. This was a documentation error only, and no results were affected or changed.

Drafting and Editing

Incidents Resolved

*	Ticket	Description
	1351	An incident was resolved where importing a .DXF file as an architectural plan would sometimes fail to add certain selected columns and walls to the ETABS model due to incorrect tolerance checks.

External Import and Export

Incidents Resolved

*	Ticket	Description
	3368	An incident was resolved where circular and semicircular floor slabs and decks could not be imported from Revit. This incident affected version 18.1.0 and previous versions of ETABS capable of defining area objects with curved edges. When the problem occurred, it was visually obvious and the results agreed with the model. The problem was resolved in v18.1.1 but inadvertently omitted from the Release Notes.
	5126	An incident was resolved which affected the import of DXF files containing arcs that are neither horizontal nor vertical. These arcs were being imported in the wrong location. When this occurred, the error was visually obvious and the results agreed with the model as imported. As a workaround, such arcs could be imported as straight lines by first exploding them in AutoCAD. This affected all versions of ETABS capable of importing arc objects, i.e., versions 13.1.2 to 18.1.1.

Graphics

Incidents Resolved

*	Ticket	Description
	1657	An incident was resolved where the deformed-shape plots of walls curved in plan were not correct. No results were affected.
*	3593	An incident was resolved where an abnormal termination could occur when trying to display uniform shell loads in the case where these loads were not directly assigned to the selected load pattern. No results were affected.

*	Ticket	Description
	3601	An incident was resolved where Load contours were not displayed on shells assigned Null properties. This was a display issue only and results were not affected.

Loading

Incidents Resolved

*	Ticket	Description
*	3586	An incident was resolved where the frame distributed span load form allowed an invalid specification of load with the start and end distances of the load being inverted. Loads defined this way would show in the display, but were not being used in analysis. Now such loads specifications are allowed, and the loads wills be applied in analysis as they are displayed.
*	3772	An incident was resolved for NBCC 2015 auto seismic load pattern where no results were available when Site Class was F.
*	4107	An incident was resolved where non-uniform loads specified on walls would not be correctly applied in analysis. This error was inadvertently introduced in v18.1.1. Uniform loads on walls were not affected and uniform or non-uniform loads on floors were not affected by this error. Any model run in v18,.1.1 with non-uniform loads on walls should be rerun.
*	4983	An incident was resolved where a nonlinear static, staged-construction, or direct-integration time-history load case that had loads with a scale factor of exactly zero (0) could apply additional frame loads that were not specified in the load case. This issue only occurred for load cases that continued from the end of another nonlinear static, staged-construction, or direct-integration time-history load case and that applied frame loads in a load pattern, scaled by zero, that was also applied in the previous nonlinear load case (or one if its predecessors). When this issue occurred, the load pattern scaled that was scaled by zero was being applied with a scale factor from some other load pattern in the same load case. The extra loads were reflected in all analysis results, including base reactions and frame reactions. This issue affected ETABS v18.0.0 to v18.1.1.
*	5176	An incident was resolved for Indian IS 1893:2016 auto-seismic load patterns where the minimum base-shear coefficient based on Table 7 of the IS 1893:2016 code was not being enforced.

Results Display and Output

Incidents Resolved

*	Ticket	Description
	1621	An incident was resolved where Named Displays were not working for DirectX graphics mode. Now a Named Display defined in either DirectX or Standard graphics mode can be recovered on screen in either graphics mode or included in a report.
	2620	An incident was resolved where an error condition was generated in certain cases when plotting a pushover curve with a target displacement to be shown. This could occur when the demand spectra was very large compared to the capacity.
	3773	An incident was resolved where the output step numbers shown in the Link Object Hysteresis form were shifted by one step when the Recovery Extent was set to "All", resulting in an extra point at the beginning of the curve and causing the last point to be missed.
	3946	An incident was resolved where, in DirectX graphics mode and with certain combinations of screen resolution settings, the File > Capture Picture commands would not correctly capture the requested portion of the window.
	4151	An incident was resolved where the hinge state for concrete parametric P-M2-M3 hinges was reported incorrectly as "C to <=D", "D to <=E", and ">E" when the hinge passed the U, C, and D points respectively. This was a reporting issue only and did not affect the analysis behavior of the hinge. This issue did not affect the steel parametric P-M2-M3.
*	4344	In incident was resolved where the Display Performance Check feature (Display menu >

*	Ticket	Description
		Show Performance Check) always displayed the D/C ratios for the entire performance check regardless of the Demand Set selection. This issue has been resolved so the D/C ratios for the selected Demand Set is shown.
	4850	An incident was resolved where the "ID" column of the table "Grid Definitions - Grid Lines" was not included in reports.
	4929	An incident was resolved where the signs of forces shown for a Section cut drawn on screen was flipped when in DirectX display mode. This did not affect standard-graphics mode.
	5051	An incident was resolved where trying to view Beam Diagrams from the Concrete Frame Design right click form would cause an error condition. No results were affected, and this was a display issue only.

Section Designer

Incidents Resolved

*	Ticket	Description
	3799	An incident was resolved where the calculated torsional constant for cold-formed Z sections was not correct. No other section properties were affected.

Structural Model

Incidents Resolved

*	Ticket	Description
	1586	An incident was resolved where, in some cases, the Mander confined stress-strain curve was not calculated resulting in an error condition when the model was analyzed. This happened when the concrete strength was high and the confinement was low. This condition is now properly handled.
*	2016	An incident was resolved where, in a staged construction load case, a wall object with an assigned wall hinge would not be added correctly using the "Add Structure" stage operation using object type Group, Story, or Shell. When the wall object with an assigned hinge was added as part of a staged construction load case, the wall object exhibited significantly reduced stiffness in the vertical axial and in-plane bending directions represented by the hinge. This issue also affected auto-construction sequence cases.
	2863	An incident was resolved where, under certain conditions, a frame nonlinear hinge could not be generated at analysis time and an error message was presented. This only occurred if all three of the following conditions were present: Hinges were set to be modeled in links; the frame had rigid end offsets; and a hinge was assigned at the face of the rigid-end offset. When this occurred, results were not available.
	3736	An incident was resolved where the shear reinforcement ratio used for generating ASCE 41-13 and ASCE 41-17 concrete column auto hinges was sometimes incorrectly taken as smaller than the values specified in design or in the frame section property data. This issue occurred only when the shear reinforcing ratio option in the Auto Hinge Assignment Data form was set to "From Current Design". When this issue occurred, the generated hinge properties accessible through the Define Frame/Wall Hinge Properties form were the actual properties used for analysis. Results agreed with the generated hinges, including the effect of the incorrect shear reinforcement ratio.
*	4134	An incident was resolved where, when a shear hinge was defined with a program-generated yield shear force, the yield force did not include the effects of shear reinforcement. The behavior is corrected to include the shear reinforcement area as calculated by Concrete Frame design in the program-generated yield shear force.The hinge yield shear force, Vy, is calculated as Vy = 2 * (Shear area) * sqrt(f'c) + (Asv) * Fye * (Height of the section). Here, f'c is the concrete strength, Asv is the area of shear rebar, and Fye is the expected yield strength of the rebar material.
	4168	An incident was resolved where the data entered or edited on the Wall Hinge Reinforcement form was not saved when the form was accessed through the Wall Information form that is available by right-clicking on the wall. This error did not affect the

*	Ticket	Description
		use of the Wall Hinge Reinforcement form when accessed from the Assign menu.
*	4608	An incident was resolved where mass coming from non-uniform area loads could be missing under certain conditions. If a user had specified that the mass be calculated from load patterns and one of those load patterns included non-uniform area loads, then that model should be rerun with the new version to confirm that the mass was included. Mass from uniform area loads was correct.
*	4819	An incident was resolved where, if a user-defined fiber P-M3 hinge was defined or modified through the interactive database and assigned to a wall object, the hinge length (relative to wall height) may have been set to a value that was not 1.0. When this issue occurred, the wall would use purely elastic properties over any remaining length not included in the fiber-hinge length. The behavior of the P-M3 hinge would also reflect the reduced hinge length, tending to reduce ductile deformation.
	4890	An incident was resolved where a Parametric Steel P-M2-M3 hinge always used program-computed yield deformations values for deformation scale factors even if the "Use Yield Deformations" option was not selected in the Hinge Properties form. When this issue occurred, the deformation scale factors in the generated hinge properties reflected the program-computed yield deformations values that were used for analysis.
*	4901	An incident was resolved where, for a Parametric P-M2-M3 hinge with a Moment-Curvature hinge specification type, the tension and compression deformation scale factors were shown with displacement units in the Hinge Property Data form, but are actually considered as strains for analysis. The Hinge Property Data form has been updated to be consistent with analysis.
	5163	An incident was resolved where adding line or quad strain gauges to a group and then later deleting some strain gauge objects could cause an error message when opening the model after the analysis had been run or when trying to display table results. When this occurred, results were not available.
*	5347	An incident was resolved in the New Zealand material property library in which the modulus and strength values of the aluminum material were previously defined in the wrong units. Existing models using the aluminum material from this library should be reviewed. Results agreed with the property values as imported into the model from the library.

User Interface

Incidents Resolved

*	Ticket	Description
	3338	An incident was resolved where adding design strips automatically using rectangular grids systems would result in an error condition if at least two grids were not defined in each direction. This condition is now trapped.
	3534	An incident was resolved where right-clicking on a model window that was displaying finite-element slab-design results would not bring up the usual context menu.
	3536	An incident was resolved affecting the display of time histories read from files that were defined with Time and Function values. (1.) The form would allow the text box for equal time interval to be enabled, allowing a value to be entered. (2.) Default value of time step (i.e. 0.02) was being used when generating and displaying response spectrum for times histories read from file. These errors only affected the form and did not affect any analysis results. These errors were not triggered if the time histories were not read from file or if the time histories were actually defined with Values at Equal Intervals.
	3797	An incident was resolved where, when using Chinese language, the auto hinge property could not be added when assigning wall hinges to shells.
	3951	An incident was resolved where in some cases text was overlapping on the Modify Undeformed Geometry form.
	4965	An incident was resolved for the Chinese 2010 auto wind load pattern where the T1 Source did not appear to be remembered correctly when it was set to "User Defined" on the load

*	Ticket	Description
		pattern definition form, and then the form was opened again. However, this was a form error only, and the User Defined value was being used for analysis and was shown correctly in the database tables.
	5268	An incident was resolved where a group assignment made through the model explorer was applied to both the desired group and also to the first group in the list.