

# ETABS® Version 18.0.2 Release Notes

© Copyright Computers and Structures, Inc., 2019

**Notice Date: 2019-09-03**

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 previous Release Notes. Emails sent when an Incident is released will now indicate this Ticket number as well.

## **Changes from v18.0.1 (Released 2019-06-27)**

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

| * | Ticket # | Description  |
|---|----------|--|
|   | 2570     | The version number has been changed to v18.0.2 for a new intermediate release. |

### **Structural Model Enhancements Implemented**

| * | Ticket # | Description   |
|---|----------|---|
|   | 2661     | The New Zealand material property library has been updated to include masonry, aluminum, cold-formed, and AS/NZS3678:2016 Grade 450PL steel materials. The steel-material naming convention has also been updated to include relevant plate-thickness ranges. |

### **Design – Concrete Frame Enhancements Implemented**

| * | Ticket # | Description  |
|---|----------|--|
| * | 2634     | An enhancement has been made to concrete frame design for those codes where column design requires a beam-column capacity check for seismic design. The column moment is now checked for capacity in one direction (major or minor) at a time, using the axial load from each considered design combination, and while taking the moment in the other direction from the load combination. Affected codes are “ACI 318-14”, “ACI 318-11”, “ACI 318-08”, “CSA A23.3-14”, “Eurocode 2-2004”, “IS 456:2000”, “Italian NTC 2008”, “KBC 2016”, “KBC 2009”, “Mexican RCDF 2017”, “Mexican RCDF 2004”, “TS 500-2000”, and “TS 500-2000(R2018)”. Previously the moment considered in the second direction was taken to be zero since version 18.0.0. |

## Design – Steel Frame Enhancements Implemented

| * | Ticket # | Description   |
|---|----------|---|
| * | 2739     | <p>The following enhancements have been made to steel frame design per the Russian SP 16.13330.2011 code:</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 1.0 for double-angle sections, all based on the 2017 specification SP 16.13330.2017 Annex G. Previously the value of <math>K</math> for double-angle sections was taken mistakenly as 1.2. All other values of <math>K</math> remain the same as before. 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). The value of <math>\psi</math> for doubly-symmetric sections is calculated following the section per SP 16.13330.2011 Table G.1 and Table G.2. For singly-symmetric section, the parameter <math>\psi</math> (or <math>\psi_a</math>) is calculated based on SP 16.13330.2011 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>. Previously there was an error in the interpolation. This interpolation error has been corrected in the current version. The parameter <math>\psi</math> is used to calculate <math>\phi_b</math>.</p> <p>(3.) For I-shaped sections buckling in the minor direction, the assumed column curve has changed. For major-axis bending, column curve “a” is used for rolled sections with a section depth larger than 500mm, and column curve “b” is used for all other cases. However, 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 column curve “b” was used for minor-axis bending.</p> <p>(4.) 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 modulus, as appropriate. Previously the minimum section modulus was always used, which could have been over-conservative.</p> <p>(5.) 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> |

## Database Tables Enhancements Implemented

| * | Ticket # | Description  |
|---|----------|--|
| * | 2562     | An enhancement was made to add a Tables tab to the model explorer. This can now be used to display individual tables or to open the interactive database editor.   |
|   | 2567     | <p>The following database tables have been implemented:</p> <ul style="list-style-type: none"> <li>Pier and Spandrel Section Properties tables</li> <li>Auto Seismic Calculation tables - Base Shear, Period, Cs, etc.</li> </ul> <p>These were present in v17, but were omitted with the interactive database editing feature that was added for v18.0.0.</p> |
|   | 2716     | An enhancement has been made to the database tables to add design load combination data. Selected design load combinations are listed in the tables under Design Data > Design Definition Data > Design Load Combinations and can be altered through the interactive database.   |

## Installation and Licensing Incidents Resolved

| * | Ticket # | Description   |
|---|----------|---|
| * | 2737     | An incident was resolved where, when using a Chinese Nonlinear license, Chinese design codes were not available. This issue has been resolved and Chinese design codes are now made available for all Chinese licenses. |

## Section Designer Incidents Resolved

| * | Ticket # | Description   |
|---|----------|---|
|   | 2271     | An incident was resolved for Section Designer where the option to display the PMM surface curves with and without the phi factor should not have been available for the following codes: BS 8110-97, Hong Kong CP 2013, Singapore CP 65:99, TS 500-2000(R2018) and TS 500-2000. This option is not applicable for these codes and had no effect on the display. Instead, the PMM display now offers scaling options that are appropriate for each code. No results were affected. |

## Structural Model Incidents Resolved

| * | Ticket # | Description   |
|---|----------|---|
|   | 2621     | An issue was resolved where some auto-generated Parametric PMM hinges based on ASCE41-13 Concrete Column (Table 10-8) created invalid Parametric PMM hinge definitions which would prevent analysis from being able to run. When this issue occurred, results were not available. Models where the analysis was able to be run were not affected. |

## Analysis Incidents Resolved

| * | Ticket # | Description  |
|---|----------|--|
| * | 2639     | An incident was resolved where nonlinear static and staged-construction load cases could converge to the wrong answer under the following circumstances: (1) The nonlinear convergence iteration used line search, which is not the default setting in the nonlinear solution parameters, (2) the line-search iteration was able to find a solution without further iteration, and (3) the unbalance force in the first step of line search was significantly larger (by orders of magnitude) than the applied load. All three conditions were necessary to cause this issue. This error was not common. In ETABS v13.2.0 to v17.0.1, if event-to-event stepping was enabled in the nonlinear solution parameters, which is the default setting, the line-search algorithm was automatically disabled. In ETABS v18.0.0 and v18.0.1, the default nonlinear solution parameters disabled line-search. Situations that can cause item (3) include load steps where the stiffness becomes much larger, such as the closing of gaps with non-zero initial opening. This does not apply to gaps with zero initial opening, such as at the base of a structure. Displacement-controlled nonlinear static load cases, such as for static pushover, do not use line search and were not affected. Nonlinear direct-integration load cases were not affected, even when line search was used. |

**Loading  
Incidents Resolved**

| * Ticket # | Description   |
|------------|---|
| 2604       | An incident was resolved where a Linear Add load combination that contained another load combination of type Envelope, SRSS, Range Add, or Absolute Add may have produced incorrect Min value results when viewed in the tables or display, although the associated Max value results were correct. This issue also affected any load combination that, in turn, contained an affected Linear Add load combination as described previously. This issue did not affect load combinations that were not nested, or nested combinations that contained only other load combinations of type Linear Add.  |
| 2725       | An issue was resolved where design results of both frame and shear wall designs may be incorrect in ETABS v18.0.0 and v18.0.1 when the following two conditions were met:<br>(1) Multi-Response Case Design set to "Step-by-Step" or "Step-by-Step - All" in the design preferences, and<br>(2) Two or more load combinations used for the design reference different multi-step load cases.<br>Multi-step load cases include multi-directional wind and seismic load cases, time history, and nonlinear static or staged-construction load cases that request more than one step. Load combinations that included more than one multi-step load case did not trigger this issue. Multiple load combinations that reference the same multi-step load case also did not trigger this issue. For affected models, the design data including design forces ("Design Force Data" tables) and design output may be incorrect. This issue does not affect the analysis results. Any affected model should rerun the design and verify the results using ETABS 18.0.2. |

**Design – Concrete Frame  
Incidents Resolved**

| * Ticket # | Description  |
|------------|--|
| 2632       | An Incident was resolved for concrete frame design per the IS 456:2000 code where As-min rebar was not correctly enforced at beam-top location for ductile frames.   |
| 2656       | An incident was resolved for NZS 3101-2006 concrete frame design code where design moments in the ductile columns were not accounted for in design. The error was evident in the concrete column design report. This error affected ETABS versions 17.0.0 to 18.0.1. |

**Design – Shear Wall  
Incidents Resolved**

| * Ticket # | Description  |
|------------|--|
| 2569       | An incident was resolved for TS 500:2000(R2018) shear wall design code where the maximum pier shear capacity was incorrectly calculated because of a units-conversion problem. This was only a problem if seismic loads were present and the model units were not N-mm. The maximum shear capacity of pier is now designated as Vm instead of Vu. Also, Seismic Design Grade was removed from Pier Overwrites as this item was not applicable for TS 500:2000(R2018) code. |
| 2577       | An incident was resolved for shear wall design per the Eurocode 2:2004 code where spandrel shear design was not being amplified by the force modification factors specified in the wall design preferences.  |

**Data Files  
Incidents Resolved**

| * Ticket # | Description  |
|------------|--|
| 2556       | An enhancement was implemented where data was added to the ETABS text file for the export and import of database table named sets. |

## Database Tables Incidents Resolved

| * | Ticket # | Description  |
|---|----------|--|
|   | 2498     | An issue was resolved where the "Fiber Hinge Fiber States" table was not able to be displayed in ETABS v18.0.0. This issue was resolved in ETABS v18.0.1 but was inadvertently left out of the ETABS v18.0.1 release notes.  |
| * | 2506     | An incident was resolved where interactive database table data sent to Excel was displayed in Excel in the model database units instead of the current model display units. When this occurred, the values in the Excel file were not consistent with the units displayed in that file. Note that the model database units are those used internally and are set when the model is initially created or imported. Results were correct when the current display units were set to be the same as the database units.         |
| * | 2563     | An incident was resolved where if a load combination had multiple load cases assigned to it, then when the database table was imported using either the menu command File > Import > ETABS Database Tables or Edit > Interactive Database, the first load case in the combination was removed from the load combination definition. This did not affect models that were saved and opened using the model file (.EDB) or edited in the user interface using the command Define > Load Combinations.                          |
|   | 2584     | An incident was resolved where changing the auto load type for an auto seismic load (e.g., from ASCE7-10 load to ASCE7-16 load) in the interactive database editor did not work correctly. Changing from one code to another instead would set the code to None. Changing the code from None to another code would work properly.  |
|   | 2603     | An incident was resolved where "Envelope" type load combinations were reported as "Linear Add" in the Load Combination Definition table in using the commands Display > Show Tables and Edit > Interactive Database Editing. The load combination type was correctly displayed in the Load Combination Data form (Define > Load Combinations).   |
|   | 2662     | An incident was resolved where, when editing the "Load Pattern Definitions", "Load Pattern Definitions - Auto National Loads", and the load assignment tables through the interactive database editor with some of the load patterns deselected in the Select Load Patterns form, the rows of the edited tables which were hidden from view because of the load pattern selection may have been deleted. When this issue occurred, the edited tables reflected this change when all load patterns were selected for display. |
|   | 2679     | An incident was resolved where the tables "Frame Assignments - Hinge - Summary", "Hinge States", "Fiber Hinge States", and "Fiber Hinge Fiber States" displayed the Hinge location as the relative distance of the hinge multiplied by the full length of the element. However, the location of the hinge is the relative distance of the hinge multiplied by the clear length of the element plus the end length offset. The tables have been corrected to accurately list the location of the hinge on the element.        |
|   | 2681     | An incident was resolved where item names using special characters (e.g., "Wind (+)") might in certain instances not import correctly from the interactive database.   |
|   | 2705     | An incident was resolved where the table "Wall Property Definitions - Specified" may show an error when being interactively imported.  |

## Results Display and Output Incidents Resolved

| * | Ticket # | Description  |
|---|----------|--|
|   | 1210     | An incident was resolved for ASCE 7-10 auto seismic load case where Tused and Coefficient used were reported incorrectly in both tabular output and the project report. This was a reporting issue only and computed auto seismic load calculations were not affected. |
|   | 2499     | An incident was resolved where an error condition could occur if the deformed shape was requested from the "Display" option of the Model Explorer. The deformed shape could be displayed by directly going to the menu item or using the toolbar button.               |

| * Ticket # | Description   |
|------------|---|
| 2575       | An incident was resolved where requests to plot input load history (e.g. input acceleration) from a modal time history case resulted in an error and returned zero response. This also affected the input function plot field shown on combined story response plots. This was a plotting issue only. No other results were affected. |

## User Interface Incidents Resolved

| * Ticket # | Description   |
|------------|---|
| 1114       | An incident was resolved where using the Add to Existing option when defining an additional point mass could, in certain instances, add the point mass to a previously deleted point mass assignment.   |
| 1682       | An incident was resolved where the combination type reported in the hovering tooltip for the load combination form was incorrect. No results were affected.   |
| 1728       | An incident was resolved where selecting by material property might incorrectly select an unfilled deck section that had previously been filled with the selected material property. No results were affected.  |
| 1909       | An incident was resolved where the window caption for plan view would show Z=0 instead of the actual value of Z for the view if a user-specified elevation was given for the plan view instead of using a story level or a reference plane.   |
| 2382       | An incident was resolved where database units were always being used instead of the display units shown when defining the "Start Zone Size" and "End Zone Size" for wall hinge reinforcement, using the menu command Assign > Shell > Reinforcement for Wall Hinge > Specified Rebar Layout. Note that database units are those in effect when the model is first created or imported, and cannot be changed. |
| 2383       | An incident was resolved for the ASCE 7-16 auto seismic load where changing the value of Ss or S1 for Site Class A to D did not immediately update the displayed values for Site Coefficients Fa or Fv. These values were correctly updated when the user clicked OK to accept all changes. This was a display issue and results were not affected.   |
| 2631       | An incident was resolved where an abnormal termination of the software could occur when deleting a construction stage using the Tree View option available when defining a staged construction load case.   |

## External Import and Export Incidents Resolved

| * Ticket # | Description   |
|------------|---|
| 2400       | An incident was resolved where changing the Drawing Size Type from ANSI Engineering to ANSI Architectural under "Drawing Options" when using the command File > Export > DXF/DWG File would cause an error condition. No results were affected, but the drawing was not exported. |