

# ETABS® Version 18.1.0 Release Notes

© Copyright Computers and Structures, Inc., 2019

**Notice Date: 2019-12-10**

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

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

## **Changes from v18.0.2 (Released 2019-09-03)**

### **Installation and Licensing Enhancements Implemented**

*	Ticket #	Description
	2776	The version number has been changed to v18.1.0 for a new intermediate release.

### **Structural Model Enhancements Implemented**

*	Ticket #	Description
	2211	An incident was resolved where the CISC10 frame-section database contained certain sections that are not listed in the CISC Handbook Tenth Edition. These sections were identified and removed from the CISC10 section database: W200x21, W250x24, W310x31, W460x61, W460x67, W530x72, W530x82, WT265x41. Additionally, auto-select lists similar to those already available in the CISC9 frame-section database have been added to the CISC10 section database.
*	2918	An enhancement has been made to allow the assignment of acceptance criteria to piers and spandrels that measure forces, stresses, or stresses as a ratio of the square-root of the concrete compressive strength $f'_c$ . The acceptance criteria can be assigned individually to each pier and spandrel label on each story by selecting any object that is assigned to the pier/spandrel label and using the commands Assign > Assign Pier Force Acceptance Criteria and Assign > Assign Spandrel Force Acceptance Criteria. Pier-force acceptance criteria can be defined for V2 or V3 shear and may depend on the axial force/stress of the pier. Spandrel force acceptance criteria can be defined for V2 shear. The pier/spandrel force status ("A to IO", "IO to LS", "LS to CP", and "> CP") are output in the Database tables under Analysis Results > Element Output > Wall Output.
*	2919	An enhancement has been made to allow the assignment of acceptance criteria to Link and Panel Zone properties. For Link properties, the acceptance criteria can be defined as either displacement or force for any degrees of freedom that are active and not fixed. For Panel Zone properties, the acceptance criteria can be defined as moment or rotation. The Link and Panel Zone status ("A to IO", "IO to LS", "LS to CP", and ">CP") are output in the Database tables under Analysis Results > Element Output > Link Output for Links and Analysis Results > Element Output > Panel Zone Output for Panel Zones.
*	3364	As an enhancement, a built-in material library for the Korean region has been added. The library contains steel, concrete, and rebar materials per the KBC 2016 standard.

**Analysis  
Enhancements Implemented**

*	Ticket #	Description
*	2701	A new feature is available for running multiple load cases on separate parallel processors, significantly speeding up models with many independent load cases. This can be particularly useful where a large suite of earthquakes needs to be considered, such as for performance-based design. This option is turned off by default, since it can affect the performance on machines used for multiple purposes. When enabled, the number of processors to be utilized can be set explicitly or can be determined automatically as the number of physical processors found on the machine. Sequential load cases will be run on a single processor. Parallel load cases will be automatically assigned to processors as they become available, after any prerequisite load cases have been run. This feature requires sufficient memory (RAM) to support the number of parallel instances of the analysis engine. Larger models may run faster with fewer processors if memory is limited. All processors will access the disk, which can become the limiting factor for speed if the disk is slow or if the number of processors utilized becomes too large.
*	3098	An enhancement has been implemented to more clearly display and save messages generated during analysis runs. These are the same messages that appear in the analysis LOG file but provided in a more concise format. Analysis messages are categorized as information, warnings, and errors. After the analysis is run, messages are automatically displayed as text if there are any warnings or errors in the latest run. This same text display is available any time using the command Analysis > Messages. The messages are also available in tabular format under table Analysis Results > Run Information > Analysis Messages. Messages are cumulative with subsequent runs until the model is unlocked, at which time the messages are deleted. Each message includes its type, message text, associated load case or stiffness case, operation being performed, date-time stamp, parallel run tag, run serial number, and machine name. The run serial number counts subsequent runs before the model is unlocked. The parallel run tag indicates which thread was used when analyses are run in parallel during the same analysis run.

**Design – Concrete Frame  
Enhancements Implemented**

*	Ticket #	Description
	2998	An enhancement has been made to concrete frame design per the Russian code SP 63.13330.2012 to allow an additional input parameter for the reliability factor, Gamma_n, in the Design Preferences. The reliability factor is used to obtain the design seismic force depending on the functional use of the structure. It is considered the level of responsibility for the structure. For essential structures, it is more than 1.0, for temporary structures it may be less than 1.0. Its default value is taken as 1.0. However, the user can change it to any positive value. By increasing the reliability safety factor Gamma_n, the design force values are practically increased by multiplying by that factor.
	3059	An enhancement has been made for concrete column design to report in the detailed output the forces corresponding to the worst D/C ratio from the many force sets that are used to design for a given loading combination. Previously, this was not happening when the minimum rebar governed for all of the force sets. In such a case the first set was stored and displayed regardless of which force set had the worst D/C ratio. By way of background, multiple force sets are considered in a load combination for design whenever the combination is multi-stepped or is an envelope requiring permutations of axial load, minor and major moments. No design results are affected.

## Design – Steel Frame Enhancements Implemented

*	Ticket #	Description
*	2147	<p>An incident was resolved for steel frame design per the Russian code "SP16.13330" that corrected or improved several issues. Some of these changes were mandated updating the code from 2011 to 2017. Other changes were corrections or updated interpretations of code clauses that did not change between versions:</p> <p>(1.) The parameter <math>I_t</math> is now taken as the sum of <math>(K/3)*b*t^3</math>, where <math>K = 1.29</math> for doubly-symmetric I-sections, 1.25 for singly-symmetric I-sections, 1.12 for channel-sections, 1.2 for T-sections, and 1.2 for double-angle sections, all based on the 2017 specification SP16.13330.2017 Annex G. Previously the value of <math>I_t</math> was taken from values specified in the frame-section database. This parameter affects the calculation of <math>\phi_b</math> and <math>c_{max}</math>.</p> <p>(2.) The expression for <math>\phi</math> for doubly-symmetric sections (such as standard I sections and double-channel sections) is calculated differently from singly-symmetric sections (such as singly-symmetric I-sections, double-angle sections, and T-sections). Previously the value of <math>\psi</math> was always calculated following the case of doubly-symmetric section per SP 16.13330.2011 Table G.1 and Table G.2. For the singly-symmetric section, the parameter <math>\psi</math> (or <math>\psi_a</math>) is now calculated based on SP16.13330.2011 G.4 Eq. (G.9), section G.4, section G.5, Table G.4, and Table G.5. All conditions of G.6 and G.7 are used. If the parameter <math>n</math> is larger than 0.9 (<math>0.9 &lt; n &lt; 1.0</math>), interpolation is used for the value of <math>n</math> between those for pure T-sections and singly-symmetric I-sections to determine the value of <math>\psi</math>. The parameter <math>\psi</math> is used to calculate <math>\phi_b</math>. The detailed design report now provides more information on the calculation of these values.</p> <p>(3.) The following changes apply for the calculation of <math>C_{max}</math>:</p> <p>(3.a) The parameter <math>h</math> is taken as the distance between the two centers-of-gravity of the top and bottom flanges for I-sections, channel sections, and double-channel sections. It is taken from the bottom of T-sections and double-angle sections to the center-of-gravity of the top flange following the 2017 specification SP 16.13330.2017 Annex E. Previously it was taken as the clear distance to the flanges for I-sections, channel sections, and double-channel sections; and from the bottom of T-sections and double-angle sections to the bottom of the top flange.</p> <p>(3.b) The parameter <math>n</math> is taken as <math>n = I_1 / (I_1 + I_2)</math> where <math>I_1</math> and <math>I_2</math> are the moments of inertia of the major and minor flanges about the section minor axis per SP 16.13330.2017 Annex Eq G.8. Previously the expression <math>n = I_y / (I_1 + I_2)</math> was used following the code, which was incorrect. The parameter <math>n</math> for double-angle sections and T-sections is now taken as 1.</p> <p>(3.c) The parameters <math>\omega</math>, <math>\alpha</math>, and <math>\mu</math> are now calculated differently for the major and minor bending axes of channel sections based on the specification SP 16.13330.2017 Annex E and Annex G. Previously the expressions for <math>\omega</math>, <math>\alpha</math>, and <math>\mu</math> did not differ for the major and minor bending axes of channel sections.</p> <p>(3.d) The case of channel section with <math>I_{major} &gt; I_{minor}</math> has been added in the implementation per the Annex E.6 for <math>c_{max}</math> calculation.</p> <p>(4.) If the shear-stress ratios (<math>\tau/R_s</math>) for both the major and minor directions are small (less than 0.01, essentially pure shear), then <math>c_m</math> for the major and minor directions is calculated based on equation SP16.13330.2011 8.2.3, Eq. (53). Otherwise, <math>c_m</math> is taken as <math>c</math>. Previously these same equations were used, but the limiting condition used was different. It used to be applied for singly-symmetric beams.</p> <p>(5.) The parameter <math>c</math> is now only calculated for singly-symmetric I sections, doubly-symmetric I sections, double-channel sections, channel sections, double angle sections, and T-sections per SP16.13330.2011 9.2.5, Table 21. For all other sections, it is taken as 1.0. Previously it was calculated for all section types using the same expressions that are applied for the listed flanged sections.</p> <p>(6.) When the section width-to-depth ratio is less than 0.3 (<math>b/h &lt; 0.3</math>), the <math>c</math> factor (SP16.13330.2017 9.2.5) is now taken equal to 0.3.</p>

*	Ticket #	Description
		<p>(7.) For I-shaped sections buckling in the minor direction, the column curve used has been changed. For major-axis bending, the column curve "a" is used for rolled sections with a section depth larger than 500mm, and the column curve "b" is used for all other cases. However, the column curve "c" is now used for minor-axis bending per the new 2017 specification SP 16.13330.2017 7.1.3, Table 7. Previously the column curve "b" was used for minor-axis bending. For singly-symmetric I-sections, the column curve "c" is now used both major and minor directions.</p> <p>(8.) When <math>\lambda_{bar} &lt; 0.6</math>, the <math>\phi</math> buckling factor is now taken as 1.0 for buckling curves "a" and "b" (SP 16.13330.2017 7.1.3).</p> <p>(9.) For singly-symmetric sections, the interaction ratios calculated for Eq. 105 and Eq. 106 of SP 16.13330.2011 section 9.1.1 now consider different compression-side or tension-side section moduli, as appropriate. Previously the minimum section modulus was always used, which could have been over-conservative.</p> <p>(10.) The interaction equations Eq. 41 and Eq. 43 of SP 16.13330.2011 section 8.2.1 are now only checked if the axial force in the member is small, specifically if <math>N &lt; (0.1 * R_y * A)</math>. Previously these equations were checked regardless of the axial force, and the design could be over-conservative for larger axial force.</p> <p>(11.) When simply-supported beams are checked against shear at the support stations, flanges are now ignored in Eq. 42 used for shear strength calculation. Therefore, the static moment of area is now calculated using only the web. For example, for the middle point of the I-section of depth h and web thickness <math>t_w</math>, the static moment is calculated as follows: <math>S = (h/2 * t_w) * h/4</math>.</p>
*	2235	An enhancement has been made to update the Russian steel frame design code to SP16.13330.2017 from the previous SP16.13330.2011 version.
	2787	An enhancement has been made to the steel frame design code "Chinese 2018" in which the factor $\gamma_{RE(S)}$ is now taken as 0.8 when the beam is designated as a Flexo-Compression member. It is taken as 1.0 when the beam is not designated as a Flexo-Compression member. Previously its default value was 1.0 irrespective of whether the beam's designation was a Flexo-Compression member or not.
	2999	An enhancement was made to concrete frame design per the Russian code SP 63.13330.2012 to allow two additional input parameters for seismic factor in the Design Preferences: $m_{tr,flexure}$ and $m_{tr,shear}$ . The parameter $m_{tr,flexure}$ is applied to enhance the steel and concrete design strengths $R_b$ and $R_s$ when the design load combination contains an earthquake load case. It is typically equal to 1.2 per Table 6 of SP 14.13330.2014 and used for column NMM (PMM) design, beam flexure design, and beam torsion design. The parameter $m_{tr,shear}$ is also applied to enhance the steel and concrete design strengths $R_b$ and $R_s$ when the design load combination contains an earthquake load case. It is typically equal to 1.0 per Table 6 of SP 14.13330.2014 and used for column shear design and beam shear design. In addition, the values of the relative neutral axis depth, $\xi_{iR}$ , is reduced for beams by a reduction factor based on seismicity when a seismic load is present in the load combination. The reduction factor is equal to 0.85 for Seismic Intensity 7, 0.7 for Seismic Intensity 8, and 0.5 for Seismic Intensity 9. However, this reduction is not performed when the structure is declared Non-Seismic, even if the load combination is seismic.

**Design – Shear Wall  
Enhancements Implemented**

*	Ticket #	Description
	2850	An enhancement was added for AS3600-2009 shear wall design where boundary zone design is now performed when compressive stress in extreme fiber exceeds the limit of $0.15f'_c$ . Previously this limit was $0.2f'_c$ .

## Results Output and Display Enhancements Implemented

* Ticket #	Description
502	An enhancement has been made to output nonlinear energy components (nonlinear hysteretic damping and nonlinear viscous damping) per group for nonlinear load cases. The nonlinear energy is reported separately for model objects (internal element energy) and support elements (support element energy). The energy output is available in the "Element Nonlinear Energy By Group" table and "Energy by Group" type function in using the Display > Plot Functions command.
* 2047	The Performance Check feature (Define menu > Performance Check) has been enhanced to provide greater control over the calculation and tabular output of the demand-over-capacity ratio (D/C ratio) for the performance check of the whole model, as well as for each object individually. The performance-check feature is available for nonlinear static and nonlinear time-history load cases. A performance check can now include acceptance criteria from links, strain gauges, pier and spandrel forces, and panel zones, along with frame and wall hinges which were previously available. Multiple demand sets can be specified in a performance check definition, allowing more control over the performance check results. To calculate the D/C ratio for an object, the demand from each load case present in a demand set will be combined using the specified combination method (Maximum, Mean, SRSS, or Absolute Add). The combined D/C ratio for each demand set in the performance check will be combined using either Maximum or using Mean plus a multiplier times standard deviation. Tabular results are available under Analysis Results > Structure Output > Performance Check Output. Performance check results can be displayed graphically using the command Display > Performance Check.
* 3060	Two new plot items have been added for displaying results: (1.) A new plot function "Acceptance Criteria D/C Ratio" has been added. This plot function can be used to display the demand-over-capacity ratio (D/C ratio) for a specified group and specified performance level for all steps of a multi-stepped load case (e.g. time-history). (2.) A new menu item (Display > Performance Check Usage Ratio Diagram) which shows the demand-over-capacity ratio (D/C ratio) for all demand sets in a performance check and for a specified performance objective. This display is a visualization tool to show the relative contribution of each demand set and/or object type in a performance check.

## User Interface Enhancements Implemented

* Ticket #	Description
2650	The order of load cases shown in drop down lists for definition forms as well as for results-display forms have been made consistent and are listed in the order the load cases are defined. The order of load combinations shown in drop down lists for definition forms as well as for results-display forms have been made consistent, and are now listed alphabetically. This enhancement was implemented for previously released ETABS v18.0.2 but was inadvertently omitted from the Release Notes.

## Graphics Enhancements Implemented

* Ticket #	Description
2774	A new option is available to set the color of strain gauges on the "Set Display Colors" form.

**Database Tables**  
***Enhancements Implemented***

*	Ticket #	Description
	2836	An enhancement was made to add new capability to perform filters and sorts in displayed database tables by right-clicking in the table column headers. These new capabilities work with a single column at a time. Advanced filtering and sorting are still available using the Format-Filter-Sort menu in the table display form.

**Structural Model**  
**Incidents Resolved**

* Ticket #	Description
2676	An incident was resolved where Check Model operation sometimes generated incorrect "too close" messages for shells that were not actually overlapping. This did not affect analysis results.
* 2959	The self-weight calculation for layered shell sections has been changed. Previously, a layer contributed to the self-weight of the section if the modeling type was "Shell" or "Membrane" and either S11 or S22 was active. This behavior has been changed such that a layer contributes to the self-weight of the section if the modeling type is "Shell" or "Membrane" and any of the directions (S11, S22, or S12) is active. This change only affects the self-weight of layered shell sections that have a layer with only S12 enabled. If the user wants this layer not to contribute to self-weight, then it should be assigned a material property with a weight density of zero. Note that this change only affects self-weight; self-mass was already being considered for all "Shell" and "Membrane" sections. Now self-weight and self-mass are consistent.
3021	An incident was resolved where specified boundary zones lengths in the Wall Hinge Reinforcement form were not being converted correctly to the database units. This could cause the values to be saved incorrectly, or it could generate error messages when changing values in the form. Note that database units are those set when the model is first created or imported. Typically, the length database units are mm or inch when a model is created, and the error would likely occur if different units were used to define the length of the Start Zone or End Zone.

**Section Designer**  
**Incidents Resolved**

* Ticket #	Description
2900	An incident was resolved where trying to display moment-curvature relationship of a concrete section in Section Designer would result in an error condition if the Young's Modulus of concrete was artificially set too low relative to its value based on the concrete strength specified. This condition is now trapped.

**Loading**  
**Incidents Resolved**

* Ticket #	Description
3339	An incident was resolved for the NBCC 2015 auto seismic load case for shear wall building systems where the minimum base shear was incorrect. Minor changes have also been made to the auto seismic report that do not affect results.

**Analysis**  
**Incidents Resolved**

* Ticket #	Description
2670	An incident was resolved where an error message was generated when trying to run a second analysis immediately after running the analysis for the first time with the model unlocked and "Calculate Diaphragm Centers of Rigidity" option selected. When this occurred, the second analysis did not run, and results were not available for the load cases selected to run. Subsequent attempts at running the analysis did not generate further errors.

*	Ticket #	Description
*	2852	An incident was resolved where changing parameters for a Response Spectrum Load Case may not reflect in the torsional response if the change to the case was made with the model locked. The additional associated torsional loads due to accidental torsion were not getting properly updated. The translational lateral loads were updated, but the old torsional loads were still being used. Re-running the load case after unlocking the model would correct the problem. Now the torsional loads are being updated even if the model is locked.
	3323	An incident was resolved where a modal time-history load case would not start during an analysis if it used the modes from a modal case having a nonzero frequency shift that was run at the same time. The modal time-history load case would complete during a subsequent analysis if the prerequisite modal case had been run in a previous analysis. This was an inconvenience only. No results were affected. This incident was resolved in version v18.0.0 but inadvertently omitted from the Release Notes.
	3324	An incident was resolved where the nonlinear behavior of a Directional-type layer of a layered shell defined with a material property that has a non-zero friction angle did not behave as documented in the Analysis Reference Manual. The behavior is now changed so that the friction behavior uses the average of the axial force in the S11 and S22 directions of the layer, if both directions are active; this is suitable for materials like sand where there is no distinct slip plane. If only one direction is active in the layer, the axial force of the active direction is used for computing the friction behavior; this is suitable for the case where the slip plane is perpendicular to the axial load. The shear-friction behavior is modeled as a Wen hysteresis model where the force-deformation relationship is smooth between the initial shear stiffness and sliding behavior. This change does not affect the behavior of Directional shell layers defined using a material property that has a zero friction angle, which is the default for all materials. Likewise, the behavior of Coupled layers is not affected. This Incident was resolved for previously released ETABS v18.0.2 but was inadvertently omitted from the Release Notes.
	3325	An incident was resolved where a model with many parametric PMM hinges running multiple time history load cases may cause the ETABS analysis process to consume a large amount of memory or, in some cases, to terminate with an "Out of Memory" error. This issue could be mitigated by manually running the load cases in several batches with the Analysis Process Options set to "Separate Process". The accuracy of results obtained from analysis were not affected. This Incident was resolved for previously released ETABS v18.0.0 but was inadvertently omitted from the Release Notes.
	3326	An incident was resolved where an error message would be generated at the start of analysis for models containing a layered shell property having a layer with "Coupled" behavior and using a concrete material property where the last point of the stress-strain curve on the compression side had zero stress. This would occur because the Darwin-Pecknold model used for coupled concrete behavior requires non-zero stress values for the compressive material behavior. When this error occurred, no analysis results were available. The behavior has been changed so that all compressive stress values in the stress-strain curve that are smaller in magnitude than 0.0001 times the peak compressive stress will be set to that small limiting value. This change only affects the behavior of the material when used in a Darwin-Pecknold model for "Coupled" shell layers. This Incident was resolved for previously released ETABS v18.0.2 but was inadvertently omitted from the Release Notes.
*	3327	An incident was resolved where an abnormal termination error occurred when a nonlinear direct integration time history case with ground displacement loading was run separately from the nonlinear case it was continuing from. The results for the nonlinear direct integration time history case were not available. This Incident was resolved for previously released ETABS v18.0.2 but was inadvertently omitted from the Release Notes.

* Ticket #	Description
* 3328	An incident was resolved where the analysis could terminate with a fatal error when a staged-construction load case was run that contained the operation Change Section on link objects, and was then followed in the same run by a nonlinear static, direct-integration time-history, or another staged-construction load case that continued from the same previous stiffness case as the stage-construction case with the Change Section operation. For example, consider initial case LC0 followed by staged-construction case SC1 and separately by nonlinear static case LC1. If case SC1 changes link sections, then case LC1 may fail if it runs after SC1. This error did not occur if LC1 was run in a separate instance from SC1. This error did not occur when the link section properties were of the same type and size (number of data points for multi-linear properties) before and after the section change operation. When this error occurred, analysis results were lost. Otherwise, results were not affected. This incident was resolved for previously released ETABS v18.0.0 but was inadvertently omitted from the Release Notes.

### Design – Concrete Frame Incidents Resolved

* Ticket #	Description
2756	An incident was resolved for concrete frame design per the IS 456:2000 code where the Q factors were not being updated when lateral load cases were updated or deleted without unlocking the model. Rerunning the load cases after unlocking the model would update the Q factors.
2868	An incident was resolved for the concrete frame design code "ACI 318-14" where the torsion design of concrete beams could be under-reinforced when $T_u$ fell between the threshold limit and the cracking value ( $T_{th} < T_u < T_{cr}$ ). In this case the required torsion rebar ( $A_t/s$ and $A_l$ ) was not calculated per section 22.7.6.1 and only the minimum rebar was being provided per section 9.6.4.2. The calculation of rebar areas $A_t/s$ and $A_l$ was correct for $T_u < T_{th}$ and for $T_u > T_{cr}$ .

### Design – Steel Frame Incidents Resolved

* Ticket #	Description
2788	An incident has been resolved for the steel frame design code "Chinese 2018" where the $\phi_b$ factors were not exactly correct for members that had a linear moment diagram. The $\beta_b$ factors were not being overwritten correctly by the equation given in Item 10 of GB50017-2017, App C, Table C.0.1.
2943	An incident was resolved in the Canadian steel frame design code CSA S16-14 where the section classification for tee, angle, and double-angle sections had an error in the determination of the slenderness for the stem (tee) and the vertical leg (angle and double-angle). Previously the flange or horizontal leg $b/t$ ratio was being used while checking for the stem or vertical leg slenderness of these sections. Note that the stem or vertical leg is parallel to the local-2 axis of the section.
3337	An incident has been resolved in the New Zealand steel frame design code "NZS 3404:1997" in which the value of slenderness $\lambda_e$ for flanges was conservatively taken based on the full width rather than clear width. For example, for I-shaped sections, the $b/t$ for flange was being taken as $(b_f/2)/t_f$ instead of $(b_f - t_w)/(2 * t_f)$ . Similarly, the $b/t$ for webs are also considered. The result was slightly conservative.

## Design – Shear Wall Incidents Resolved

* Ticket #	Description
2351	An incident was resolved for shear wall design per the ACI 318-14, ACI 318-11 and ACI 318-08 codes where ACI Equation 11-28 was not properly disregarded when term $(\mu_u/\nu_u - l_w/2)$ was negative. This was limiting the shear capacity to $0.6\lambda f_c'$ instead of using ACI equation 11-27. Results were always conservative.
2680	An incident was resolved for shear wall design of piers where, for certain sections, the pier was broken into more legs than necessary. This happened when area objects comprising a co-linear leg were drawn out of sequence.
2851	An incident was resolved for Eurocode 2-2004 concrete frame shear design where the shear capacity $V_{rdc}$ could be incorrect when the model database units were not in N-mm. Note that the database units are those in effect when the model is created or imported, regardless of the current units used for display.
2884	An incident was resolved where the pier and spandrel shear capacity for CSA A23.3-14 code could be incorrect because the $E_{psx}$ value was using the rebar ratio instead of the rebar area. Shear design reporting has been enhanced to report the values of $A_{st}$ , $E_{psx}$ , beta and theta values for pier and spandrel design. Boundary zone reporting has also been enhanced.

## Design – Composite Beam Incidents Resolved

* Ticket #	Description
2820	An incident was resolved that affected the composite design of beams whose Vibration Criterion was set to Rhythmic in their composite beam design overwrites. Such beams were not being designed but instead displayed a generic error message. This error did not occur when the Vibration Criterion was set to Rhythmic in the design preferences, only when using the design overwrites. This issue affected ETABS versions 17.0.0 to 18.0.2.
2855	An incident was resolved which affected the composite design of beams supported by girder beams whose own design method was set to None. Unless the Vibration Criterion was set to None in the composite beam design preferences or in the composite beam design overwrites of the composite beams framing into the girder beams, such composite beams could not be designed but instead displayed a generic error message. This issue affected ETABS versions 17.0.0 to 18.0.2.
3189	An incident that affected composite beam design per the various CSA design codes has been resolved. When designing composite beams per any of the CSA design codes and evaluating the deflections caused by concrete shrinkage, shrinkage-specific composite section properties were not being computed, but instead the regular composite section properties were being used. Because the shrinkage-specific composite section properties are computed based on a reduced age-adjusted modulus of elasticity for concrete, the design underestimated the deflections caused by concrete shrinkage. The strength of the beams was not affected, and the effect on their serviceability was minor. All previous versions of ETABS that designed composite beams per the various CSA design codes were affected.
3376	An incident was resolved which affected the design of composite beams with a cover plate per either of the AISC 360-16, CSA S16-14, or Eurocode 4-2004 codes. These codes include a bottom flange stress check and ETABS would not produce any design for beams with a cover plate because it mistakenly evaluated their bottom flange stress as infinity. This could be observed when attempting to design the beam interactively. This incident affected all versions of ETABS greater than v17.0 and when it occurred, no designs were produced for these beams. The bottom flange stress is now properly evaluated for beams with a cover plate.

*	Ticket #	Description
	3396	An incident was resolved which affected the composite beam design of girders loaded by beams when the selected design code was BS5950-1990, CSA S16-14 or Eurocode 4-2004 and the model database units were US customary units. If the point loads applied by the various beams supported by the girder differed from one another, the equivalent line load on the girder was incorrectly reported in the vibration check girder output – and could even be reported as negative. This was a reporting error only and the reported equivalent line load was not being used when computing the girder deflection. The strength and serviceability of the designed girders were unaffected. The equivalent line load is now reported correctly.

## Design – Slab Incidents Resolved

*	Ticket #	Description
	2870	An incident was resolved where the Slab PT stresses used for the stress check under service loads could be incorrect in versions 17.0.0 to 18.0.2. The stresses could be calculated at the wrong point, and when incorrect they were reported too high. This affected the reported demand-capacity ratio. Strength design and analysis results were not affected. The corresponding verification examples have been changed to reflect values reported in the "Concrete Slab Design" tables rather than the analysis tables. The change was insignificant for the purposes of these verification examples.
	2871	An incident was resolved where slab-strip integrated moments were not accounting for any slab vertical offsets specified if prestressing tendons were not present in the model. No analysis results were affected, and this was only a post-processing issue. This affected versions 17.0.0 to 18.0.2.
*	3259	An incident was resolved where the slab strip design for one-way shear could have been incorrect for single-valued combinations. This error did not affect flexural design or punching shear checks. This error affected ETABS versions 18.0.0 to 18.0.2.

## Results Display and Output Incidents Resolved

*	Ticket #	Description
	2574	An incident was resolved where animations generated using the command File > Create Video > Multi-step Animation were always plotting relative displacements even when the option Absolute Displacements was selected. By way of background, absolute displacements include the ground displacements when acceleration loads are applied, whereas relative displacements do not. No other results were affected.
	2635	An incident was resolved where the load pattern, load case, and load combination filters for reports were not being applied. Instead, the latest selections made for displaying tables were being used. Now the selections made in the report options are kept separately from those used for displayed tables. This was a reporting issue only and did not affect results.
	2733	An incident was resolved where design results for wall-pier boundary zone checks were not available in reports. They were available when looking at individual wall-pier detailed design results on-screen. This was a reporting omission. No results were affected.
	2789	An incident was resolved for the steel frame design code "Chinese 2018" where the calculated K factors were not exactly correct for a column connected to a beam that had a moment hinge at the far end or to a column above or below that had a moment hinge at the far end. Both sway and non-sway moment frames were affected. Similarly, the K factors were affected for the sway frames of all other codes for the columns connected to beams or columns (above or below) that had moment hinges at their far ends. The error was small and the effect on design results was generally insignificant.

*	Ticket #	Description
*	2949	<p>An incident was resolved where, for nonlinear static, staged-construction, direct-integration load cases, and sequences of such load cases, the frame member forces and stresses used for display and design could have been incorrect for a specific frame member when the following conditions are met:</p> <p>(1.) The load case (or sequence of load cases) contained more than one load pattern that applies loads directly to the frame member, including at least one load pattern with self-weight loads.</p> <p>(2.) A load pattern containing self-weight load (A) was applied after another load pattern containing frame loads (B); in other words, A was listed after B in the list of applied loads in the load case definition (or sequence).</p> <p>(3.) The load patterns A and B had different scale factors.</p> <p>Frame members that did not meet the above conditions, such as those without frame loads assigned to the load patterns used in the load case sequence, were not affected. Load case sequences without self-weight applied were not affected. Load case sequences having only one self-weight load pattern applied and with that load pattern being listed first in the load case definition were not affected. Note that this error did not affect how the frame loads were transferred to the structure, and therefore all other analysis results were correct (displacements, reactions, forces and stresses in other objects). Nonlinear behavior, including frame hinges and P-delta, was not affected. Only the reported forces and stresses within the affected frame members themselves were in error, including the forces used for frame design of those members. Because self-weight is most commonly applied first, most models were not affected by this error. Linear load cases were not affected, even if they used the stiffness from a nonlinear load case. This error affected ETABS versions 17.0.0 to 18.0.2.</p>
*	3045	<p>An incident was resolved where, for nonlinear static, staged-construction, direct-integration load cases, the frame member forces and stresses used for display and design could have been incorrect for a specific frame member when the following conditions were met:</p> <p>(1.) Results were requested for multiple load cases at the same time, including situations where one or more requested load combinations referenced multiple load cases.</p> <p>(2.) Among all the requested load cases, only one of these was a nonlinear static, staged-construction, or direct-integration time-history load case, and only a single step was requested from that load case.</p> <p>(3.) Among all the requested load cases, at least two of these were linear load cases (including modal or response-spectrum), at least two of these load cases used the stiffnesses from different nonlinear load cases (or zero initial conditions), and at least one of these was a linear static, linear multi-step static, or modal time-history load case.</p> <p>(4.) The affected frame object had loads assigned to it as part of the single nonlinear load case. This could include self-weight.</p> <p>This was not common. When this error occurred, the frame response reported in the affected object could be incorrect at all stations along the length of the object except at the start (l end). For frame objects that were discretized into multiple elements for analysis, the results would be correct at the start of each element and deviate along the length of the individual elements. This deviation in response, when present, was due to using the wrong element load for equilibrium calculations. This error would be most likely to affect table results when multiple load cases or load combinations were requested, and frame design when the load combinations used satisfy the conditions listed above. Note that this error did not affect how the frame loads were applied to the structure, and therefore all other analysis results were correct (displacements, reactions, forces and stresses in other objects).</p>
	3081	<p>An incident was resolved where refreshing plot functions or quick hysteresis plots after unlocking and re-running the analysis could generate an error or lead to unexpected termination of the software. The results were not affected.</p>

*	Ticket #	Description
*	3084	<p>An incident was resolved where tabulated analysis results may have been displayed as zero for load combinations that contained one or more multi-step load combinations. A load combination can be multi-step if the load combination is of type Linear Add and the load combination contains a multi-step nonlinear static or time-history load case or another multi-step load combination. When this issue occurred, the tabulated results for a Linear Add type load combination that contained a multi-step load combination could have been displayed as zero. This issue only occurred when the following conditions were met:</p> <p>(1.) The model included multiple load combinations that each contained the same multi-step Linear Add type load combination (say "A") in its definition.</p> <p>(2.) In the Choose Tables for Display form (Display menu &gt; Show Tables), more than one of the containing load combinations were selected for output.</p> <p>(3.) At least one of the selected load combinations containing combination "A" must have been of type Linear Add, and at least one must have been of a different type (e.g. Envelope).</p> <p>(4.) In the Database Table Output Options form (Modify/Show Options button in the Choose Tables for Display form), "Multiple Values, If Possible" was selected for Load Combination Results.</p> <p>Any load combination containing affected load combinations could also be affected. This issue only affected the tables under the Analysis Results section. This issue did not affect visually displayed results or design results. This issue did not affect load combinations that were not nested. This issue was only present in ETABS versions 18.0.0 to 18.0.2.</p>
	3200	<p>An incident was resolved where the strip design details report was only showing text values for the top longitudinal rebar, omitting the bottom values, on the longitudinal rebar diagram. This was a reporting issue only and did not affect results.</p>
	3223	<p>An incident was resolved where, for a certain model, a load combination that included response spectrum loads was only displaying the maximum results instead of both the maximum and minimum results in the on-screen display and in database tables. This was not common.</p>
	3377	<p>An incident was resolved that corrected two issues with pushover analysis:</p> <p>(1.) The monitored displacement shown using the command Display &gt; Static Pushover Curve was always zero for nonlinear static load cases where the load application control was set to "Full Load".</p> <p>(2.) For nonlinear static load cases where the load application control was set to "Quasi-Static (run as time history)" and the monitored displacement was a generalized displacement, only the first item in the generalized displacement definition was being used as the monitored displacement.</p> <p>Neither of these issues affected displacement-controlled nonlinear static load cases, which are the type most commonly used for pushover analysis.</p>

**User Interface**  
**Incidents Resolved**

*	Ticket #	Description
	1414	<p>An incident was resolved where columns cross-sections were not shown in Plan views when the shading option was selected using command Options &gt; Graphics Preference for Standard Graphics. This was just a display issue and no results were affected.</p>
	2083	<p>An incident was resolved where link objects were always present in the view even if they fell outside the specified X and Y limits set using command View &gt; Set Building View Limits.</p>

* Ticket #	Description
2394	An incident was resolved where the Wall Hinge Reinforcement assignment form did not behave correctly when used following the import of the model text file (.E2K, .SET). For example, the bar size and number of bars could not be changed, the stations in the Reinforcement section were all reported as "Start", and program-computed "Number of Bars" values were incorrect. Results were not affected if the OK button was not clicked after viewing the form. Opening the model from the .EDB file did not have this issue.
2505	An incident was resolved where items in the Tools menu disappeared when changing to an alternate language.
2578	An incident was resolved where wall stacks were not being drawn for just the selected stories but were instead being drawn for all stories in a single tower.
2838	An incident was resolved related to editing the definition of section-cut quadrilaterals when more than one quadrilateral was defined. When using arrow buttons to move between the definitions for the different quadrilaterals, the coordinates of the quadrilateral being edited may have been copied from a different quadrilateral. Section-cut results were consistent with the quadrilaterals as ultimately defined. No other results were affected.
2970	An Incident was resolved where the boundary zone lengths could not be specified for the wall hinge reinforcement assignment when Chinese language was chosen for the user interface.

### Drafting and Editing Incidents Resolved

* Ticket #	Description
2497	An incident was resolved where, when the graphics mode was set to "Standard Graphics", drawing a line strain gauge and ending it with a right click could cause an additional line gauge to be added that connected to a random point. No results were affected.

### Graphics Incidents Resolved

* Ticket #	Description
2084	An incident was resolved where a drawn link object was not visible in the model window when a link was added if the link visibility was turned off in that view using command View > Set Display Options. This issue then prevented the newly drawn link from being displayed later in that view even after link visibility was turned back on. Only using the command View > Show All Objects would restore visibility.
2495	An incident was resolved that addressed two issues with DirectX graphics mode: (1.) Switching between plan-view levels using the "Move Up" or "Move Down" buttons could cause the dimension lines to disappear. (2.) The model sometimes disappeared when displaying the deformed shape in a plan view. These issues were graphical only, and no results were affected.
2504	An incident was resolved where point springs were not visible at point objects that were not associated with any tower. This issue did not affect models that never enabled the multi-tower option. This was a display issue only. No results were affected.
2585	An incident was resolved where Greek and Cyrillic symbols (obtained by using the Alt+### key combination) were not shown using the DirectX graphics option. The basic alphabetic characters have been added in the range 913 to 969 (Hex 391 to 3C9) and 1040 to 1103 (Hex 410 to 44F), respectively.
2595	An incident was resolved where the menu command View > Show Rendered View was inadvertently available in the user interface. This command has now been removed since DirectX graphics mode provides better rendered views directly in the model windows.

* Ticket #	Description
2636	An incident was resolved for DirectX graphics where the right-click display of contour plots of forces, moments, or stresses for an individual area object were sometimes upside down with respect to the contour plots of the same object in the model window. This was a display issue only and no results were affected.
2683	An incident was resolved where diaphragm forces (command Display > Force/Stress Diagrams > Diaphragm Forces) were not shown in DirectX graphics mode. This display worked correctly in Standard Graphics mode.
3044	An incident was resolved where the command Draw > Draw Section Cut was not working for DirectX graphics. Standard Graphics mode was not affected.

## Data Files

### Incidents Resolved

* Ticket #	Description
2880	An incident was resolved where opening a very large model file could take a much longer time than in v17. The issue was caused by additional checking for duplicated points, which was introduced in v18. This check has now been made more efficient.

## Database Tables

### Incidents Resolved

* Ticket #	Description
2583	An incident was resolved where results for section cuts and generalized displacements would not show up in the output tables if their definitions had been imported from database tables or created using the interactive database editor. This issue was resolved for previously released ETABS v18.0.2 but was inadvertently omitted from the Release Notes.
2747	An incident was resolved where a fatal import error was reported when importing certain database tables if there were no records in the table. This has been changed to a warning so that the import can continue.
2793	An incident was resolved where the item "Floor Area" in the "Material List by Story" table had units of length instead of length <sup>2</sup> (area). When the table was displayed in units different from the database units, the values shown were incorrect. Note that database units are those used internally, and these are set when the model is first created or imported.
3027	An incident has been resolved where frame section properties specified as "From File" = "Yes" may be deleted upon import if the specified property file was not found or if the specified section property was not found in the property file. Now the frame section and its saved section-property values are imported, even if the original file source is not available.
3091	An incident was resolved where importing more than 20 rows in the Pier Label Definitions or Spandrel Label Definitions tables resulted in an error. This also affected Interactive Database Editing. No results were affected, but the tables could not be fully imported when this occurred.
3137	An incident was resolved that addressed three issues with the frame section property files (XML): (1.) The EDI labels were incorrect for some sections in the frame section property files for the AISC14, AISC14M, AISC15, AISC15M, Australia-NewZealand, CISC9, CISC10, Euro and Nordic section database. This label is used by ETABS to determine the shape types to be used for steel frame design. This issue affected ETABS versions 18.0.0 to 18.0.2. (2.) Missing auto select lists were added to the AISC14 and AISC15 property files. (3.) As an enhancement, a new section property file was added for Korean frame sections.

* Ticket #	Description
3241	An incident was resolved where the table of auto-seismic load pattern definitions was showing certain spurious values in the records for the defined seismic load with multiple directions of loading. These values (period used, weight used, base shear) were meaningless for multiple directions of loading and have been removed. The records for the individual loading directions did, and still do, show these values correctly.

### Application Programming Interface (API) Incidents Resolved

* Ticket #	Description
2541	An incident was resolved for the import of P, V2, V3, T, M2 and M3 hinge definition tables where fields with the field keys SSPOSFORSF and SSNEGFORSF were not fully implemented and could in some cases lead to import errors being reported. This issue was resolved for previously released ETABS v18.0.2 but was inadvertently omitted from the Release Notes.
* 2843	An incident was resolved for the Application Programming Interface (API) where a load combination created using the methods in cCombo sometimes did not display the correct envelope results. This issue did not affect load combinations that were created using the graphical user interface (Define > Load Combinations) or through Interactive Database Editing (Edit > Interactive Database). This issue only affected load combinations created in versions 18.0.0 to 18.0.2 using the API.

### External Import and Export Incidents Resolved

* Ticket #	Description
2917	An incident was resolved where the Story Drift and Story Forces tables might not export correctly to Excel from the File menu. When this occurred, the exported tables were blank or incomplete. This behavior was intermittent and was due to parallelization of the export of these tables. No results were affected.

### Documentation Incidents Resolved

* Ticket #	Description
2643	An incident was resolved where the items listed on the Index tab of the context sensitive help (F1) were truncated by one character. This was only an issue in the help and had no impact on the functionality of ETABS.
2866	An incident was resolved to correct one minor typographical error each in Table 3-1 for Singly Symmetric I-Shaped sections (Flexure in Web) and Table 3-2 for Doubly Symmetric I-Shaped sections (Axial only compression in flanges of built-up I-Shapes) of the AISC 360-10 and AISC 360-16 Steel Frame Design manuals. These were documentation errors only and no results were affected.
3175	An incident was resolved where Equation 4-7 of the Material Time-Dependent Properties Technical Notes was incorrect. The equation has been corrected in the documentation. This was a documentation issue and did not affect the results.