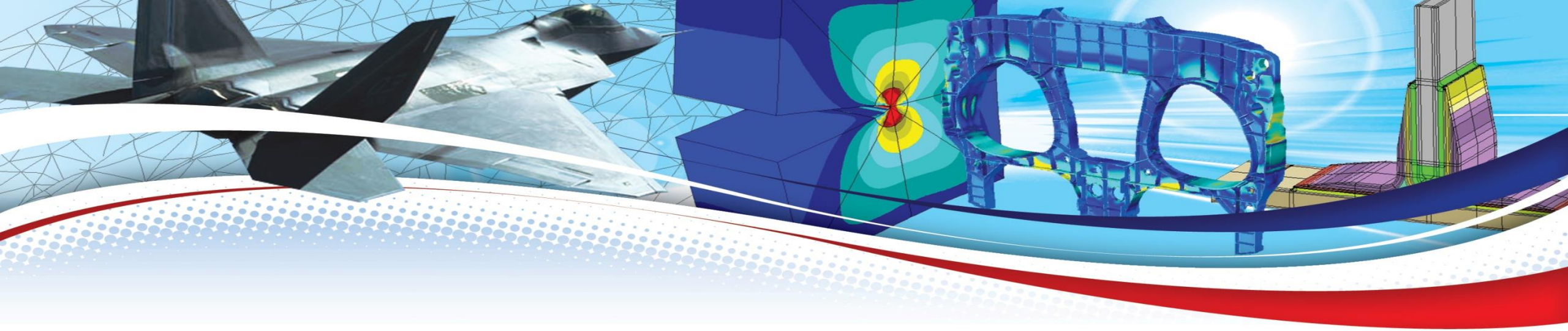
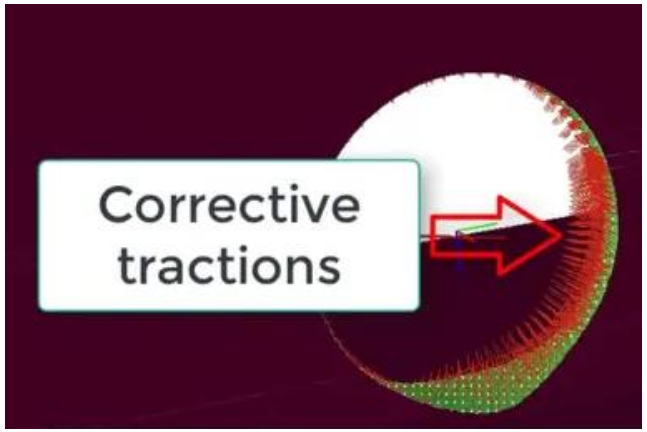
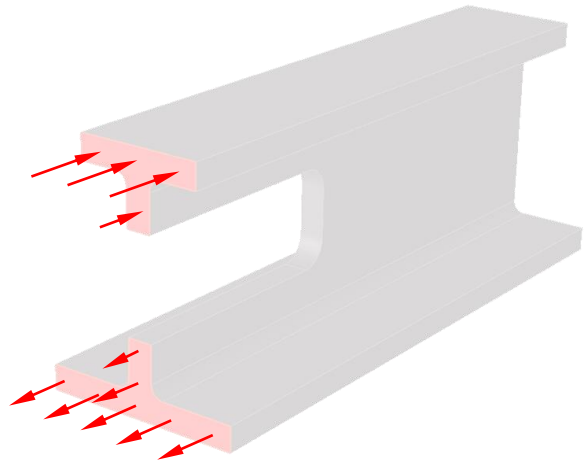


New Features and Enhancements in StressCheck 10.5



ESRD, Inc.

May 2019



esrd

StressCheck®

New Features and Enhancements in SC 10.5

STRESSCHECK

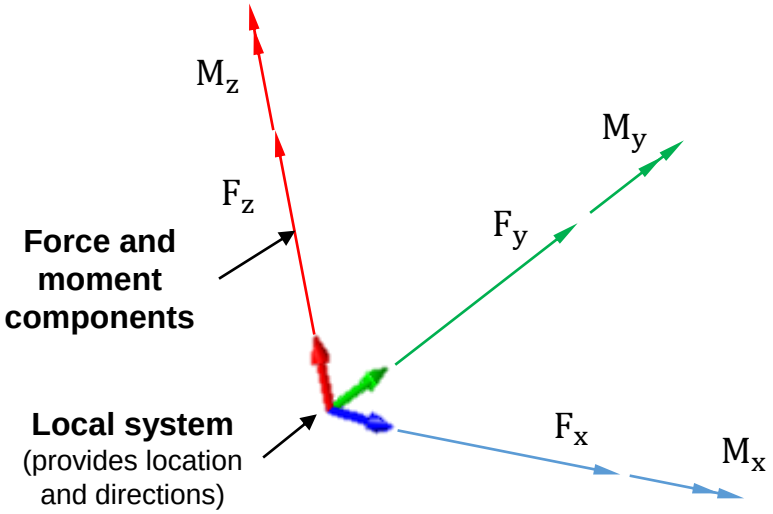
- New Force/moment method
- New TLAP Cross section option
- TLAP Bearing and Bearing loads correction
- New Any Body option for Assignment
- Plastic Strain extractions
- Log Improvements for Incremental Solutions
- Name sorting for ITP solutions
- Face/Face Surface element blank
- Curves resolution can now be controlled
- File Build number is now recorded in the Project Log
- Long solution names are now wrapped on the Plot legend
- Automatic And Manual Selection Of Graphic Drivers

NEW FORCE/MOMENT LOAD METHOD

New Force/Moment Option

- **Force/Moment:** The directional components of the force and moment vectors are converted into a statically equivalent linear traction distribution, applied over the selected element faces.

Input type: Manual

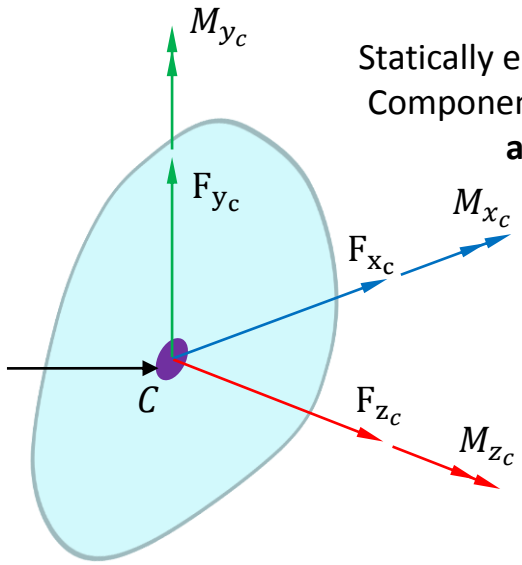


Define Force and Moment Components with respect to a selected local system

Assignment

Centroidal System aligned with principal directions (created internally)

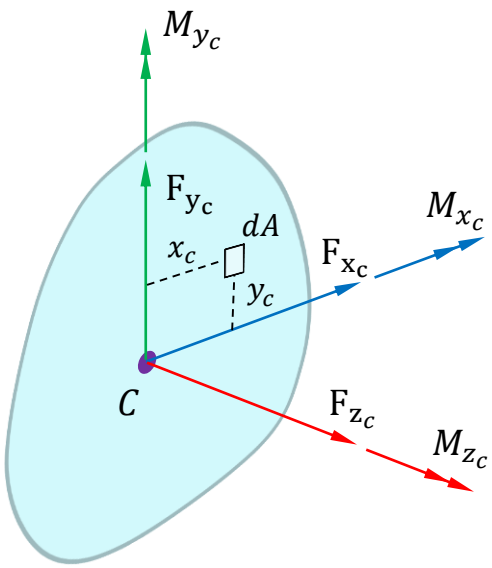
Assignment on selected Cross Section



Statically equivalent Force and Moment Components for cross section centroid **along principal axis**.

Load assignment

- The linear traction distribution will be computed along the orthogonal principal axis through the centroid of the section, based on the following definitions:



$$F_{x_c} = \iint T_{x_c}\, dA \qquad M_{x_c} = \iint T_{z_c}\, y_c\, dA$$

$$F_{y_c} = \iint T_{y_c}\, dA \qquad M_{y_c} = -\iint T_{z_c}\, x_c\, dA$$

$$F_{z_c} = \iint T_{z_c}\, dA \qquad M_{z_c} = \iint T_{y_c}\, x_c\, dA - \iint T_{x_c}\, y_c\, dA$$

⚠ CAUTION

The shear components of the linear traction distributions generated by the Force/Moment option do not satisfy the stress free boundary conditions for general cross sectional shapes. Extractions near the area of load application are discouraged.

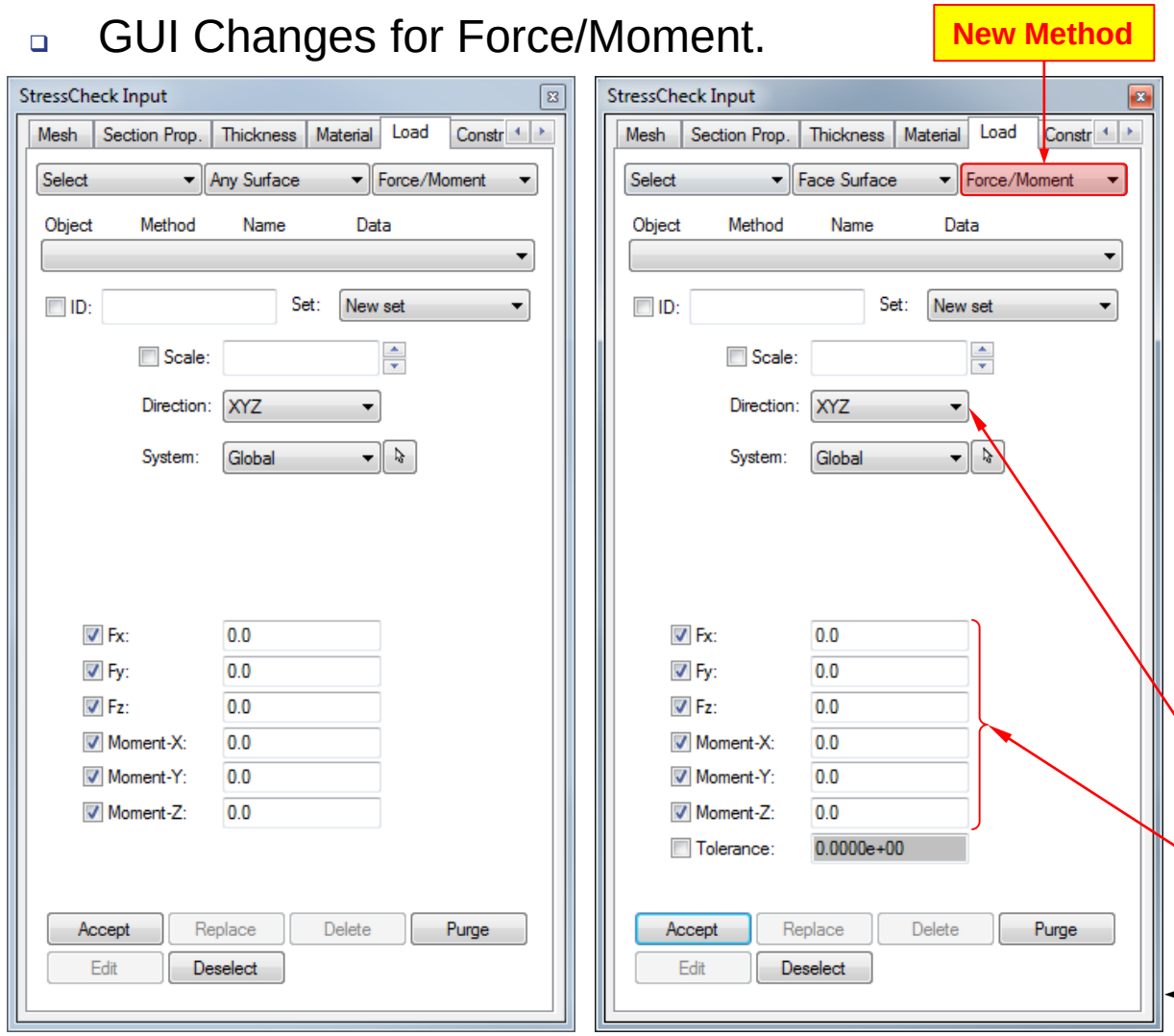
$$T_{x_c} = \frac{F_{x_c}}{A} - \frac{1}{2}\frac{M_{z_c}}{I_{x_c}} y_c$$

$$T_{y_c} = \frac{F_{y_c}}{A} + \frac{1}{2}\frac{M_{z_c}}{I_{y_c}} x_c$$

$$T_{z_c} = \frac{F_{z_c}}{A} - \frac{M_{y_c}}{I_{y_c}} x_c + \frac{M_{x_c}}{I_{x_c}} y_c$$

Force/Moment Option

- GUI Changes for Force/Moment.



- The selection needs to satisfy a coplanar condition. If the this condition is not satisfied an error issued and the record creation is prevented:

```
ERROR
      Load error found in surfaces/elements = # and #.
      Force/Moment load applied to faces that are not coplanar.
      Tolerance may be controlled with parameter _COPLANAR_TOL
      (default=0.0001, use a larger value to loosen the
      tolerance).
```

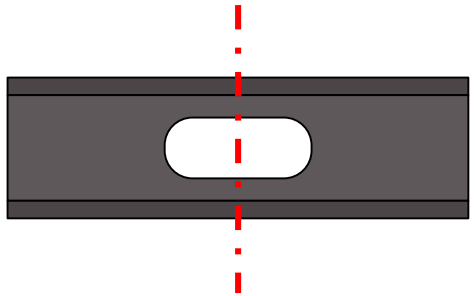
- While continuity is not a requirement (multiple selections are allowed) this assignment requires checking a flatness tolerance for the selection (similar as it is done for symmetry or anti-symmetry). If the flatness check does not pass an error is issued and the record is not created:

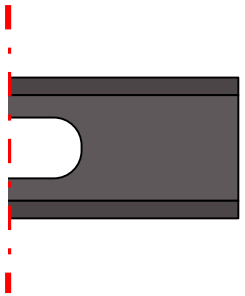
```
ERROR
      Load error found in surface/element = #. Force/Moment
      load applied to a surface/face which is not flat.
      Tolerance may be controlled with parameter _SURFACE_TOL
      (default=0.001 deg.).
      Number of points checked on each surface/face controlled
      with parameter _SURFACE_MIDPT (default=2). Select
      surface/face set "_FAIL_LOAD" to see problem
      surfaces/faces.
```

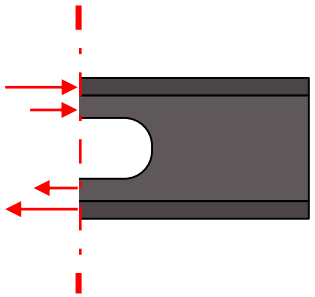
- Only Direction: XYZ is allowed, if a non-Cartesian system is selected an error is issued and the record is not created.
- Only numeric or parametric input are to be allowed, with exception of formulae used as parametric expression (i.e., with the pipe "|" symbol preceding the formula name).

← Shown for "Face Surface" but also available for "Face" Object

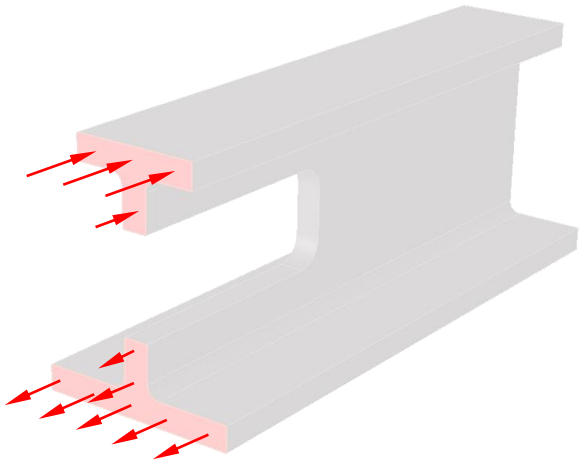
Sample Selection: Use Case 1



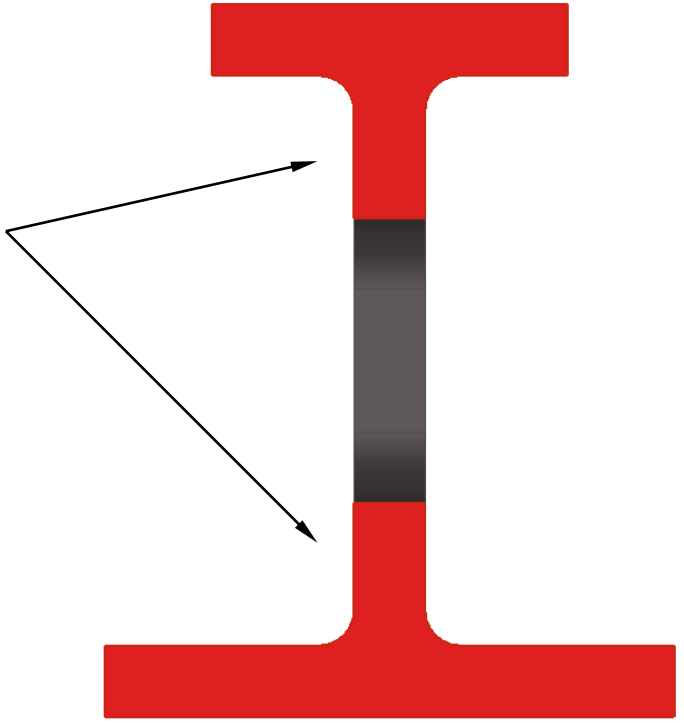




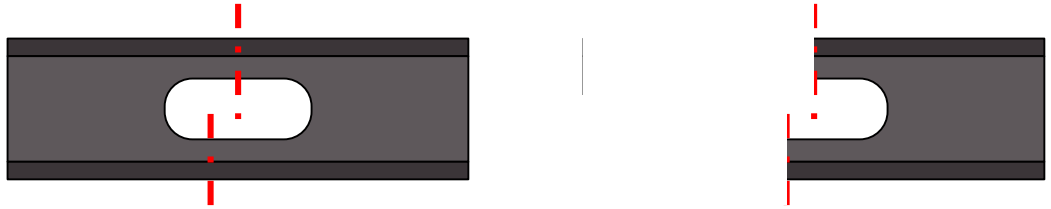
Multiple coplanar selections are allowed



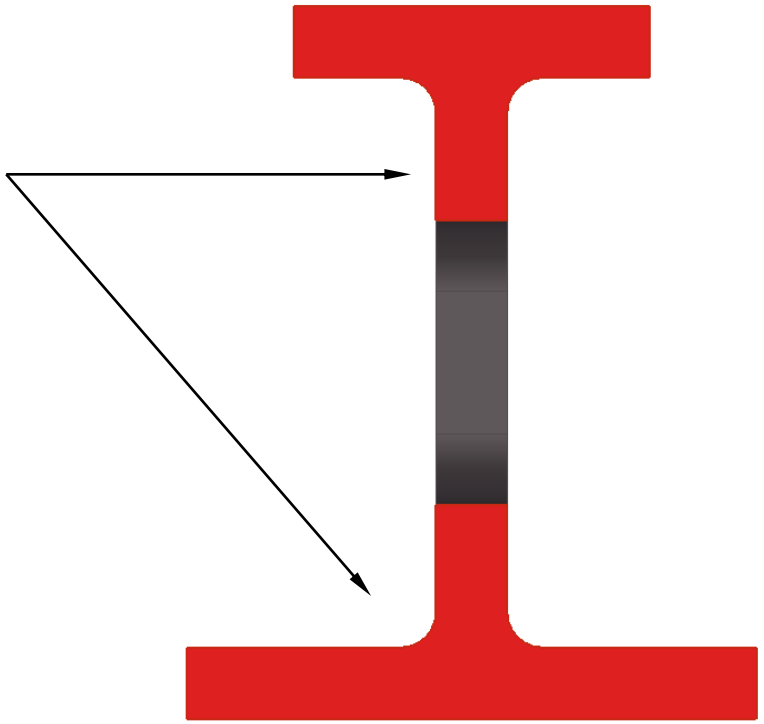
Cross Section



Sample Selection: Use Case 2



Cross Section



Selections are flat but not coplanar, an error is issued when pressing “Accept”

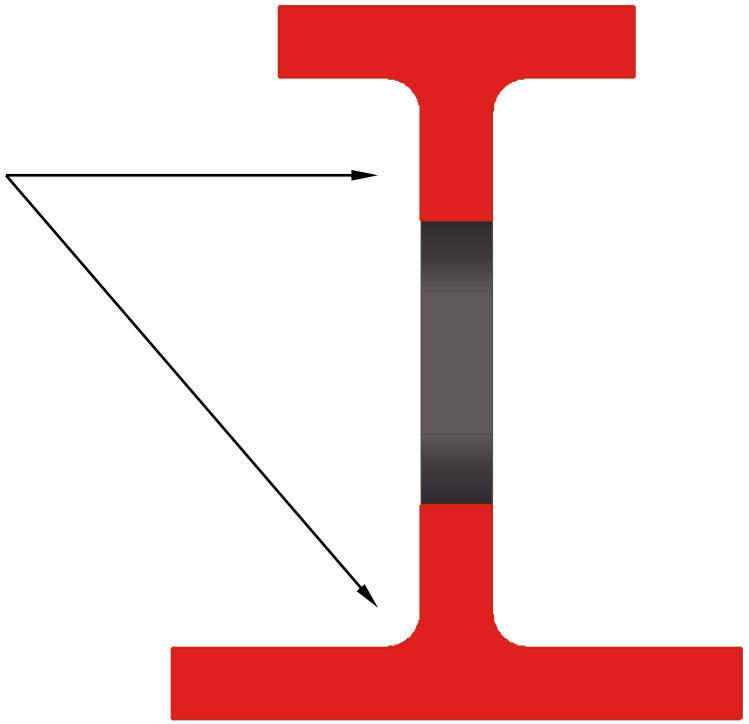
```
ERROR
      Load error found in elements = # and #, case = LOAD_ID.
      Force/Moment load applied to faces that are not coplanar.
      Tolerance may be controlled with parameter _COPLANAR_TOL
      (default=0.0001, use a larger value to loosen the tolerance).


ERROR
      Load error found in surfaces # and #.  Force/Moment load
      applied to surfaces that are not coplanar.  Tolerance may be
      controlled with parameter _COPLANAR_TOL(default = 0.0001, use a
      larger value to loosen the tolerance).
```

Sample Selection: Use Case 3

Selections are not flat, an error is issued when pressing “Accept”

```
ERROR
      Load error found in surface = #.  Force/Moment load applied to a
      surface which is not flat.  Tolerance may be controlled with
      parameter _SURFACE_TOL (default = 0.001 deg.).  Number of points
      checked on each surface controlled with parameter
      _SURFACE_MIDPT(default = 2).Select surface set "_FAIL_LOAD" to
      see problem surfaces.

ERROR
      Load error found in element = #. Force/Moment load applied to a
      surface which is not flat.  Tolerance may be controlled with
      parameter _SURFACE_TOL (default=0.001 deg.).
      Number of points checked on each face controlled with parameter
      _SURFACE_MIDPT (default=2). Select surface set "_FAIL_LOAD" to
      see problem surfaces.
```

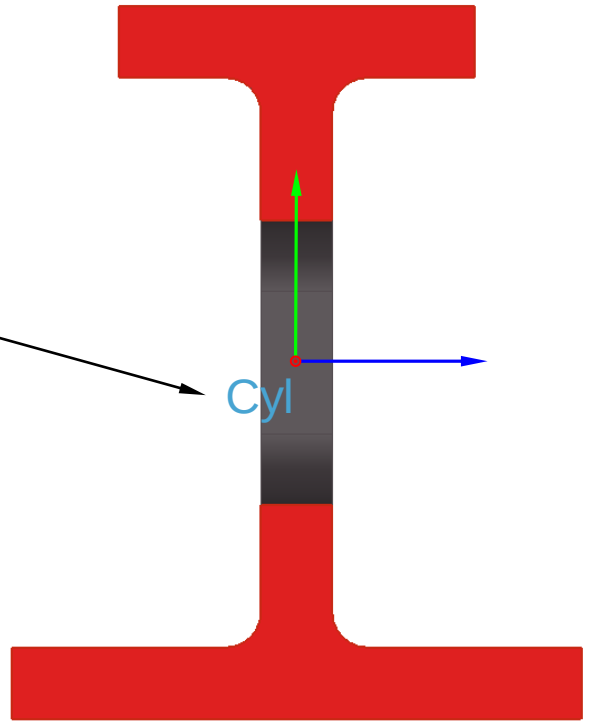
Cross Section



Sample Selection: Use Case 4

Cylindrical system selected

Cross Section



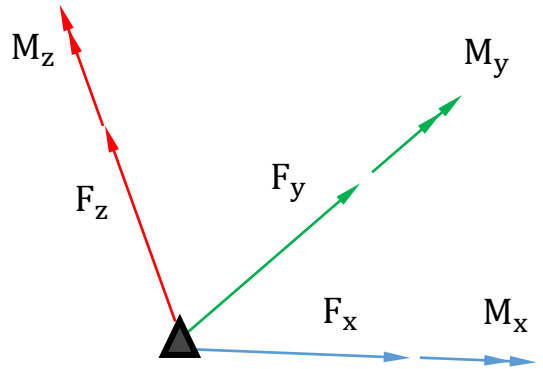

```
ERROR
   Force/Moment method requires a Cartesian system.
```

NEW TLAP CROSS SECTION OPTION

New TLAP Cross Section Option

- **TLAP-Traction (Cross Section)**: The TLAP force and moment components are converted into a statically equivalent linear traction distribution, applied over the selected element faces.

Input type: TLAP Point

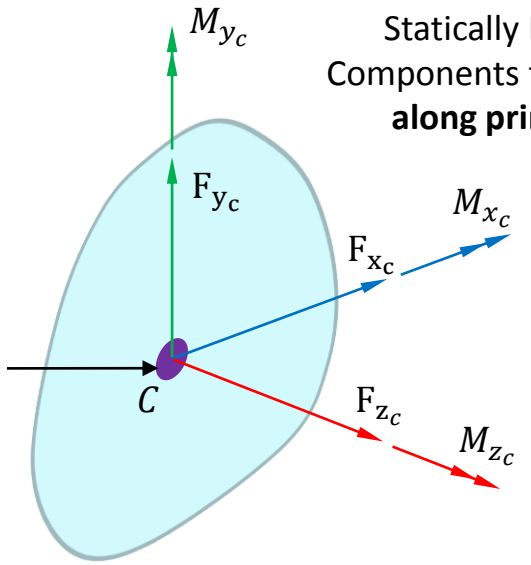


Force and Moment Components for TLAP
(before rotation and/or translation)

Assignment

Centroidal System aligned with principal directions (created internally)

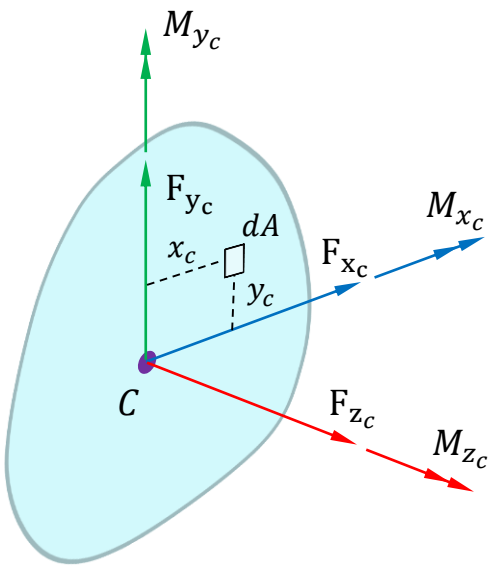
Assignment on selected Cross section



Statically Equivalent Force and Moment Components for TLAP on cross section centroid **along principal axis** (after rotation and translation)

Load assignment

- The linear traction distribution will be computed along the orthogonal principal axis through the centroid of the section, based on the following definitions:



$$F_{x_c} = \iint T_{x_c}\, dA \qquad M_{x_c} = \iint T_{z_c}\, y_c\, dA$$

$$F_{y_c} = \iint T_{y_c}\, dA \qquad M_{y_c} = -\iint T_{z_c}\, x_c\, dA$$

$$F_{z_c} = \iint T_{z_c}\, dA \qquad M_{z_c} = \iint T_{y_c}\, x_c\, dA - \iint T_{x_c}\, y_c\, dA$$

CAUTION

The shear components of the linear traction distributions generated by the TLAP Cross Section option do not satisfy the stress free boundary conditions for general cross sectional shapes. Extractions near the area of load application are discouraged.

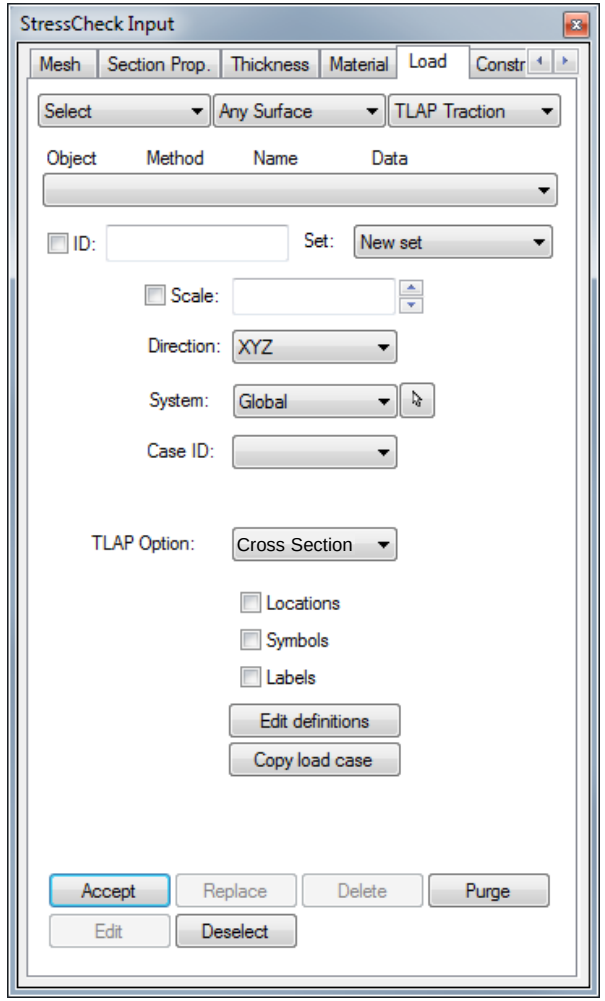
$$T_{x_c} = \frac{F_{x_c}}{A} - \frac{1}{2}\frac{M_{z_c}}{I_{x_c}} y_c$$

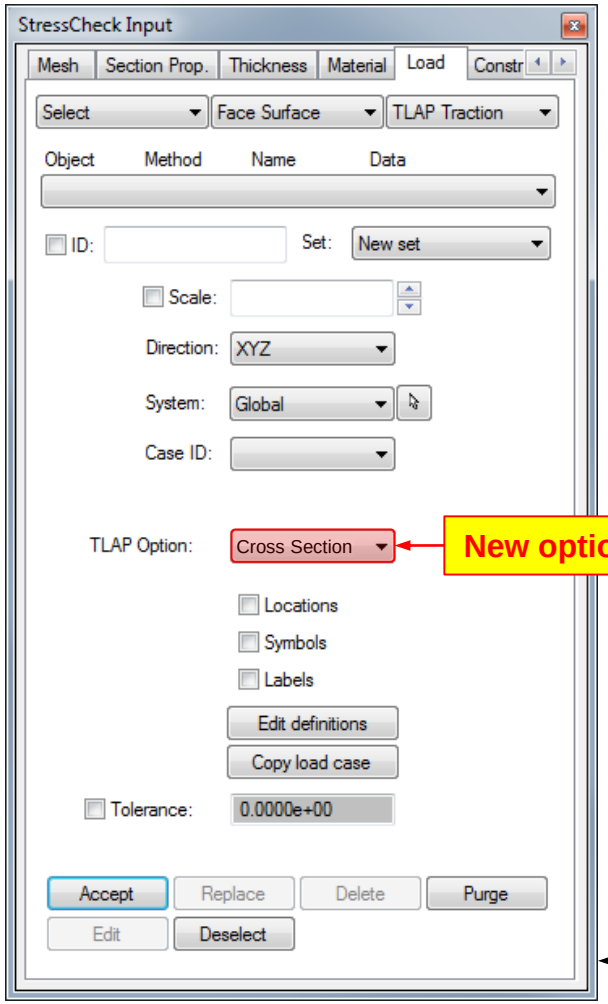
$$T_{y_c} = \frac{F_{y_c}}{A} + \frac{1}{2}\frac{M_{z_c}}{I_{y_c}} x_c$$

$$T_{z_c} = \frac{F_{z_c}}{A} - \frac{M_{y_c}}{I_{y_c}} x_c + \frac{M_{x_c}}{I_{x_c}} y_c$$

TLAP Cross Section Option

- GUI Changes for TLAP Cross Section.





← Shown for “Face Surface” but also available for “Face” Object

- The selection needs to satisfy a coplanar condition. If the this condition is not satisfied an error is issued and the record creation is prevented:

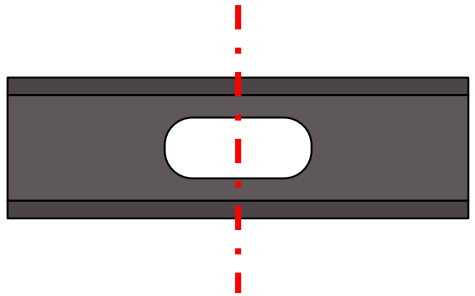
```
ERROR
      Load error found in surfaces/elements = # and #. TLAP
      Cross Section load applied to faces that are not coplanar.
      Tolerance may be controlled with parameter _COPLANAR_TOL
      (default=0.0001, use a larger value to loosen the
      tolerance).
```

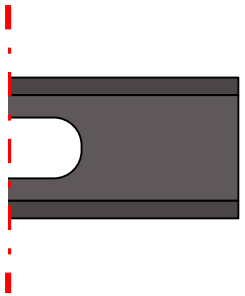
- While continuity is not a requirement (multiple selections are allowed) this assignment requires checking a flatness tolerance for the selection (similar as it is done for symmetry or anti-symmetry). If the flatness check does not pass an error is issued and the record is not created:

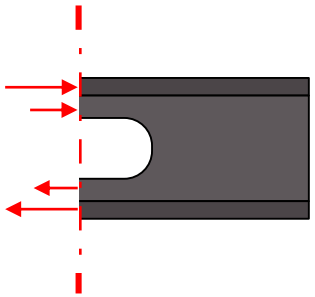
```
ERROR
      Load error found in surface/element = #. TLAP Cross
      Section load applied to a surface/face which is not
      flat.
      Tolerance may be controlled with parameter _SURFACE_TOL
      (default=0.001 deg.).
      Number of points checked on each surface/face controlled
      with parameter _SURFACE_MIDPT (default=2). Select face
      set "_FAIL_LOAD" to see problem surfaces/faces.
```

- Same as with other TLAP Traction options multiple load selections are allowed.

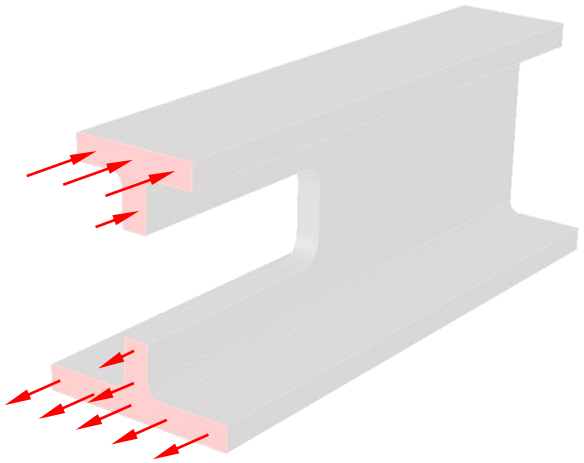
Sample Selection: Use Case 1



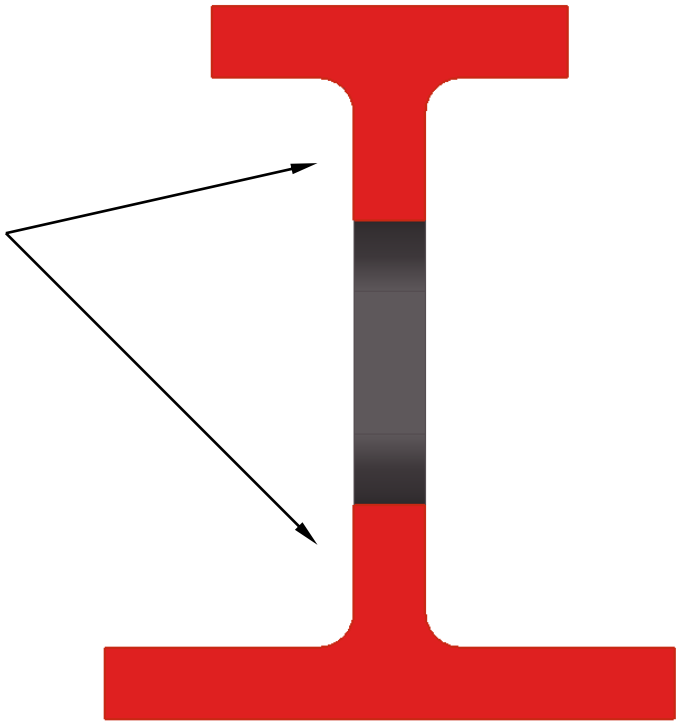




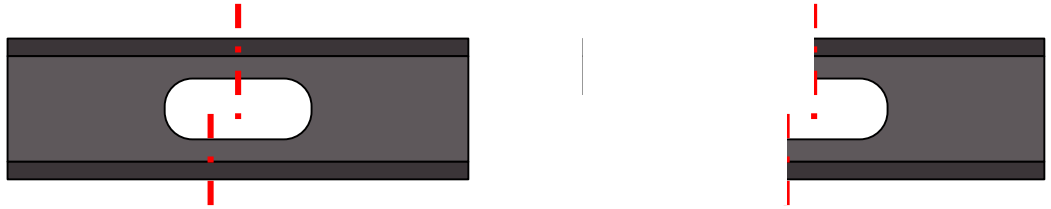
Multiple coplanar selections are allowed



Cross Section



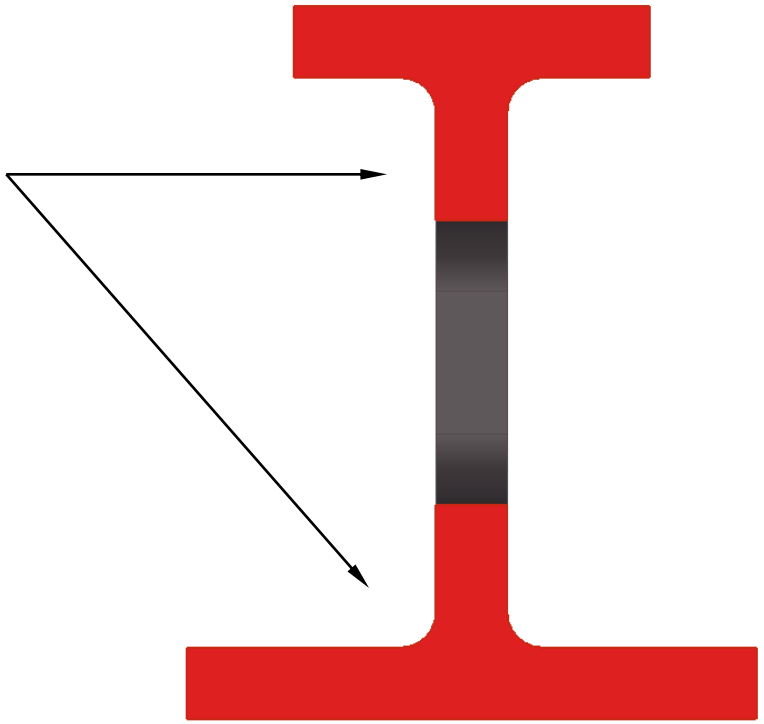
Sample Selection: Use Case 2



Selections are flat but not coplanar, an error is issued when pressing “Accept”

```
ERROR
      Load error found in elements = # and #, case = LOAD_ID. TLAP
      Cross Section load applied to faces that are not coplanar.
      Tolerance may be controlled with parameter _COPLANAR_TOL
      (default=0.0001, use a larger value to loosen the tolerance).


ERROR
      Load error found in surfaces # and #. TLAP Cross Section load
      applied to surfaces that are not coplanar.  Tolerance may be
      controlled with parameter _COPLANAR_TOL(default = 0.0001, use a
      larger value to loosen the tolerance).
```

Cross Section

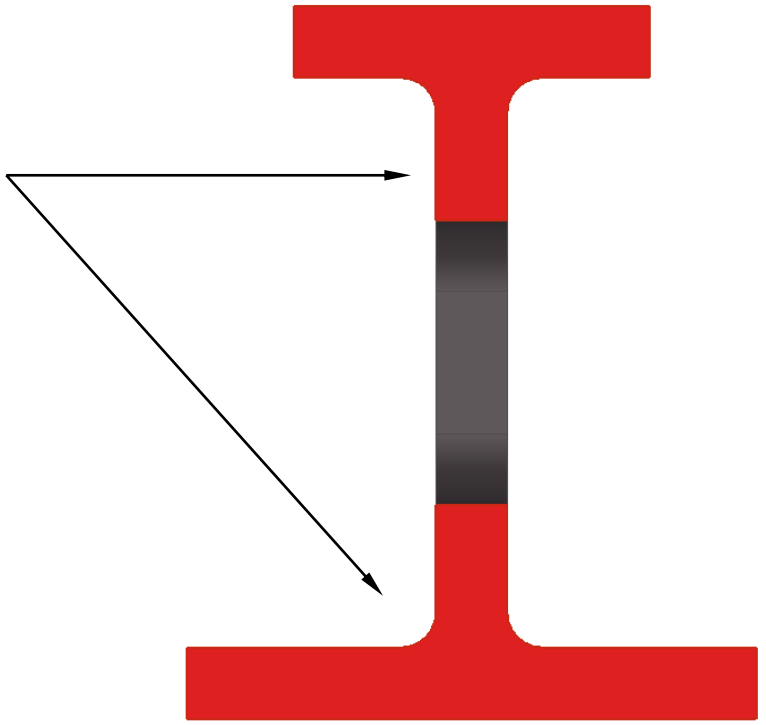


Sample Selection: Use Case 3

Selections are not flat and not coplanar, an error is issued when pressing “Accept”

```
ERROR
      Load error found in surface = #. TLAP Cross Section load applied
      to a surface which is not flat.  Tolerance may be controlled
      with parameter _SURFACE_TOL (default = 0.001 deg.).  Number of
      points checked on each surface controlled with parameter
      _SURFACE_MIDPT(default = 2).Select surface set "_FAIL_LOAD" to
      see problem surfaces.

ERROR
      Load error found in element = #. TLAP Cross Section load applied
      to a surface which is not flat.  Tolerance may be controlled
      with parameter _SURFACE_TOL (default=0.001 deg.).
      Number of points checked on each face controlled with parameter
      _SURFACE_MIDPT (default = 2). Select surface set "_FAIL_LOAD" to
      see problem surfaces.
```

Cross Section



TLAP BEARING AND BEARING LOADS CORRECTION

Definition of Correction loads

Corrective tractions are *added* to the **TLAP bearing** or **Bearing** traction distribution in order to correct the resultants for non-cylindrical holes or suboptimal meshes.

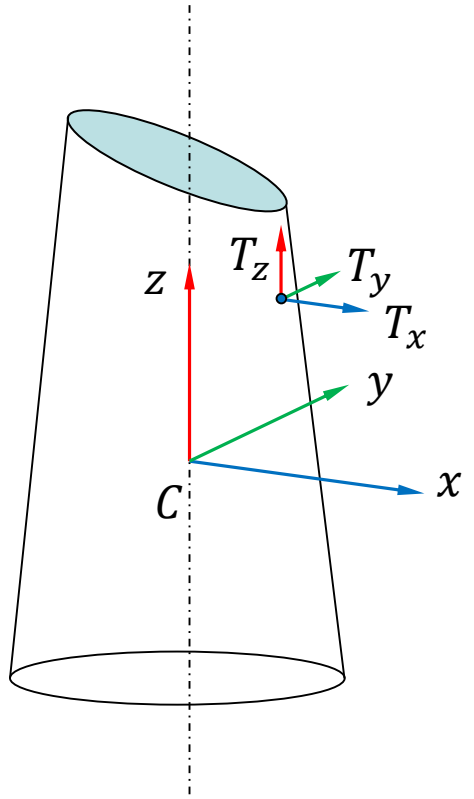
They are ***arbitrarily*** defined as:

$$T_x = C_1 + C_2 y + C_3 z \qquad T_y = C_4 + C_5 z \qquad T_z = C_6$$

In other words, the C_i coefficients are calculated so that the following resultant integrals match the required corrections loads:

$$F_x = \iint T_x dA \qquad F_y = \iint T_y dA \qquad F_z = \iint T_z dA$$

$$M_x = \iint \left(-T_y z + T_z y\right) dA \qquad M_y = \iint \left(T_x z - T_z x\right) dA \qquad M_z = \iint \left(-T_x y + T_y x\right) dA$$



(x, y, z) is centroidal

Calculation of correction loads

The expanded resultant correction integrals are:

$$F_x = \iint (C_1 + C_2 y + C_3 z)\,dA = C_1 A \Longrightarrow \boxed{C_1 = \frac{F_x}{A}}$$

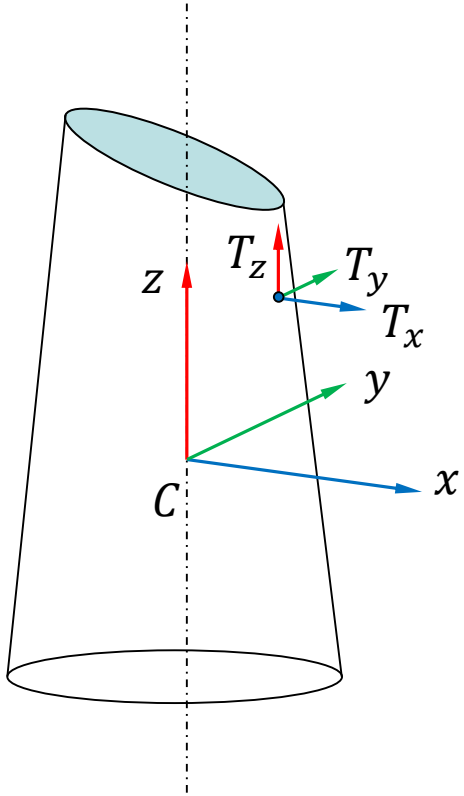
$$F_y = \iint (C_4 + C_5 z)\,dA = C_4 A \Longrightarrow \boxed{C_4 = \frac{F_y}{A}}$$

$$F_z = \iint C_6\,dA = C_6 A \Longrightarrow \boxed{C_6 = \frac{F_z}{A}}$$

$$M_x = \iint (-C_4 z - C_5 z^2 + C_6 y)\,dA = -C_5 I_{zz} \Longrightarrow \boxed{C_5 = -\frac{M_x}{I_{zz}}}$$

$$M_y = \iint (C_1 z + C_2 yz + C_3 z^2 - C_6 x)\,dA = C_2 I_{yz} + C_3 I_{zz}$$

$$M_z = \iint (-C_1 y - C_2 y^2 - C_3 yz + C_4 x + C_5 zx)\,dA = -C_2 I_{yy} - C_3 I_{yz} + C_5 I_{zx}$$



(x, y, z) is centroidal

Calculation of correction loads

$$M_y I_{yz} = C_2 I_{yz}^2 + C_3 I_{yz} I_{zz}$$

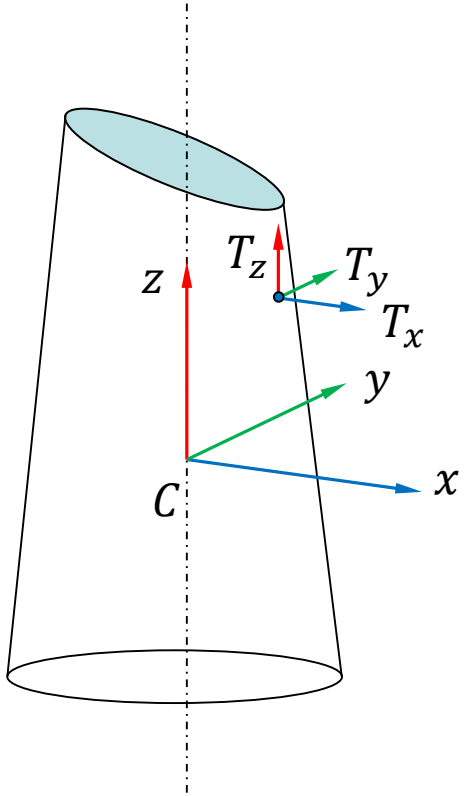
$$M_z I_{zz} + M_x I_{zx} = -C_2 I_{yy} I_{zz} - C_3 I_{yz} I_{zz}$$

$$C_2 = -\frac{M_y I_{yz} + M_z I_{zz} + M_x I_{zx}}{\left(I_{yy} I_{zz} - I_{yz}^2\right)}$$

$$M_y I_{yy} = C_2 I_{yz} I_{yy} + C_3 I_{yy} I_{zz}$$

$$M_z I_{yz} + \frac{M_x}{I_{zz}} I_{zx} I_{yz} = -C_2 I_{yz} I_{yy} - C_3 I_{yz}^2$$

$$C_3 = \frac{M_y I_{yy} + M_z I_{yz} + \frac{M_x}{I_{zz}} I_{zx} I_{yz}}{\left(I_{yy} I_{zz} - I_{yz}^2\right)}$$



(x, y, z) is centroidal

Calculation of correction loads

The corrective tractions are therefore given as:

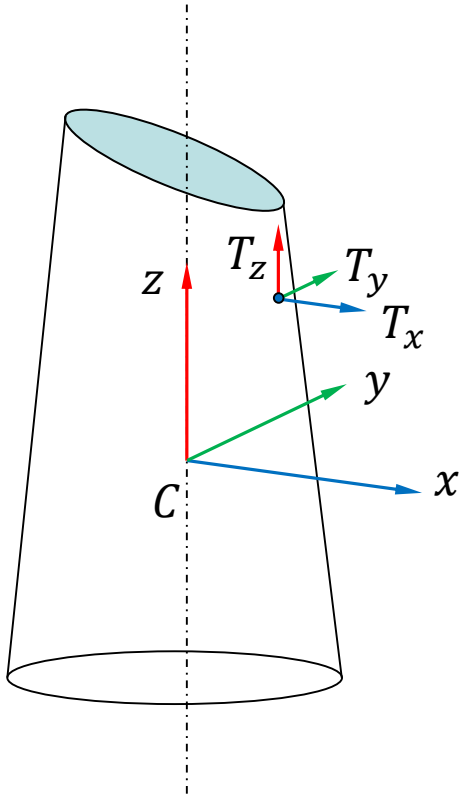
$$T_x = \frac{F_x}{A} - \frac{M_y I_{yz} + M_z I_{zz} + M_x I_{zx}}{\left(I_{yy} I_{zz} - I_{yz}^2\right)} y + \frac{M_y I_{yy} + M_z I_{yz} + \frac{M_x}{I_{zz}} I_{zx} I_{yz}}{\left(I_{yy} I_{zz} - I_{yz}^2\right)} z$$

$$T_y = \frac{F_y}{A} - \frac{M_x}{I_{zz}} z$$

$$T_z = \frac{F_z}{A}$$

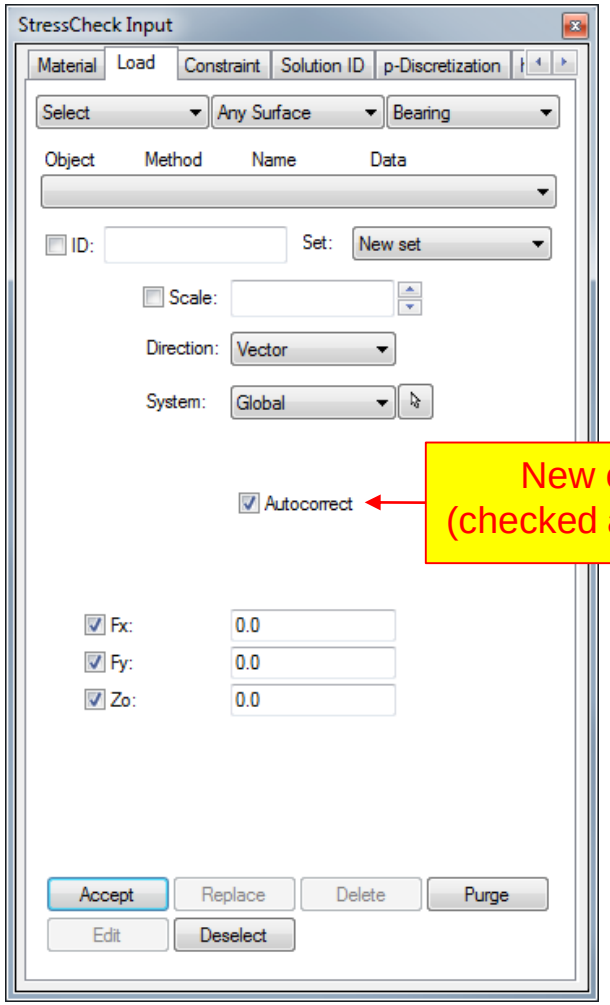
⚠ CAUTION

Bearing loads are defined as an approximation used to represent the pressure distribution caused by a lightly loose fit pin in contact with a perfect cylindrical hole. Deviations from that case (pin interference or hole geometry) will result on entirely different pressure distributions to that of the bearing formula.

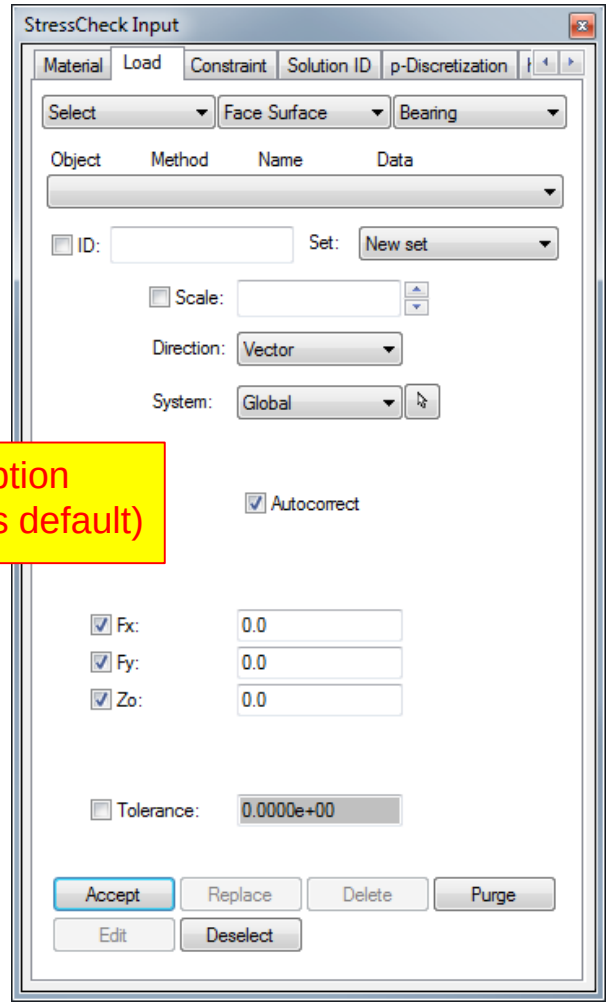


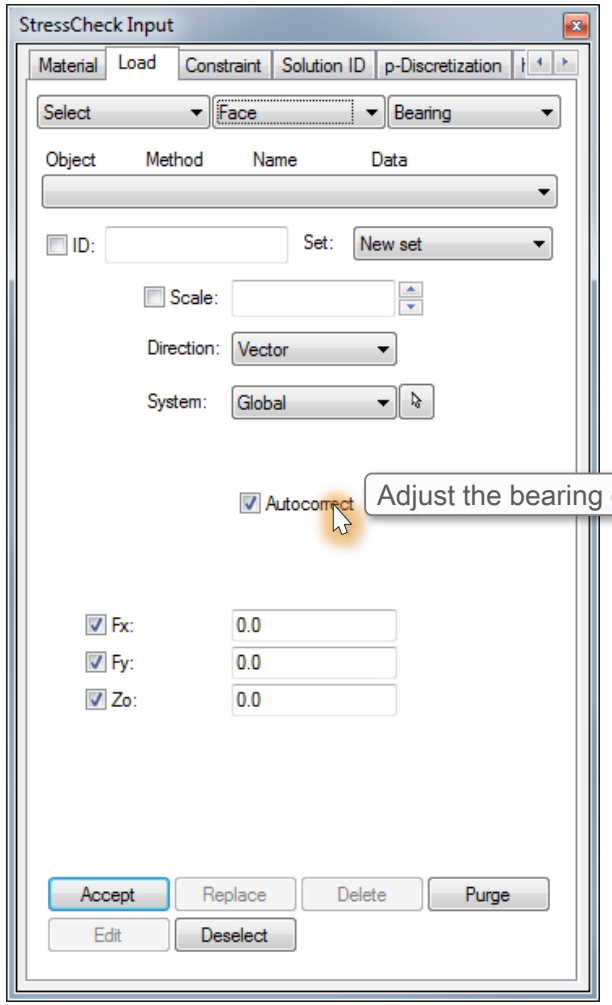
(x, y, z) is centroidal

Changes on the GUI: Bearing Load



New option
(checked as default)





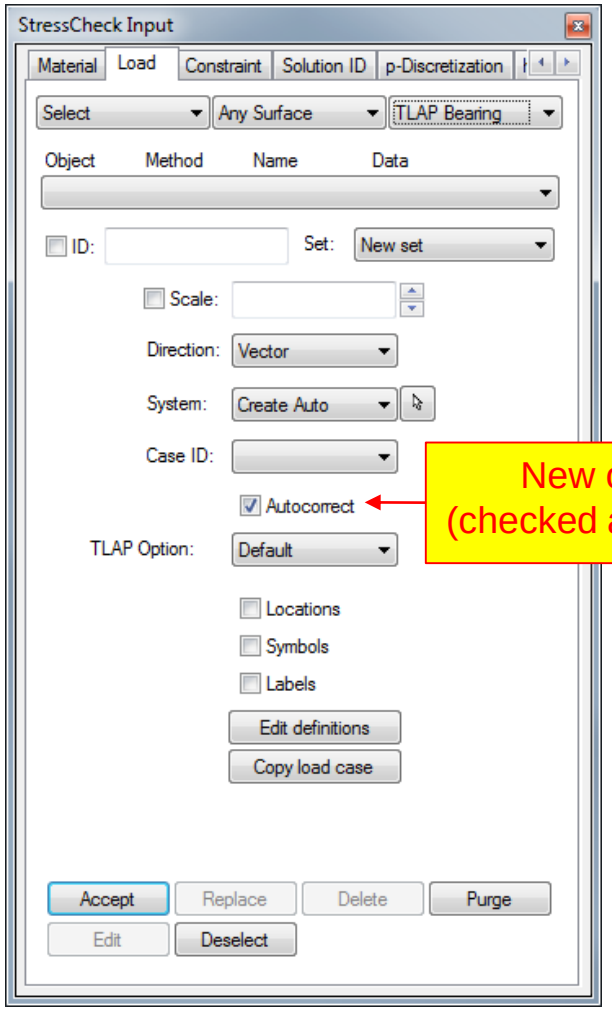
Adjust the bearing distribution to correct the resultant load

Tooltip

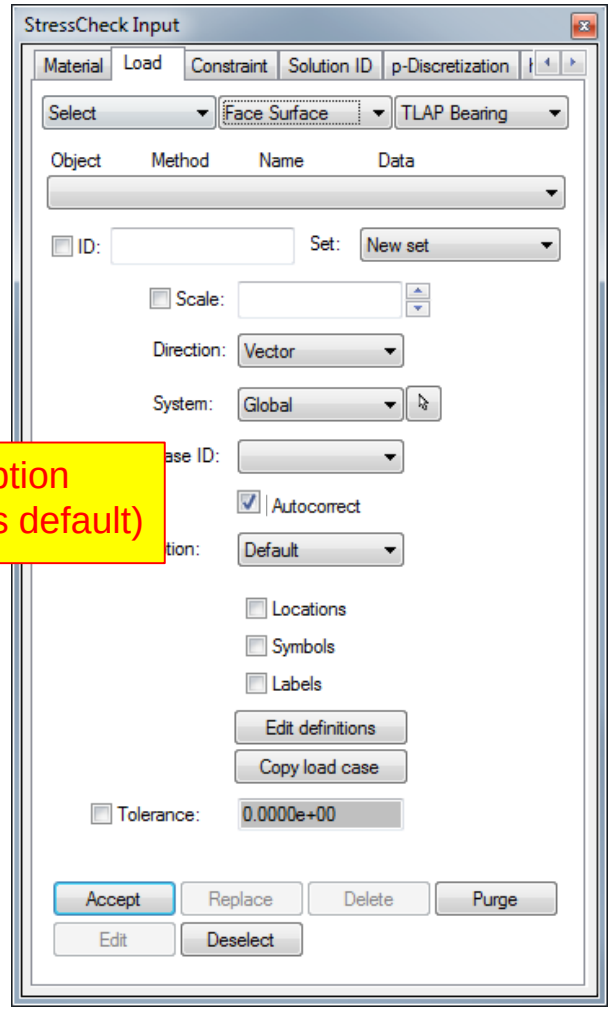
For files prior 10.5 any existing Bearing record will be read with the toggled off (no correction applied)

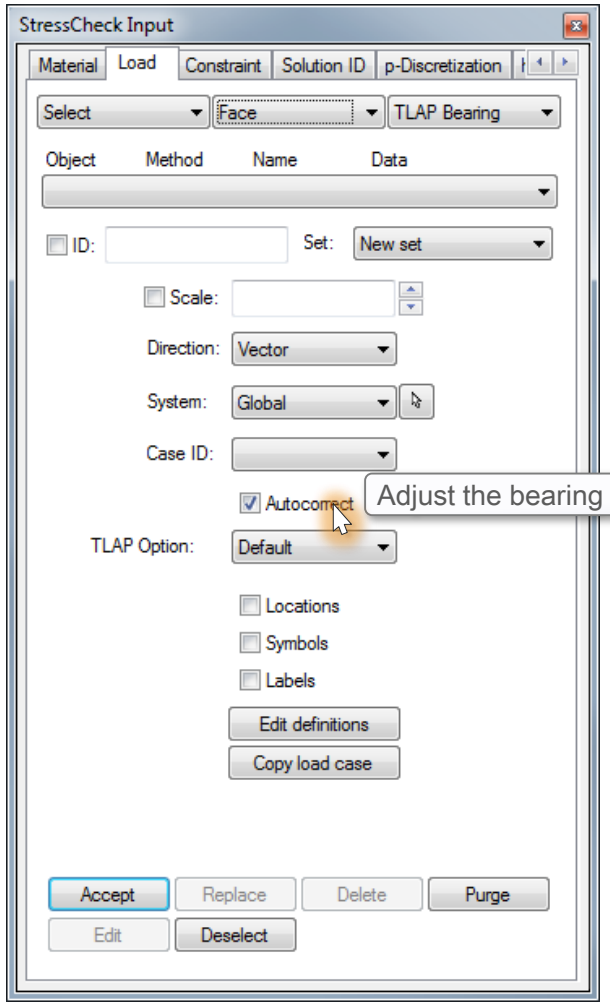
(Note: 3D Only)

Changes on the GUI: TLAP Bearing



New option (checked as default)





Adjust the bearing distribution to correct the resultant load

Tooltip

For files prior 10.5 any existing TLAP Bearing record will be read with the toggle off (no correction applied)

Additional changes

- Error and Warning messages for **TLAP bearing**:
  - When using Autocorrect, previous circularity and tapper error have been replaced with errors related to invalid selection and warning messages for circularity and correction tolerances:

```
ERROR

The selected surface type is invalid for TLAP bearing loads (only cylindrical type surfaces can be used). The record has not been created.


WARNING

   Circularity tolerance exceeded for TLAP bearing load ID: LoadID (SET#):

   The difference in max. and min. radius (#.######) is ##.####% different than the estimated avg. radius (#.######). This may indicate
that element faces do not resemble a cylindrical hole, or that the chosen system is not aligned with the hole centerline. If this is
unexpected, check your selection for unintended surfaces or faces. Define a value for parameter _bearing_tol to redefine the tolerance
check.


WARNING

   The total resultant force and/or moment for TLAP bearing load ID: LoadID (SET#) are being corrected by #% with respect to the local
system. This may occur if the element faces do not resemble a cylindrical hole. If this is unexpected, check your selection for unintended
surfaces or faces. Alternatively you can define a value for parameter _bearing_correction_tol to redefine the tolerance check (currently
0.05).
   Input resultants (local dir.) are    Fx:  #.##e+##, Fy:  #.##e+##, Fz:  #.##e+##, Mx:  #.##e+##, My:  #.##e+##, Mz:  #.##e+##
   Computed resultants (local dir.) are Fx:  #.##e+##, Fy:  #.##e+##, Fz:  #.##e+##, Mx:  #.##e+##, My:  #.##e+##, Mz:  #.##e+##
   Correction values (local dir.) are   Fx:  #.##e+##, Fy:  #.##e+##, Fz:  #.##e+##, Mx:  #.##e+##, My:  #.##e+##, Mz:  #.##e+##
```

Additional changes

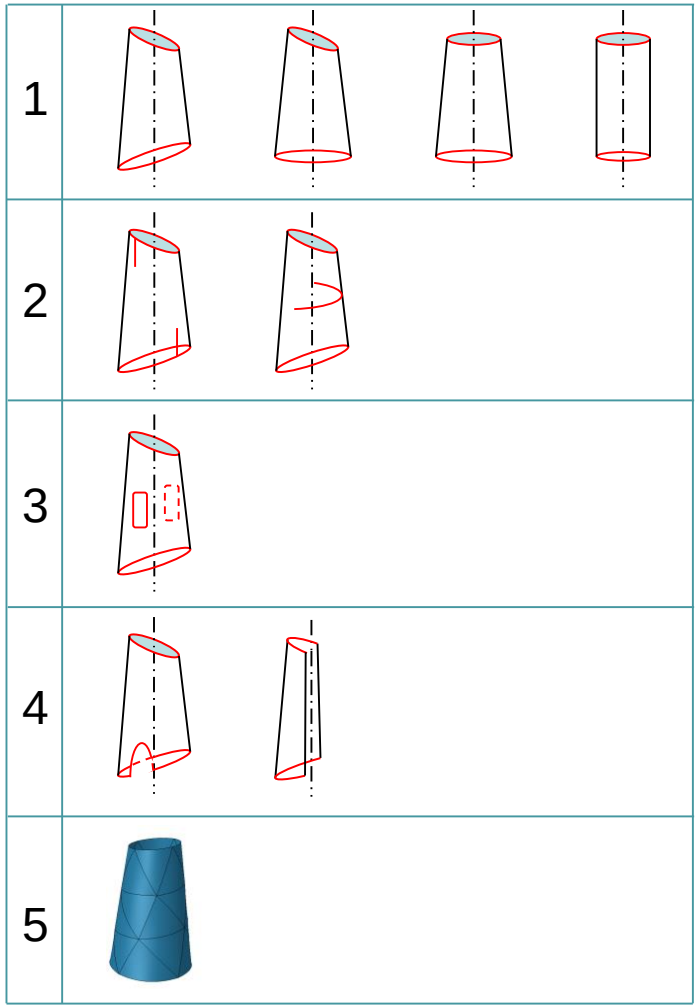
- Error and Warning messages for **Bearing** loads:
  - When using Autocorrect, previous circularity and tapper error have been replaced with errors related to invalid selection and warning messages for system selection, circularity and correction tolerances:

```
ERROR

The selected surface type is invalid for Bearing loads (only cylindrical type surfaces can be used). The record has not been created.


WARNING

   The chosen system for bearing load ID: Load (SET#) is significantly offset from the hole centroidal axis. Please check your system selection to ensure
it is aligned with the hole.


WARNING

   Circularity tolerance exceeded for bearing load ID: LoadID (SET#):

   The difference in max. and min. radius (#.######) is ##.####% different than the estimated avg. radius (#.######). This may indicate
that element faces do not resemble a cylindrical hole, or that the chosen system is not aligned with the hole centerline. If this is
unexpected, check your selection for unintended surfaces or faces. Define a value for parameter _bearing_tol to redefine the tolerance
check.


WARNING

   The total resultant force and/or moment for bearing load ID: LoadID (SET#) are being corrected by #% with respect to the local system.
This may occur if the element faces do not resemble a cylindrical hole. If this is unexpected, check your selection for unintended
surfaces or faces. Alternatively you can define a value for parameter _bearing_correction_tol to redefine the tolerance check (currently
0.05).
   Input resultants (local dir.) are    Fx:  #.##e+##, Fy:  #.##e+##, Fz:  #.##e+##, Mx:  #.##e+##, My:  #.##e+##, Mz:  #.##e+##
   Computed resultants (local dir.) are Fx:  #.##e+##, Fy:  #.##e+##, Fz:  #.##e+##, Mx:  #.##e+##, My:  #.##e+##, Mz:  #.##e+##
   Correction values (local dir.) are   Fx:  #.##e+##, Fy:  #.##e+##, Fz:  #.##e+##, Mx:  #.##e+##, My:  #.##e+##, Mz:  #.##e+##
```

Acceptable deviations from cylindrical holes

Acceptable variations include:

1. General case: Conical cylinder with elliptical tapered ends.
2. Embedded cracks
3. Additional (any shape) holes
4. Truncated*
5. Faceted (element faces not mapped directly on the cylindrical surface)

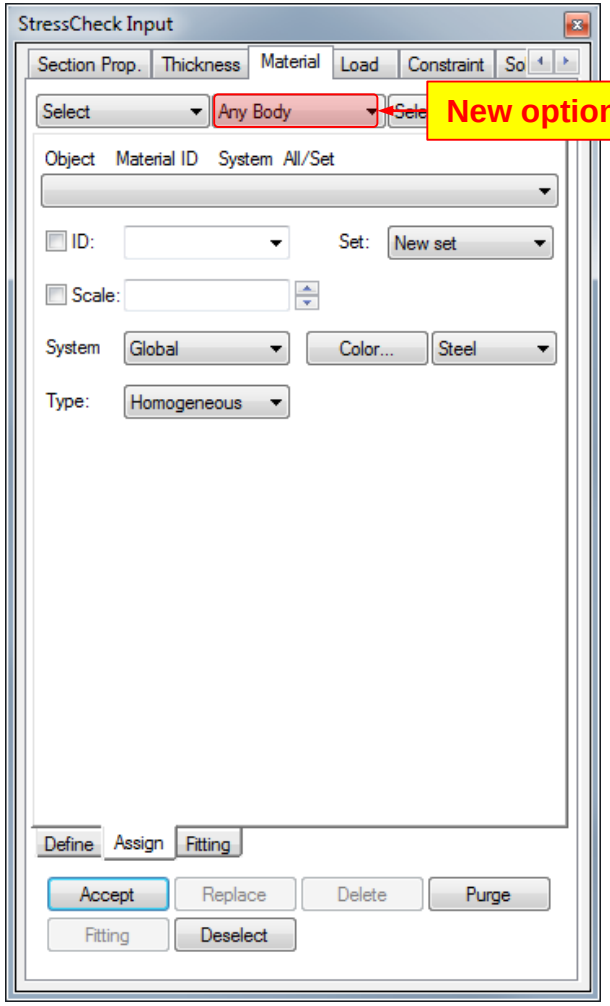


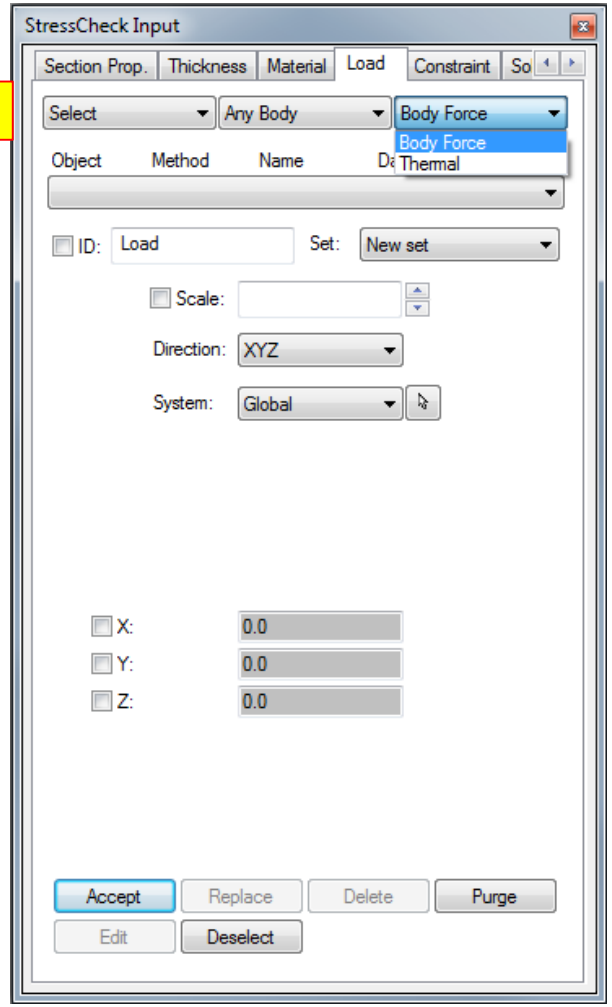
⚠ *CAUTION

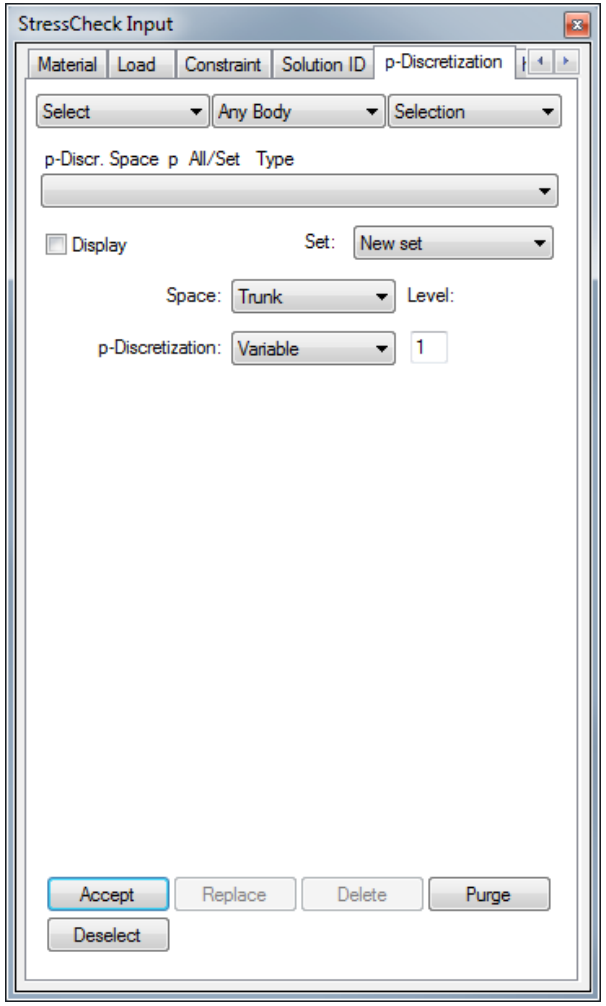
Note that the Bearing load correction acts correcting the resultants based on input. Therefore if the bearing load direction is so that part or all of the bearing formula is not acting on any elements the resultant will still be corrected so that it is the same as the input.

NEW ANY BODY OPTION FOR ASSIGNMENT

Any Body option for Assignment:
Material Assign – Load – p-Discretization tabs







Note: "Mesh Region" and "Any Body" methods are complementary in the following sense:

- Elements created inside the body but not associated to the body will not be recognized by the option "Any Body" but will be by the option "Mesh Region".
- Elements associated to the body that are not fully contained in the body will not be recognized by the option "Mesh Region" but will be recognized by the option "Any Body".

PLASTIC STRAIN EXTRACTIONS

Extraction of plastic strains

- Plastic strains can be computed from the total strain and mechanical strain components.

Total Strain — Elastic strain — Plastic strain — Thermal strain

$$\varepsilon_{ij} = \varepsilon_{ij}^{e} + \varepsilon_{ij}^{p} + \alpha T_{\Delta}\delta_{ij}$$

$$\varepsilon_{ij}^{p} = \varepsilon_{ij} - \varepsilon_{ij}^{e} - \alpha T_{\Delta}\delta_{ij}$$

$$\varepsilon_{ij}^{p} = \varepsilon_{ij} - \frac{1+\nu}{E}\sigma_{ij} + \frac{\nu}{E}\sigma_{kk}\delta_{ij} - \alpha T_{\Delta}\delta_{ij}$$

$$\varepsilon_{x}^{p} = \varepsilon_{x} - \frac{1}{E}\left(\sigma_{x} - \nu\sigma_{y} - \nu\sigma_{z}\right) - \alpha T_{\Delta}$$

$$\varepsilon_{y}^{p} = \varepsilon_{y} - \frac{1}{E}\left(-\nu\sigma_{x} + \sigma_{y} - \nu\sigma_{z}\right) - \alpha T_{\Delta}$$

$$\varepsilon_{z}^{p} = \varepsilon_{z} - \frac{1}{E}\left(-\nu\sigma_{x} - \nu\sigma_{y} + \sigma_{z}\right) - \alpha T_{\Delta}$$

$$\gamma_{xy}^{p} = 2\varepsilon_{xy}^{p} = \gamma_{xy} - \frac{2(1+\nu)}{E}\tau_{xy} = \gamma_{xy} - \frac{\tau_{xy}}{G}$$

$$\gamma_{yz}^{p} = 2\varepsilon_{yz}^{p} = \gamma_{yz} - \frac{\tau_{yz}}{G}$$

$$\gamma_{zx}^{p} = 2\varepsilon_{zx}^{p} = \gamma_{zx} - \frac{\tau_{zx}}{G}$$

Extraction of plastic strains

- Principal, and Equivalent plastic strains:

$$\sigma_{eq} \overset{\text{def}}{=} \sqrt{\frac{3}{2} S_{ij} S_{ij}} = \sqrt{\frac{3}{2}\left(\sigma_{ij} - \frac{1}{3}\sigma_{kk}\delta_{ij}\right)\left(\sigma_{ij} - \frac{1}{3}\sigma_{kk}\delta_{ij}\right)} = \frac{E}{(1+\nu)}\sqrt{\frac{3}{2}\left(\varepsilon_{ij} - \frac{1}{3}\varepsilon_{kk}\delta_{ij}\right)\left(\varepsilon_{ij} - \frac{1}{3}\varepsilon_{kk}\delta_{ij}\right)}$$

$$\sigma_{eq} = \frac{E}{(1+\nu)}\sqrt{\frac{1}{2}\left[(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2\right]} \qquad \varepsilon_{eq} \underset{\sigma_{eq} := E\,\varepsilon_{eq}}{=} \frac{1}{(1+\nu)}\sqrt{\frac{1}{2}\left[(\varepsilon_1 - \varepsilon_2)^2 + (\varepsilon_2 - \varepsilon_3)^2 + (\varepsilon_3 - \varepsilon_1)^2\right]}$$

For plastic strains $\nu = 0.5$

$$\Rightarrow \quad \boxed{\varepsilon_{eq}^{p} = \sqrt{\frac{2}{9}\left[\left(\varepsilon_1^p - \varepsilon_2^p\right)^2 + \left(\varepsilon_2^p - \varepsilon_3^p\right)^2 + \left(\varepsilon_3^p - \varepsilon_1^p\right)^2\right]}}$$

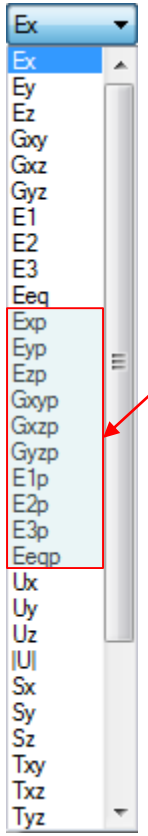
Principal plastic strains are computed as:

$$\left|\varepsilon_{ij}^{p} - \varepsilon^{p}\delta_{ij}\right| = 0$$

$$\varepsilon_1^p \geq \varepsilon_2^p \geq \varepsilon_3^p$$

Changes to the UI

- The new extraction functions are available for the Plot, Min/Max and Points tabs in the Results StressCheck dialog.



- Listed between Eeq and Ux (right after Eeq).
- Available in COM

Exp
Eyp
Ezp
Gxyp
Gxzp
→ Directional plastic strains

E1p
E2p
E3p
→ Principal plastic strains

Eeqp
→ Equivalent plastic strain

Note: The implementation is restricted to 3D Incremental Theory of Plasticity (ITP) solutions. Therefore plastic strain extraction functions are only available when a 3D IPT solution is selected.

LOG IMPROVEMENTS FOR INCREMENTAL THEORY OF PLASTICITY (ITP) SOLUTIONS

Log Improvements for ITP Solutions


```
EXECUTION SETUP SETTINGS:

      Title = Plate with offset hole
      Reference = 2D
      Theory = Elasticity
      Solution ID = NSOL
      Startup = Initialize
      Convergence = Upward
      Iteration = Automatic
      Run type = Nonlin-Mat
      Run Limit = 1
      Elastic-Plastic Analysis (Incremental Theory of Plasticity)
      Nonlinear ID: NSOL
      Linear ID: SOL, Linear run: 8
      Nonlinear convergence criteria = Displacement
             Tolerance (%) = 5.000e-03, Iteration Limit: 10
      Number of Events: 4
```

- Coalesced the information shared by all Events/Steps and included selected theory

Log Improvements for ITP Solutions

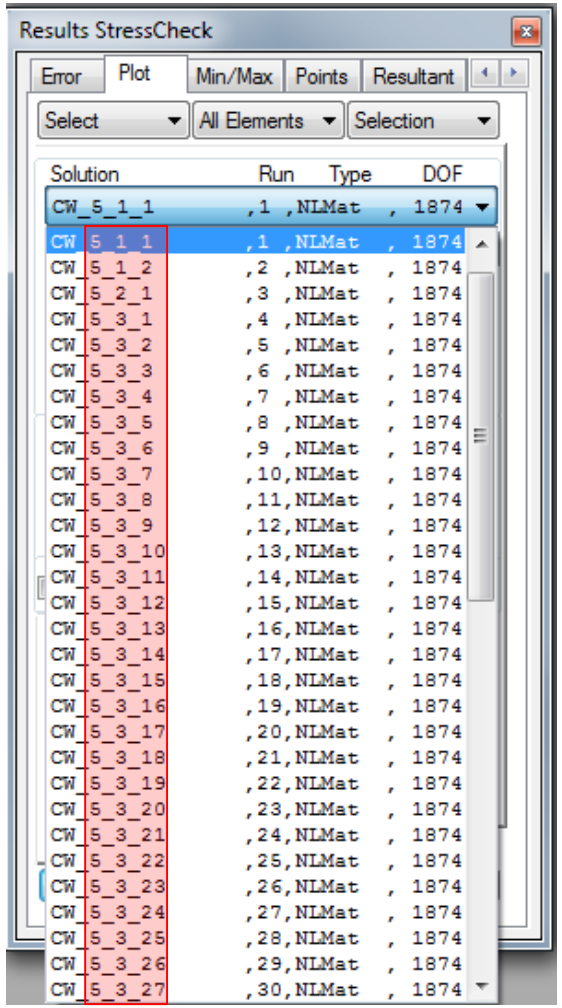
STRESSCHECK

```
Event: 1 - Cold-Working
       Step: 1 of 2
       Solution ID:   NSOL_8_1_1
       Boundary Condition Parameters:
              Delta = 0.0115
              Emf = 3e+07
              Tx = 0
              Pf = 0
       Convergence information:
              Iteration 1: Largest error = 100.000% at element 2, total unconverged elements = 38
              Iteration 2: Largest error = 27.547% at element 38, total unconverged elements = 38
              Iteration 3: Largest error = 5.575% at element 5, total unconverged elements = 38
              Iteration 4: Largest error = 1.238% at element 5, total unconverged elements = 38
              Iteration 5: Largest error = 0.072% at element 5, total unconverged elements = 38
              Iteration 6: Largest error = 0.000% at element 5, total unconverged elements = 0
       Step solution time = 16 seconds.

       Step: 2 of 2
       Solution ID:   NSOL_8_1_2
       Boundary Condition Parameters:
              Delta = -0.001
              Emf = 1000
              Tx = 0
              Pf = 0
       Convergence information:
              Iteration 1: Largest error = 77.675% at element 38, total unconverged elements = 38
              Iteration 2: Largest error = 0.540% at element 41, total unconverged elements = 38
              Iteration 3: Largest error = 0.003% at element 8, total unconverged elements = 0
       Step solution time = 8 seconds.
```

- Event name and number, as well as the solution name for each step are now logged.
- Parameters defined as B. Cond. are added to the log together with their corresponding value for a given Event/Step.
- Convergence information for each load step is contained within the Event

NAME SORTING FOR ITP SOLUTIONS

Name sorting for ITP solutions

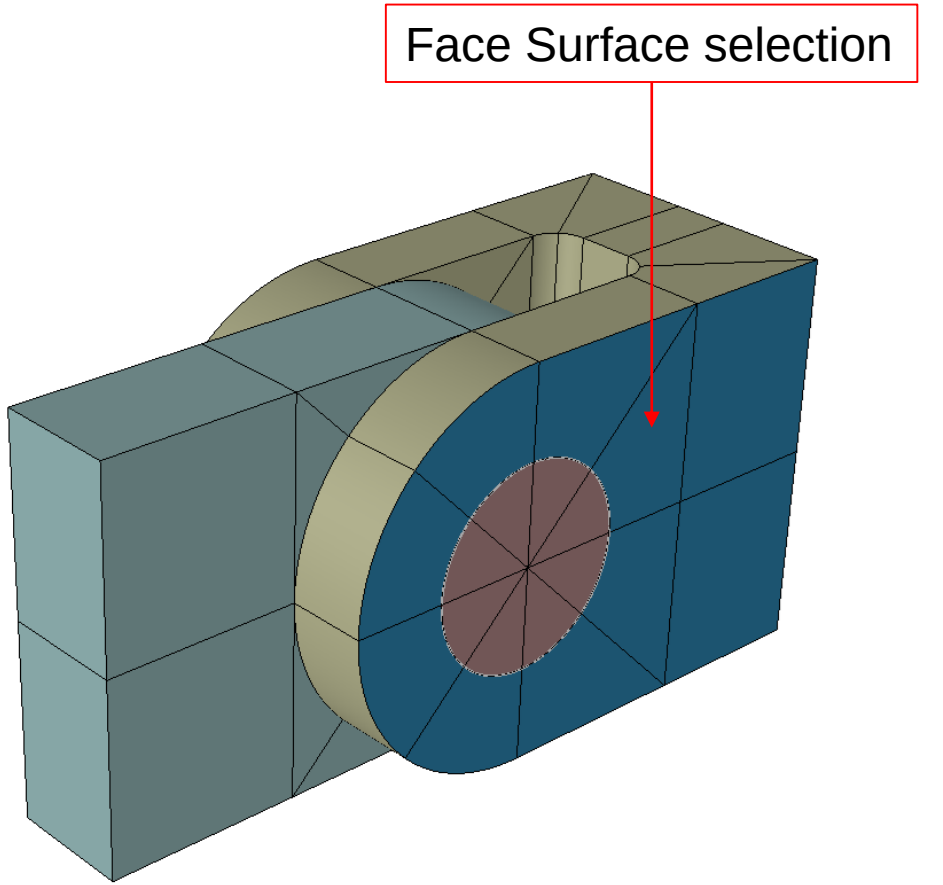


IPT solutions are now sorted *sequentially* based on linear solution run, Event and Step number respectively.

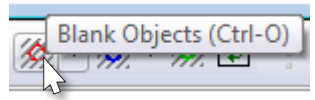
FACE/FACE SURFACE ELEMENT BLANK

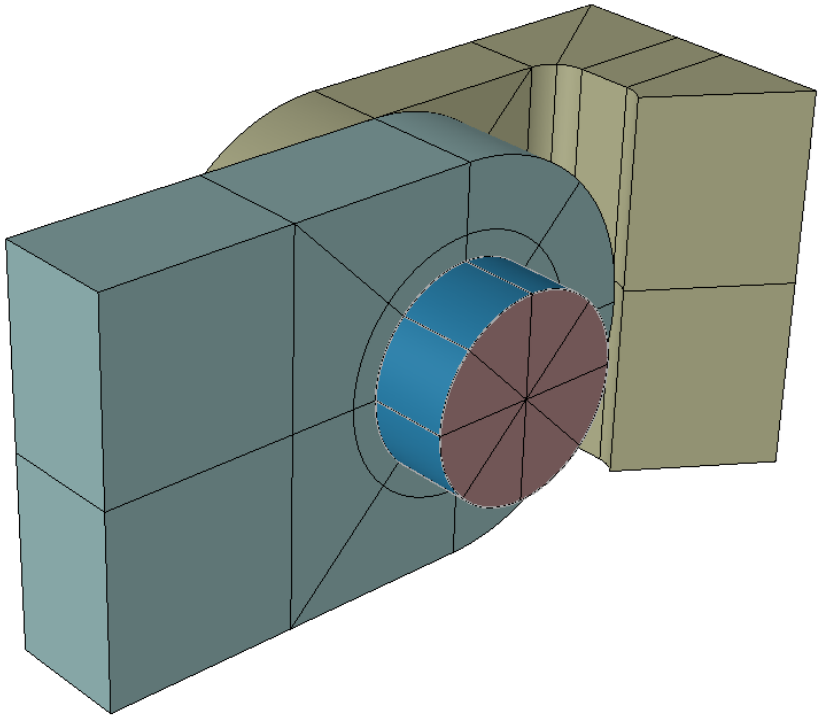
Face/Face Surface element blank

- Use Face or Face Surface selection to blank elements:



Blank Objects



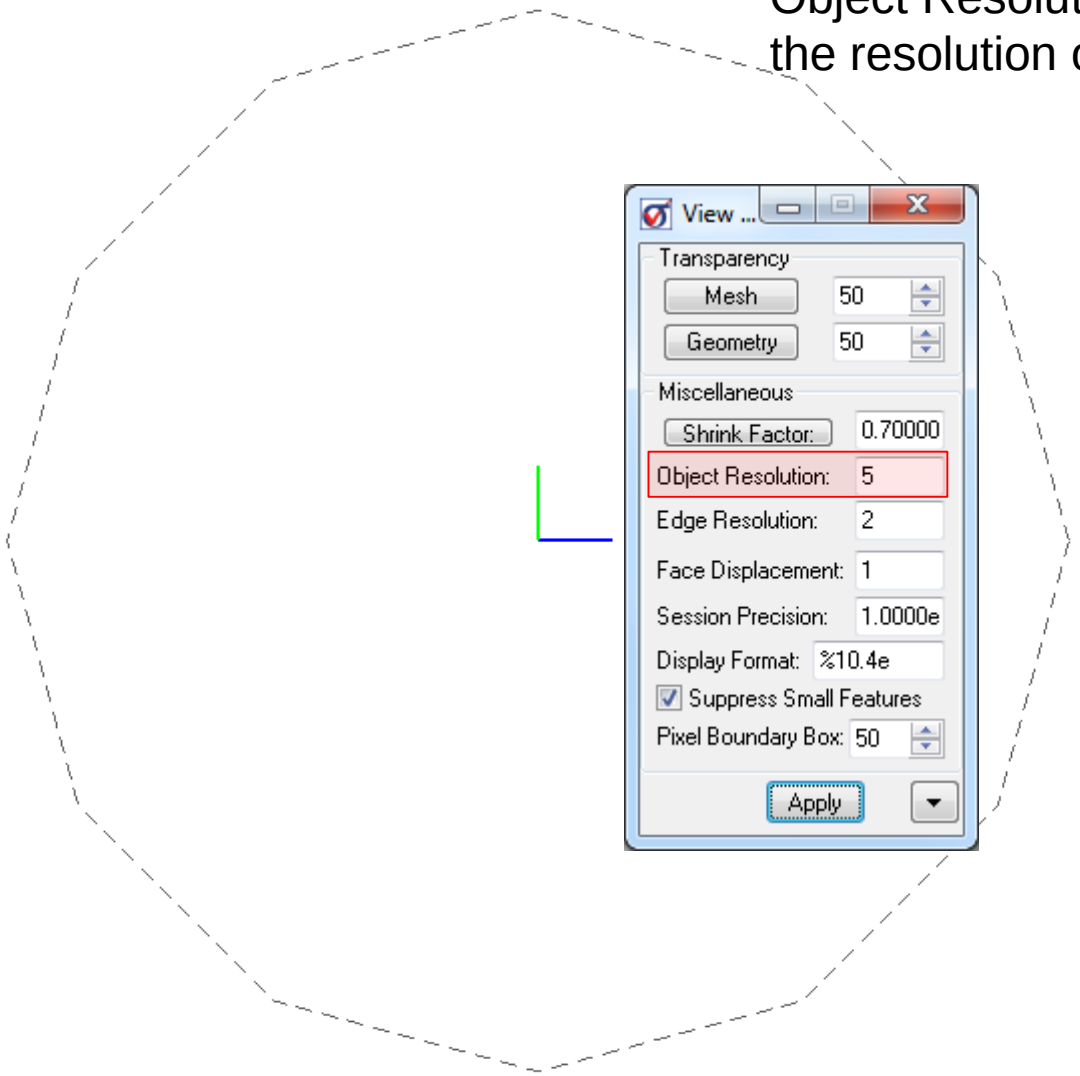


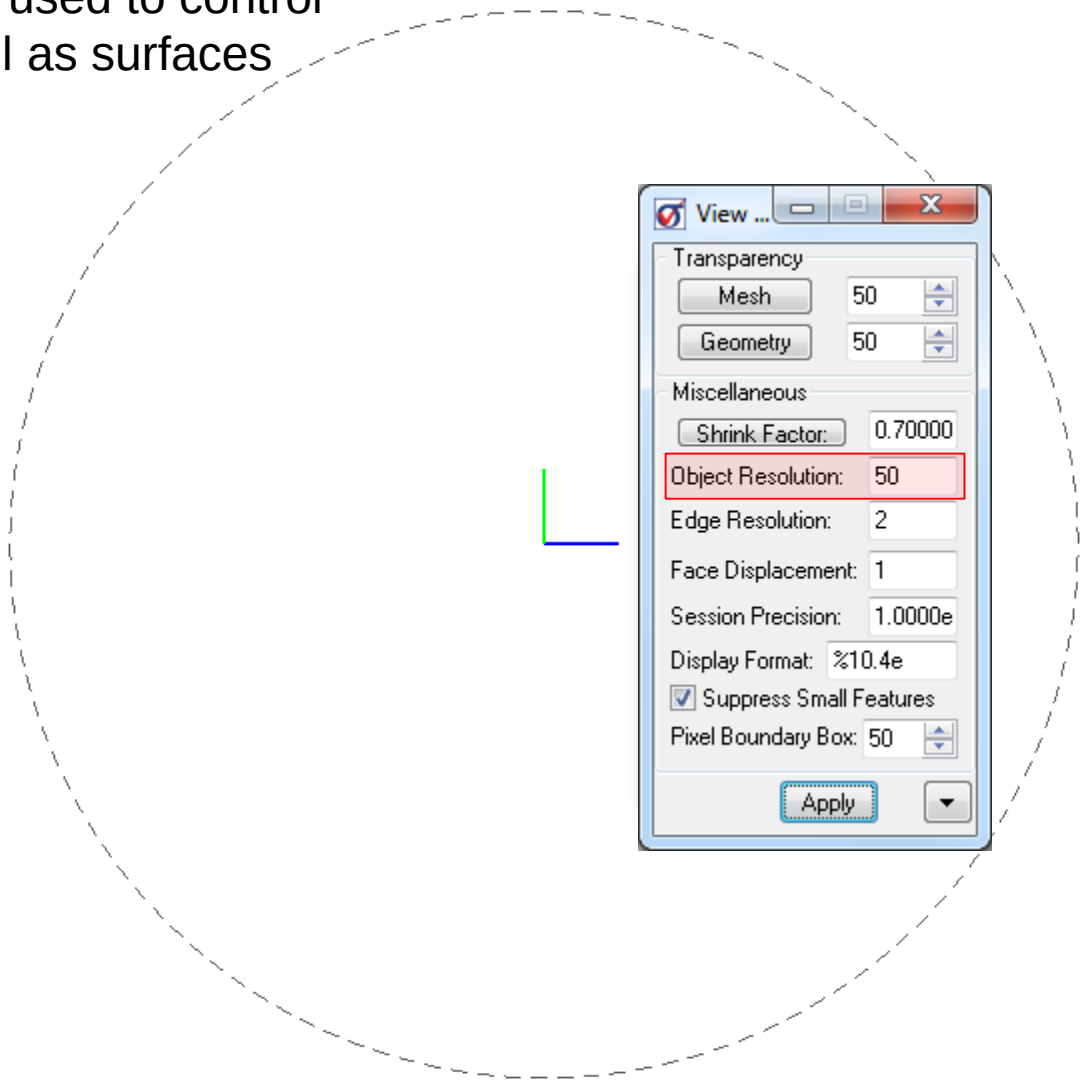
CURVES RESOLUTION CAN NOW BE CONTROLLED

Curves resolution can now be controlled

STRESSCHECK

Object Resolution can now be used to control the resolution of curves as well as surfaces

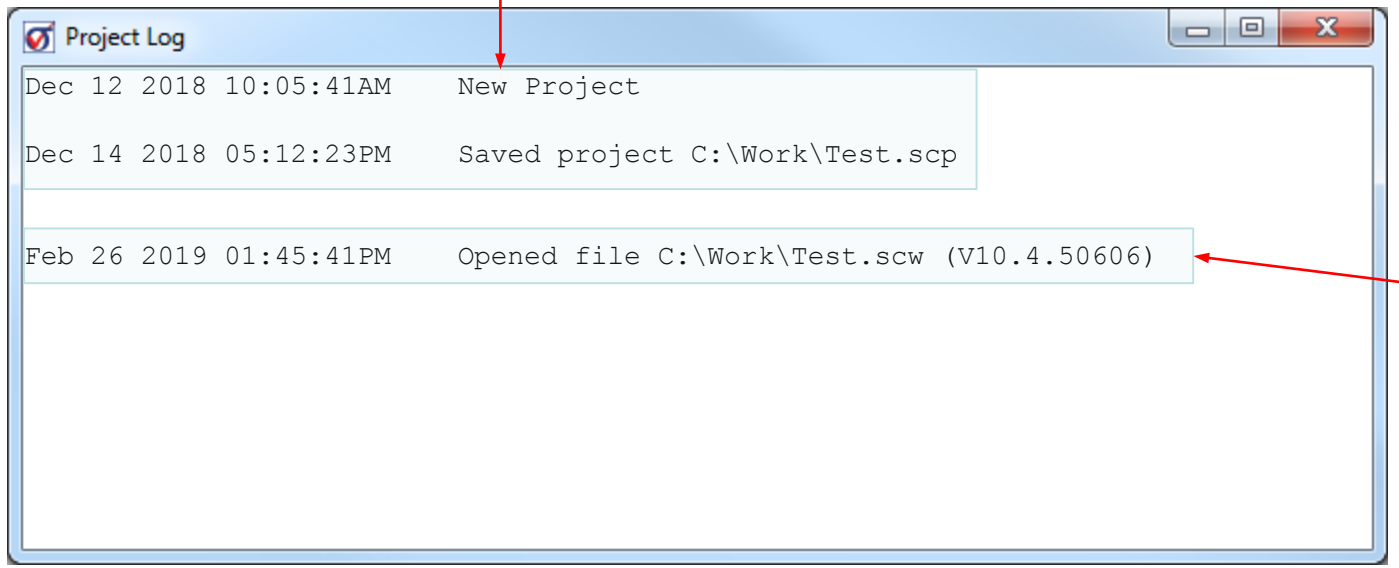




File Build number is now recorded in the Project Log

File Build number is now recorded in the Project Log

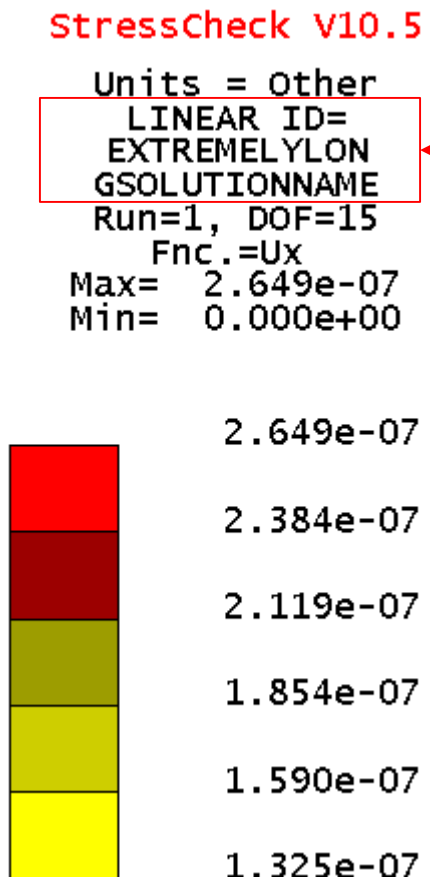
Previous project log entries from the original file



New log entry. When opening an existing file, the version and build number from which it was last saved are displayed.

LONG SOLUTION NAMES ARE NOW WRAPPED ON THE PLOT LEGEND

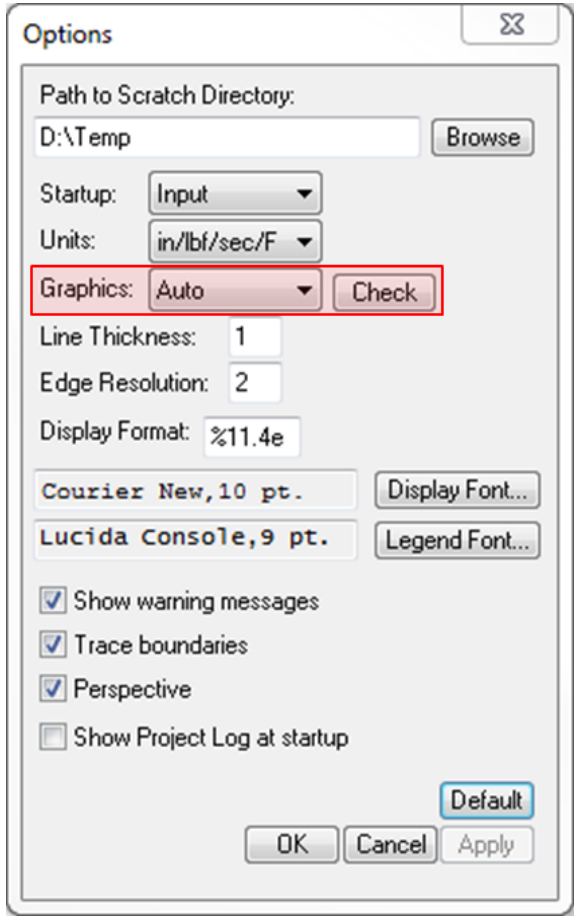
Long solution names



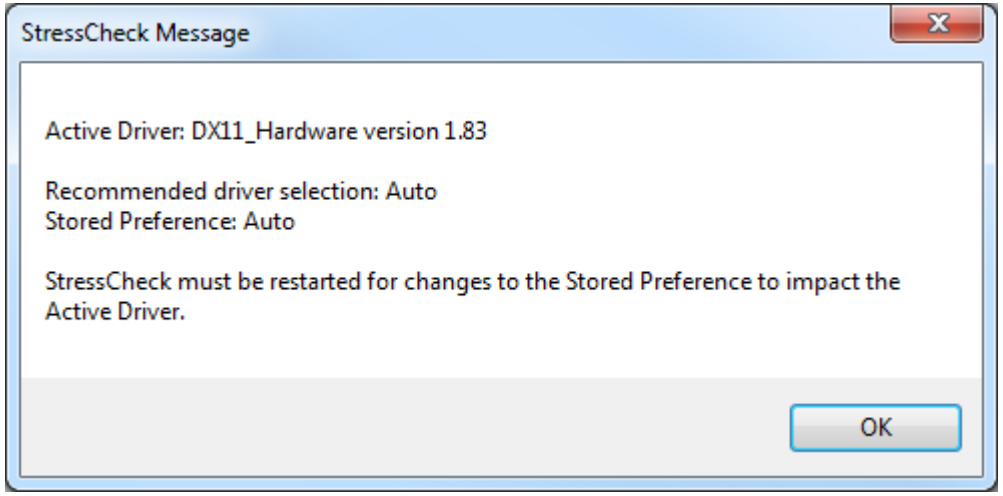
Long solution names are longer truncated but wrapped in the Plot legend (up to 3 lines)

Automatic And Manual Selection Of Graphic Drivers

Automatic and manual selection of graphic drivers



- By default, StressCheck® automatically selects the most appropriate graphics driver based on the current runtime environment.
- In case needed or desired by the user an option has been added in the Options dialog to choose between driver preferences: Automatic, DirectX 11, DirectX 9, OpenGL 2, and OpenGL.
- The Check button displays a message indicating which driver is currently in use, the recommended selection (Auto), and the stored preference



QUESTIONS OR COMMENTS?

Contact support@esrd.com