

STRESSCHECK 10.4 RELEASE NOTES

MARCH 2018

Changes incorporated in the 64-bit StressCheck 10.4 Professional Edition. All work files (SCW) and project files (SCP) created in StressCheck 10.3 are compatible with StressCheck 10.4. However, any file created in StressCheck 10.4 cannot be opened in previous versions due to the file format changes incorporated into this release to support additional features and future enhancements.

RESOLVED ISSUES

- Several bug fixes including resolving issues with:
 - Automesh
 - Batch Solver
 - Boolean operations
 - BRS and SRS load application
 - COM API functions
 - Contact
 - Cutting Plane
 - Display and UI
 - Error and Warning message handling
 - Extraction methods and saved settings
 - File I/O
 - H-discretization and Autolamination
 - Loads and Constraints
 - Parameters and Formulae
 - TLAP assignments for models with large element count
 - Saved Settings

A more detailed list is provided at the end of the document.

UPDATES

In this release we have incorporated several updates to StressCheck Professional and to the Master Guide documentation. They include new behavior for the Traction Loads; Load record check; new TLAP bearing option; Formulae Importation/Exportation; new Total Displacement extraction function; new Bearing Load error messages; inclusion of Contact Zones on Copy operations; Tabs memory; expansion of the StressCheck Input dialog size; new Option for Automatic Contact Constant generation; Contact Zones shrinking; COM version updated to 3.4; several GUI improvements and added functionality as described in the following.

EXPANDED UI SIZE FOR THE INPUT DIALOG

The size of the Input dialog has been increased to better accommodate for the different selection options (Figure 1), buttons and input fields. Buttons, dialogs and other input have been rearranged for better spacing and the UI is no longer collapsible. Additionally, the Cancel icon has been replaced with a “Deselect” button.

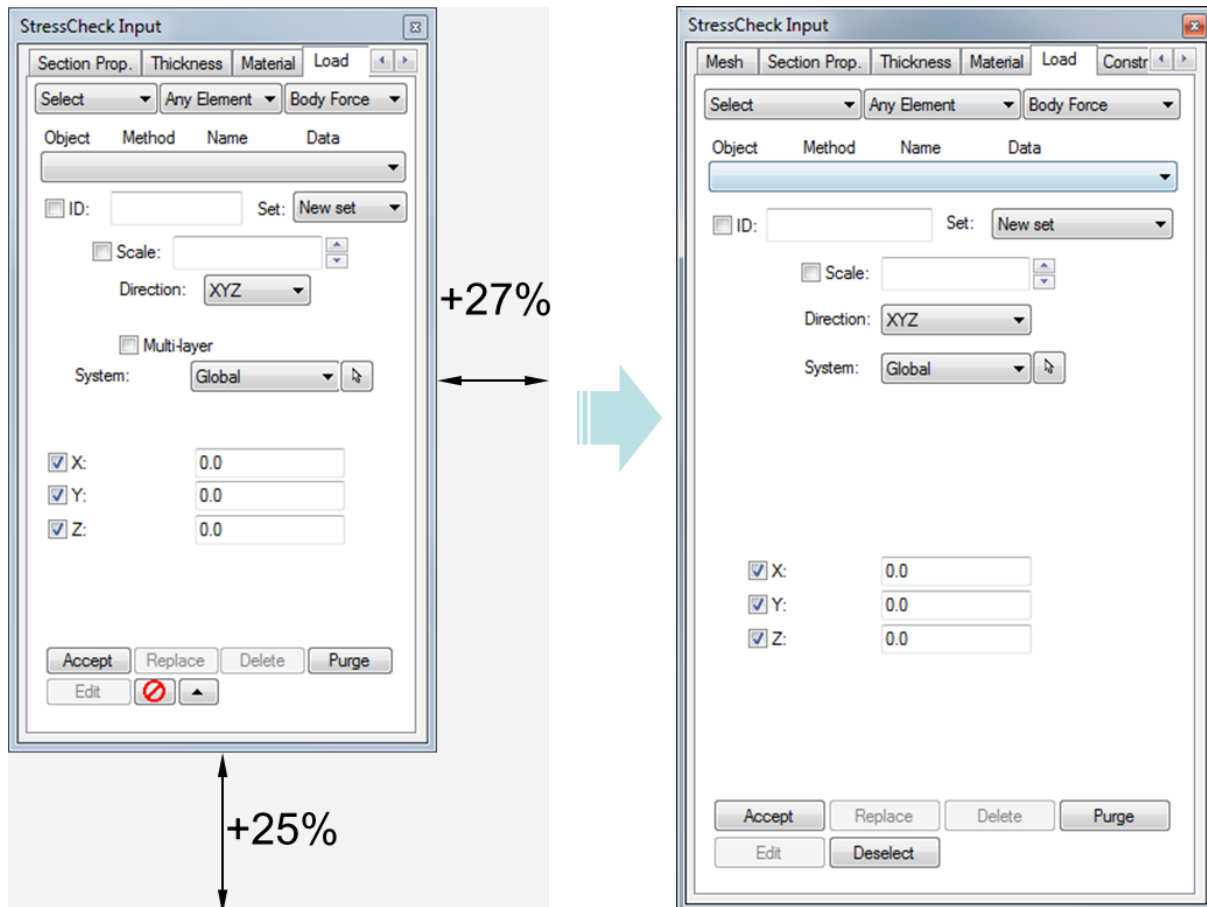


Figure 1: Size increase for the StressCheck Input dialog.

See *Section 3 Model Input* on PDF page 93 of the Master Guide for details.

SINGLE LOAD RECORD CHECK

Now resultants of applied loads can be checked for single load records. The new option “Any Record” (Figure 2) allows the user to select any existing load record or a combination of records to compute the resultants of the corresponding applied loads with respect to an arbitrary coordinate system and location in the selected system. Using this options brings up the Edit Load Cases Dialog (Figure 3) allowing the user to easily select individual or multiple load records. All the selected records are evaluated with respect to

the same selected system and moment center, and the resultants of the applied loads for each selected record are reported individually (i.e., not combined in a total resultant).

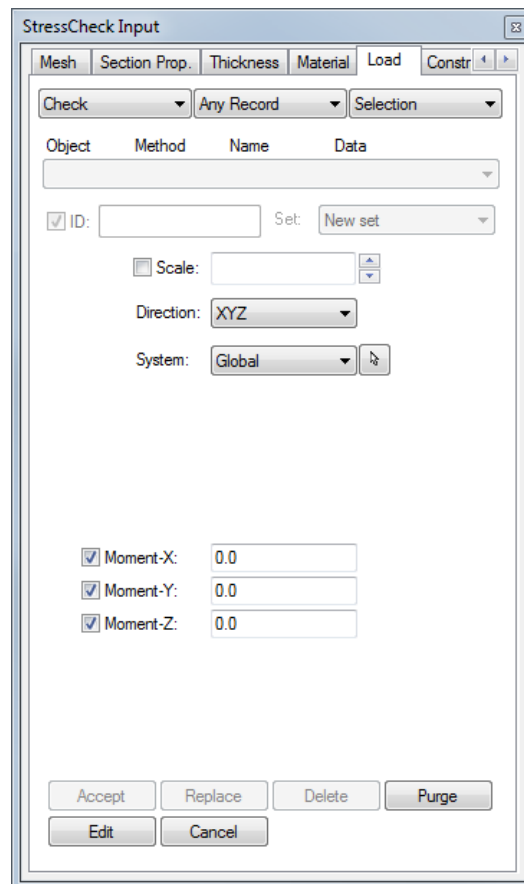


Figure 2: Any Record Check.

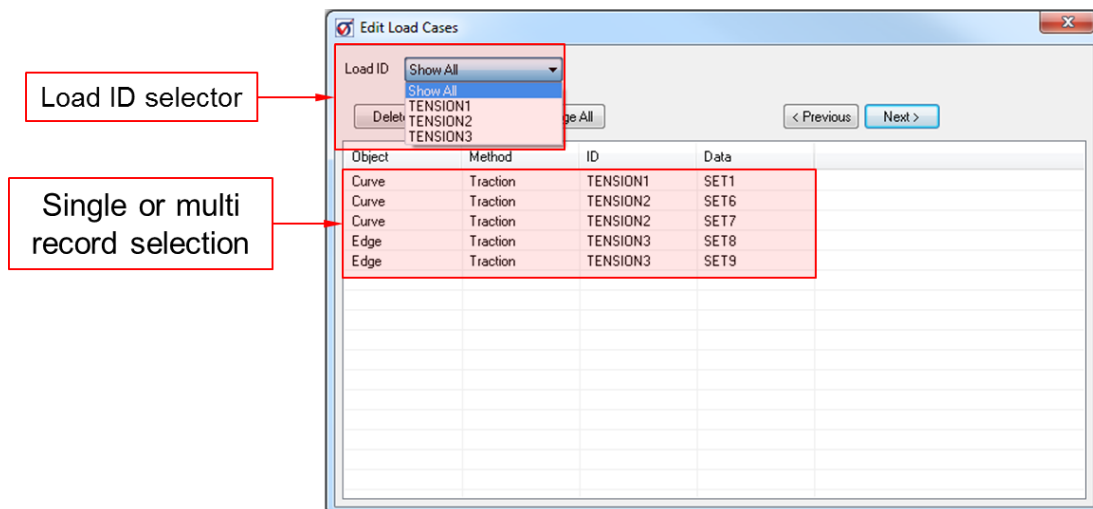


Figure 3: Record selection using the Edit Load Cases dialog.

For the special case of TLAP-Bearing load records, the load check table will report the resultants of the applied tractions at each hole and the corresponding input load components rotated and translated to the selected reference system and moment center (see Figure 4 as an example).

```

Load Summary
...
Load ID: LOAD, Elements=106, System=SYS4
Method: TLAPBearing, Object: PointLoad, Set: SET11
Local moment center: (1.000000e+000,5.000000e+000,8.000000e+000)
Global moment center: (-5.000000e+000,-9.000000e+000,7.500000e-001)
-----
      Fx          Fy          Fz          Mx          My          Mz
-----
  9.9045e-007   1.0000e+003  -9.9999e+002   1.3000e+004  -6.2499e+002  -6.2501e+002
-----

Reference Input Load: TLAP Bearing, Case ID: Case 1, #1
-----
      Fx          Fy          Fz          Mx          My          Mz
-----
  0.0000e+000   1.0000e+003  -1.0000e+003   1.3000e+004  -6.2500e+002  -6.2500e+002
-----
    
```

Figure 4: Example of Any Record check for a TLAP Bearing record.

See PDF page 114 of the Master Guide for details.

OPTION FOR AUTOMATIC GENERATION OF CONTACT CONSTANT

A new option has been added for the automatic generation of a contact constant based on the material properties assigned to each body in contact (see Figure 5).

To generate an automatic contact constant each selected contact pair has to belong to a body that has both a mesh and material assignments. The value is generated based on the following rules:

For US units (lbf/in³):

- $Kc = 0.1 * \min(E^i)$ if isotropic or
- $Kc = 0.1 * \min(E_{11}^i, E^j)$ if orthotropic or laminate materials

For SI units (n/mm³) use:

- $Kc = 0.004 * \min(E^i)$ if isotropic or
- $Kc = 0.004 * \min(E_{11}^i, E^j)$ if orthotropic or laminate materials

Where $\min(E^i)$ or $\min(E_{11}^i, E^j)$ is the minimum value of the elastic modulus (on the first direction for orthotropic or laminated materials) for all elements i, j associated to a given contact pair.

The steps for the creation of a contact constant constraint record have been updated to the following:

1. Enter constraint ID.

2. Select (pick) two contact zones, then either enter a Contact Constant value manually or click on Generate.
3. Clicking on Generate will place a value on the input field (if all conditions are satisfied). If a value or parameter was present in the input field, the automatically generated value will replace it.
4. Click Accept to create the record.

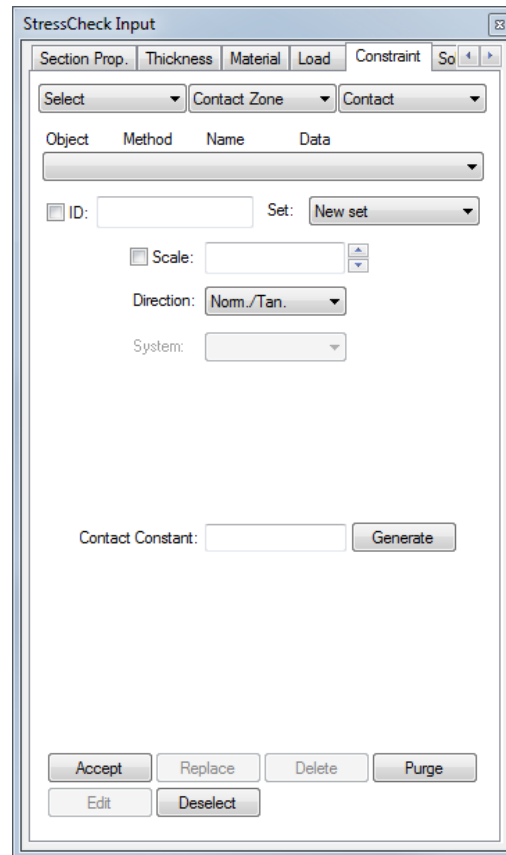


Figure 5: New option for the automatic generation of contact constants.

Remark: If no elements are associated with at least one of the selected contact zones, or no material properties have been assigned for at least one of the elements associated with the contact zones, the contact constant is not generated and an error is issued indicating the missing conditions. The automatic generation of the contact constant does not produce an associative relationship with the material properties. If the material properties referenced by the contact zones are changed, the generated value of an existing record will not be updated.

See PDF page 944 of the Master Guide for details.

EXTRACTIONS IN LOCAL SYSTEMS

For Points or Min/Max extractions performed using a local coordinate system, the location of the extraction points are now reported in both local and global coordinates. Local coordinates are identified based on selected system number (#) and system type:

- X(SYS#), Y(SYS#), Z(SYS#) for Cartesian.
- R(SYS#), T(SYS#), Z(SYS#) for Cylindrical/Polar.
- R(SYS#), T(SYS#), P(SYS#) for Spherical.

Global coordinates are given as Cartesian regardless of the system type selected and labeled as X(Global), Y(Global), Z(Global). Additional information about the location and rotation of the local system with respect to the global system is given in the Graph legend (see Figure 6 for an example).

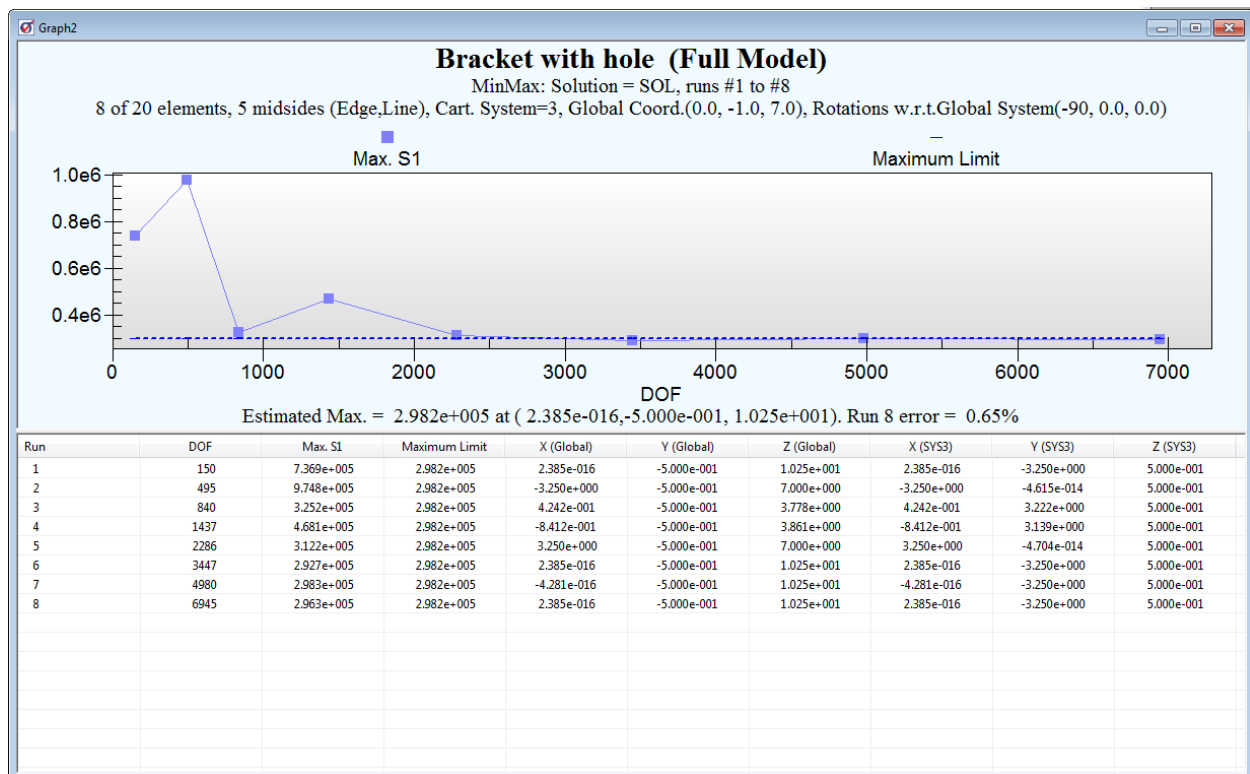


Figure 6: Extractions with local systems.

See PDF page 196 of the Master Guide for details.

NEW OPTION TO INPUT TRACTION LOADS

A new option to use force components as input for traction loads has been implemented (see Figure 7). The force input is internally converted to traction loads by dividing the input load by the area of the element faces resulting from the selected Objects (Face, Face Surface and Any Surface). This option is only available for 3D Elasticity.

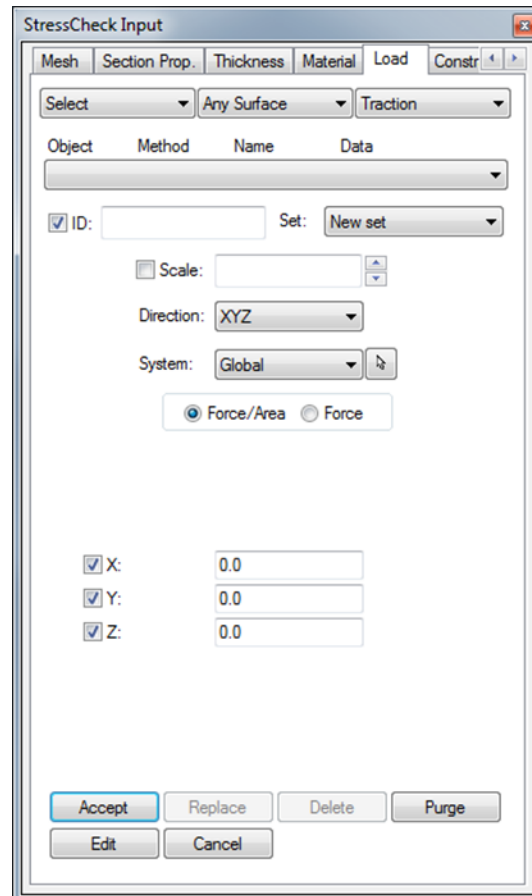


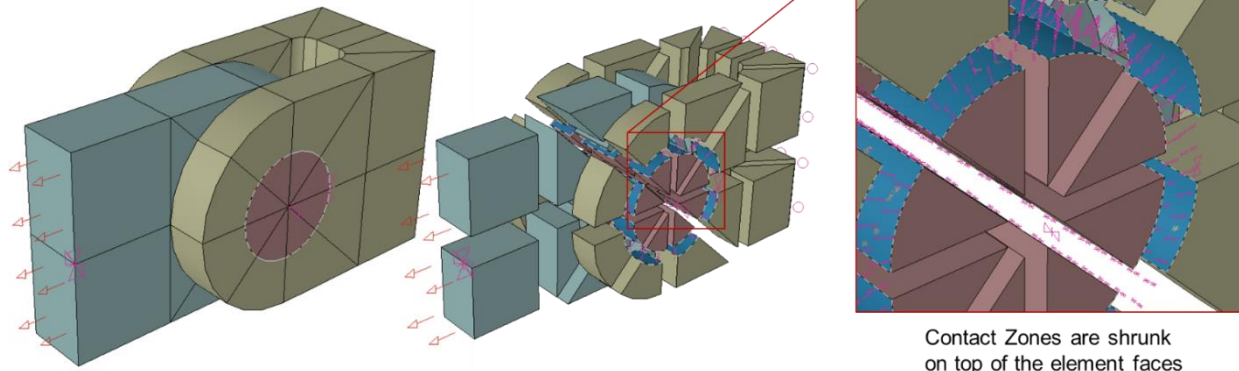
Figure 7: Force load input option for traction loads

See PDF page 583 of the Master Guide for details.

CONTACT ZONES SHRINKING

Contact zones can now be shrank together with elements to simplify selection. This feature can be particularly useful during constraint assignments and when checking resultants for load transfer across a contact zone.

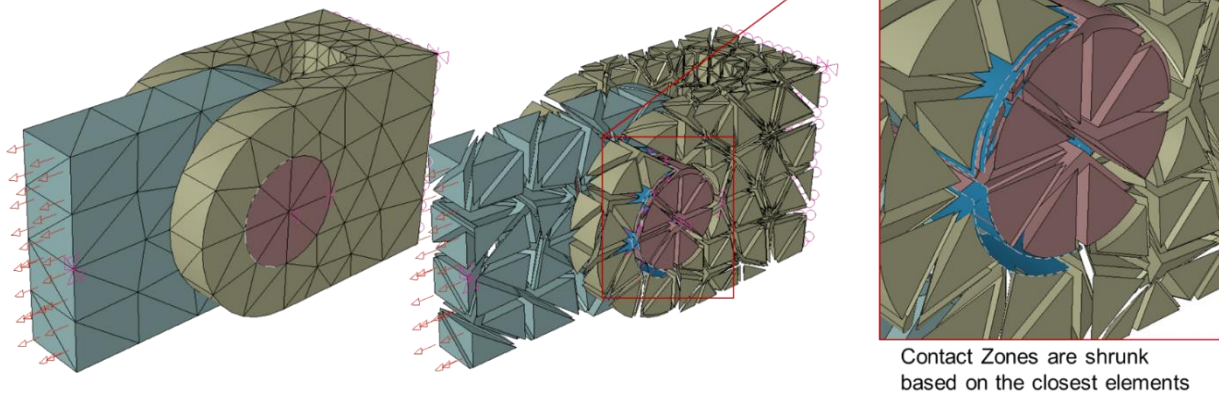
Face based contact zones



Contact Zones are shrunk on top of the element faces

Figure 8: Shrink of face based contact zones.

Surface based contact zones



Contact Zones are shrunk based on the closest elements

Figure 9: Shrink of surface based contact zones.

NEW TLAP BEARING OPTION

The TLAP Bearing method has been expanded to provide the new “IMO” (Ignore Moments and Offsets) option (see Figure 10). This option can be used to generate a bearing load record ignoring the contribution of the original point load moment components and the moments caused from the offset distance between the location of the point load and the center of the hole. The original TLAP Bearing option, now labeled “Default”, generates a statically equivalent bearing load.

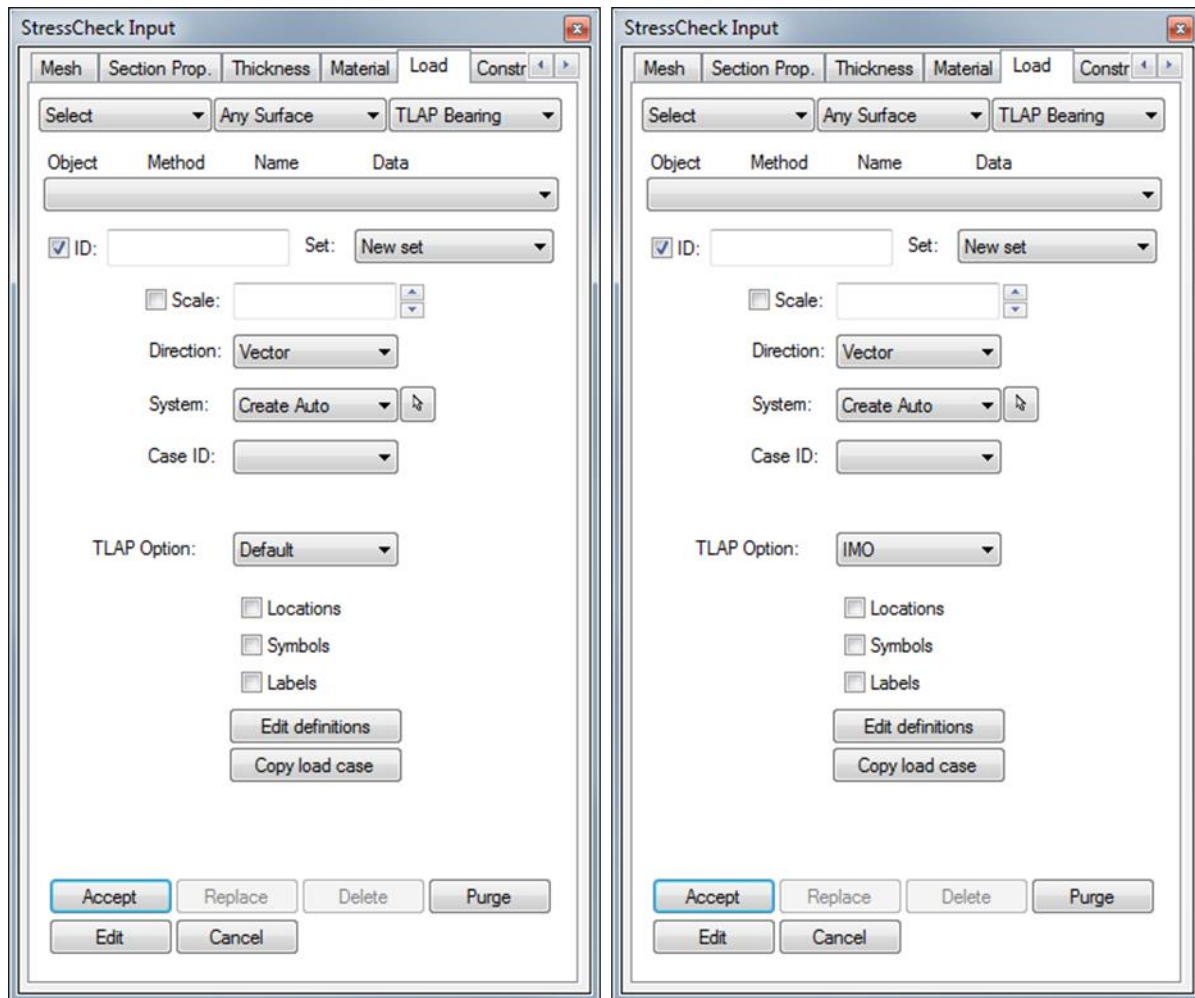


Figure 10: New IMO TLAP Bearing option.

Older files with existing TLAP bearing records will be displayed as Default when retrieved.

See PDF page 586 of the Master Guide for details.

IMPORT AND EXPORT PARAMETERS AND FORMULAE

The import and export process for parameters has been expanded to include formulas. Users now have the ability to import and export parameters and formulas using the Import and Export options provided in the File Menu and readily accessible in the Main Toolbar buttons, selecting the Parameter File (*.par) file type.



Figure 11: Import/Export buttons

See PDF page 37 of the Master Guide for details.

TOTAL DISPLACEMENT

The total displacement $|U|$ has been added to the list of extraction functions and it is available for Plot, Points, and Min/Max extractions. By definition, $|U| = (U_x^2 + U_y^2 + U_z^2)^{1/2}$.

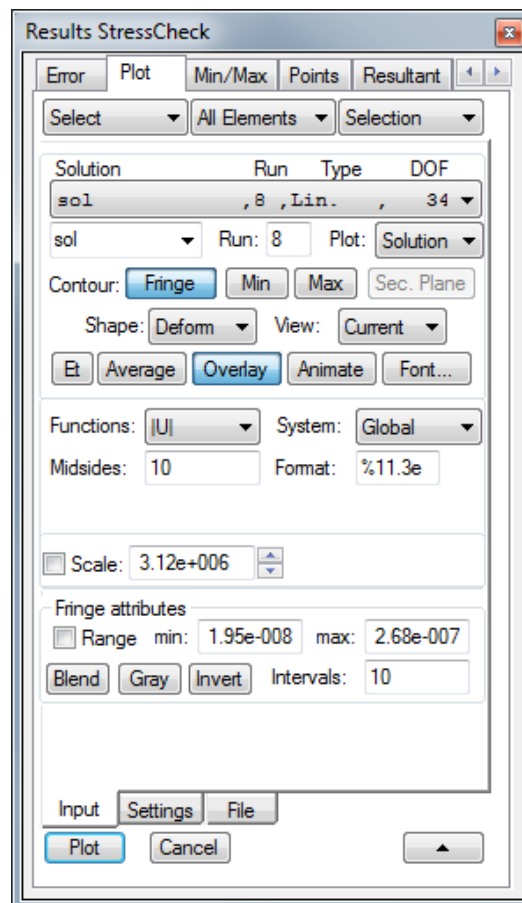


Figure 12: Total displacement extraction function $|U|$.

INCLUDE CONTACT ZONES IN COPY OPERATION

Contact zones have been added to the list of objects included on a copy operation.

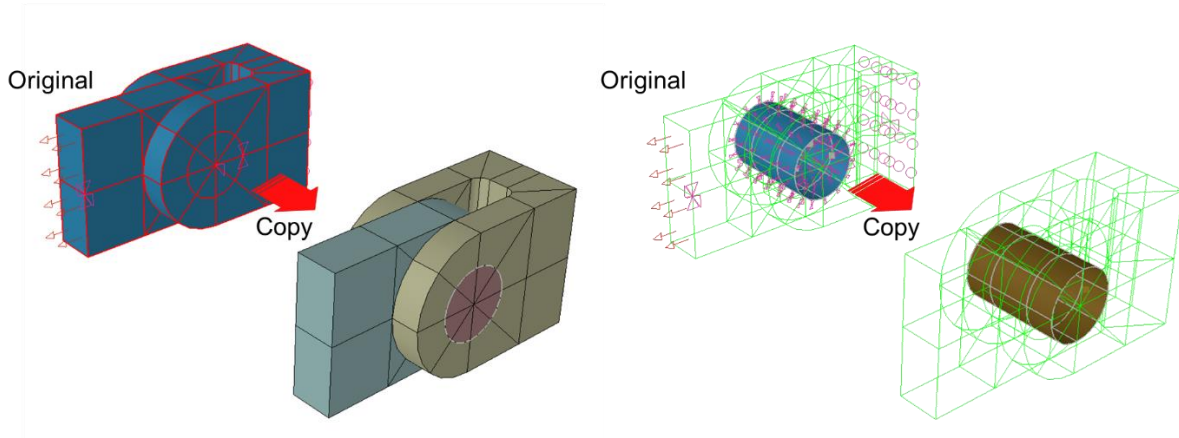


Figure 13: Copy operation on models with contact zones.

Note that BC's are not included on a Copy operation.

UPDATED BEARING LOAD ERROR MESSAGES

Bearing and TLAP bearing load error messages have been updated to provide more clear information to the user and to distinguish between Circularity and Tapper errors.

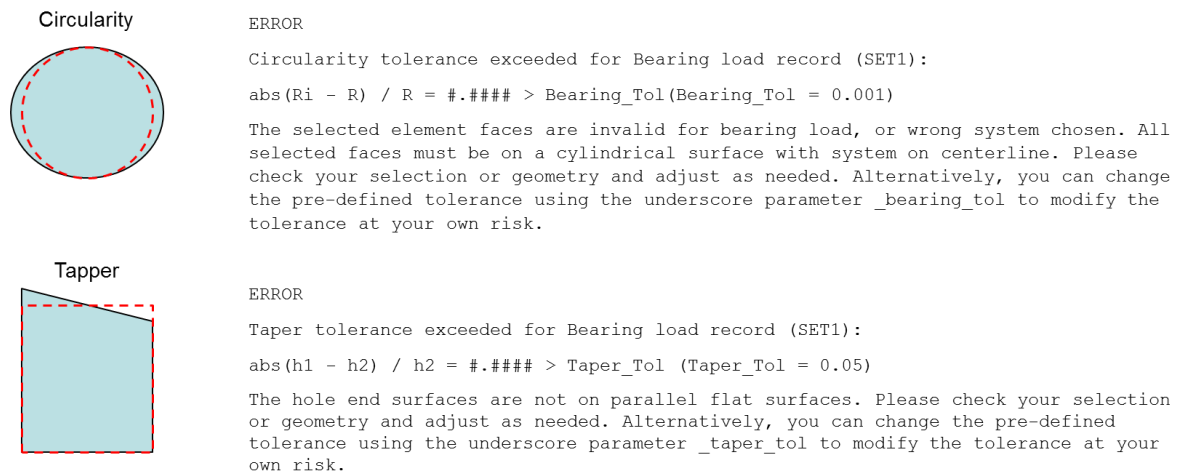


Figure 14: Updated Bearing Load Error Messages.

Note that if the underscore parameter is used, the tolerance will be updated to the new user input value.

See PDF page 585 of the Master Guide for details.

TAB CLASS MEMORY

In this new release inputs and selections are retained when switching back and forth between tabs on a given UI (Input, Solver, and Result window tabs):

- While within the same UI, the selections for each Tab are retained, but if the UI is dismissed and opened again then the Tab selections are reset to its defaults.
- Only selections within a tab are retrieved. Returning to a tab does not recall screen selections or any other settings.
- Only applicable for settings during the current StressCheck session.
- Tab selections are not retained when saving a StressCheck file or opening a previously saved file. When starting a new session of StressCheck, all tab options are set to their defaults.

LIST OF RESOLVED ISSUES AND MINOR IMPROVEMENTS

Topic	Description
AOM Help	Several typos fixed Refreshing and update AOM display when switching from Solver to Results and Formula to Laminate Info selections
Autolamination	Deletion of plies in Lam Stack window (two plies being deleted instead of one) Incorrect solution for some stack lumping options
Automesh	Deleting nodes after deleting an Automesh on imported geometry Marquee pick for creation of Automesh records Automesh crashes for particular cases Issues when changing and adding parameters generated an invalid mesh Issues with models with Automesh and Handmesh elements with h-discretization
COM	Delete selection method of Model Item is now working. Setting a parameter to an invalid value via COM now reverts parameter back to last valid value Laminate stacks can now be added through COM Selection not working after loading SCP model Number of SplineCurves was being reported incorrectly for some cases
Constraints	Point constraint set name was not referenced Nodal constraint can now be applied to a corner node in Axisymmetric theory Case where Constraints were applied to the wrong child element faces
Cutting Plane	Fixed condition of a Cutting Plane appearing empty when a Contour Plot was being displayed Fringe plot on Cutting Plane was showing undesired gray color
Display	Free Edge display no longer disappears when using zoom in/out Fixed instance where Crack Faces were not registered as wet surfaces. Fixed condition in which marquee pick did not work to create quadrilateral elements Fixed Blanking after Invert Selection Fixed inverted Fringe Plot colors out of order Invert selection was not working for some contact models. Prevent crash when using invalid Display format Fixed case where nodes could not be unblanked Fixed issue with the display of bending and twisting moments for shells being swapped An error is issued if more than two contact zones are selected with marquee pick when creating a contact constraint Creating new elements does not longer causes loads to show if loads display is off
Error/Warning Messages	Collapsed repeated error messages pop-up when applying ill-defined formula. An error is now issues when floating symmetry is not used in the correct context

Extractions	Fixed crashing conditions during Min/Max extraction
	Failing extraction on Face Set failing for Points or Min/Max is now fixed
	Case where Max value was returned as 0 for all runs in Min/Max extraction when Min and Max were both selected
	Failing Points extraction on nodes with parent mesh displayed
	Fixed case when new local systems created after solving where not available for extractions selector
	Column header for mechanical strain was missing for some strain components
	SIF extraction failing at boundary between adjacent elements
File I/O	Indefinite expansion of crash recovery file caused StressCheck to open very slowly
	Condensed errors when too many errors are given while importing a specific NASTRAN file (genset.dat)
	StressCheck v10.2: Point at (0,0,0) cannot be deleted for imported CATIA v5
Formulae	Fixed crash condition when using formula for extraction
	Intrinsic atan2(y;x) formula results were off by 90 degrees
	Expression with atan2(a,b) was evaluated with inputs reversed
Geometry	Fixed issues with changing parameters after a Boolean union and on a body with a copy child
GUI	Replaced Cancel icon for Deselect button.
	The tab order for the StressCheck Input UI has been fixed to follow the typical left to right, top to bottom order.
H-Discretization	H-discretization record not being fully displayed in drop down list
	Particular cases where h-discretization caused applied loading to change
	Fixed crash condition when using h-discretization
	A warning is issued when trying to Automesh a geometry with h-discretization
ITP	Case where h-discretization generated a wrong answer
	Incomplete fringe plot using calculator function in incremental plasticity model
	Fixed crash condition when showing details on nonlinear events
Parameters	Fixed condition in which one could not switch back to a previously good set of parameters
	Fixed a case where parameter changes created an invalid model
	Commas are now allowed in parameter expressions when importing *.par files
Sets	A recently created/modified Set is no longer deleted when replacing a material assignment record
	New sets are now added to Results window Sets dropdowns
Solver	Fixed crashing condition for the Batch Solver
	Fixed conditions in which could not use Esc to stop a solving process
	Fixed a condition in which the solver would stop without issuing an error
SRS/BRS	An error is now issued when an SRS load creation fails
	Fixed condition in which a SRS projection to element points could fail at tangency boundaries

TLAP	Copy Load Case does not allow the use of repeated Case IDs TLAP assigned to surfaces failed to apply several points in model with local mesh refinement Fixed crash when entering illegal Point Load data on the Case Definitions UI
------	--