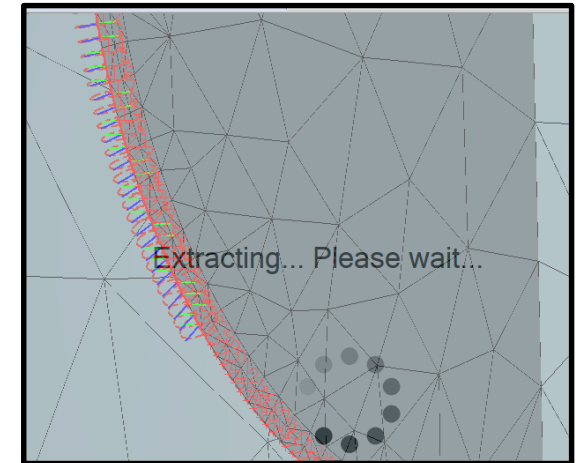
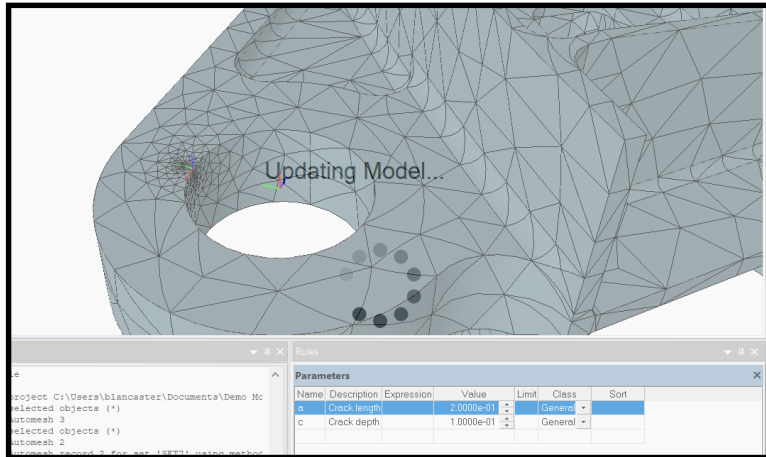


# New Features and Enhancements in StressCheck 11

ESRD, Inc.  
December 2020



# New Features and Enhancements in SC 11



STRESSCHECK

- ❑ Upgraded User Interface and User Flow Improvements
- ❑ Enhanced Automeshing Functionality
- ❑ New Crack Front Automeshing Method
- ❑ Automatic Extraction Radius for Fracture Mechanics Parameters
- ❑ Enhanced Solver Settings for Multi-Body Contact Analysis
- ❑ New Extraction Functions for Multi-Body Contact Results
- ❑ Enhanced Load/Constraint Experience
- ❑ Improved Cutting Plane Performance
- ❑ New Force/Length Option for Spring Coefficients
- ❑ Improved Mesh Region Assignment
- ❑ New Error and Warning Message Handlers
- ❑ Online Documentation Portal



TM

# **UPGRADED USER INTERFACE AND USER FLOW IMPROVEMENTS**

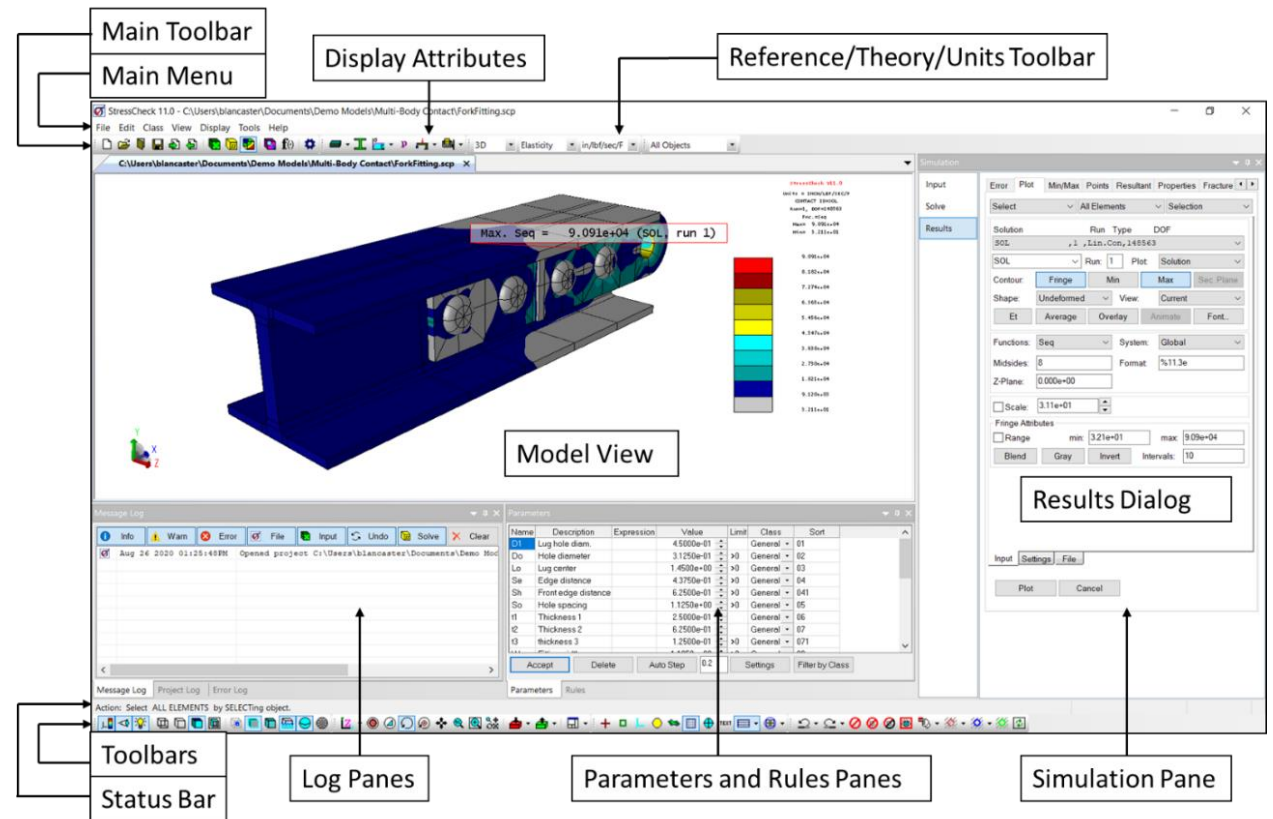
# Upgraded User Interface Experience



STRESSCHECK™

- The StressCheck user interface presentation, layout and user experience has been significantly upgraded.

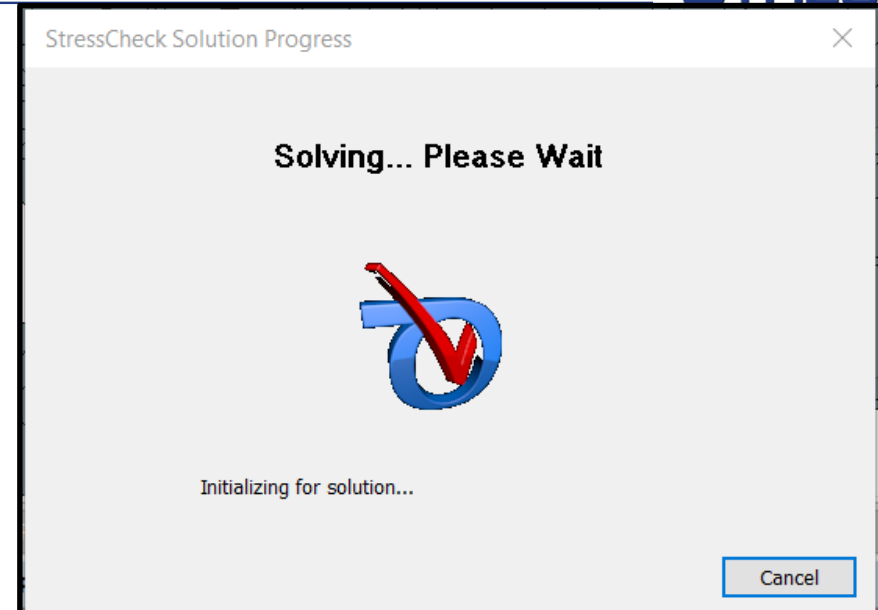
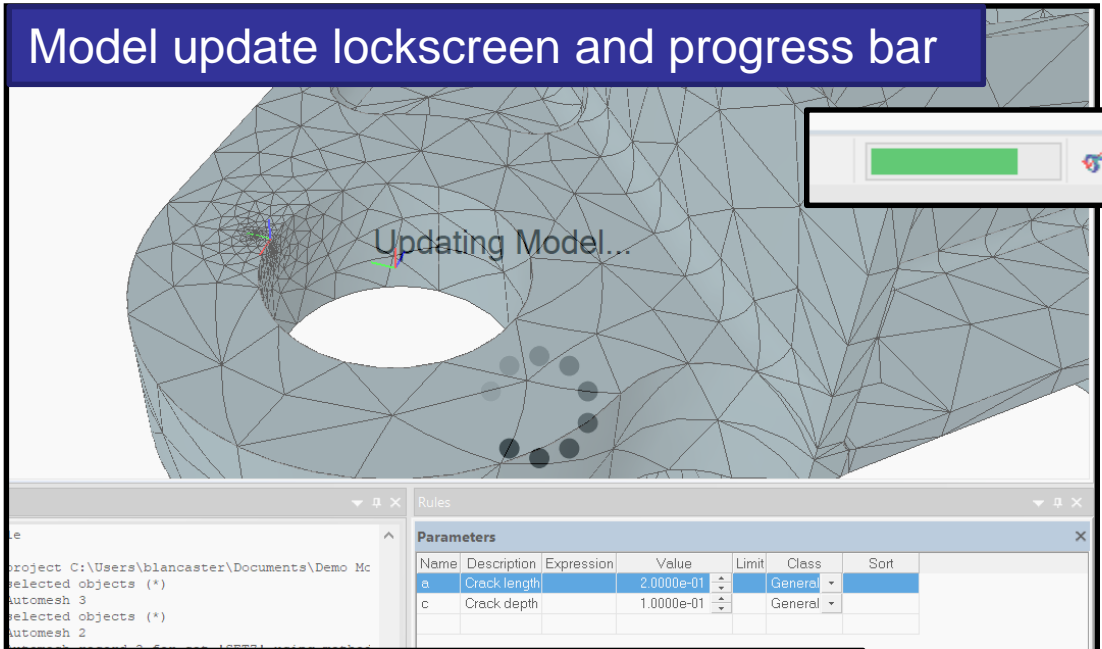
- Dockable panes
- Lockscreens for file I/O, model updating, plots, extractions, etc.
- Progress bar on SC taskbar icon
- Undo/Redo lists
- Recent file list under File > Open
- Semi automatic file upgrade utility



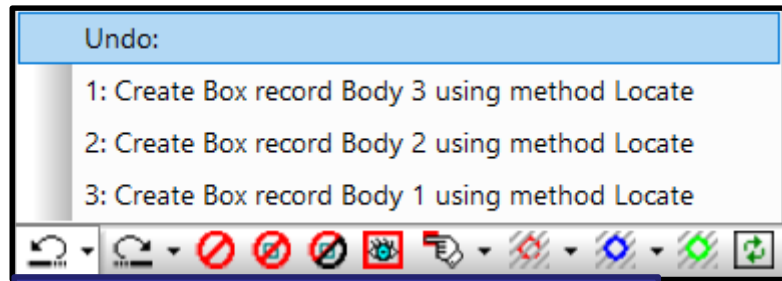
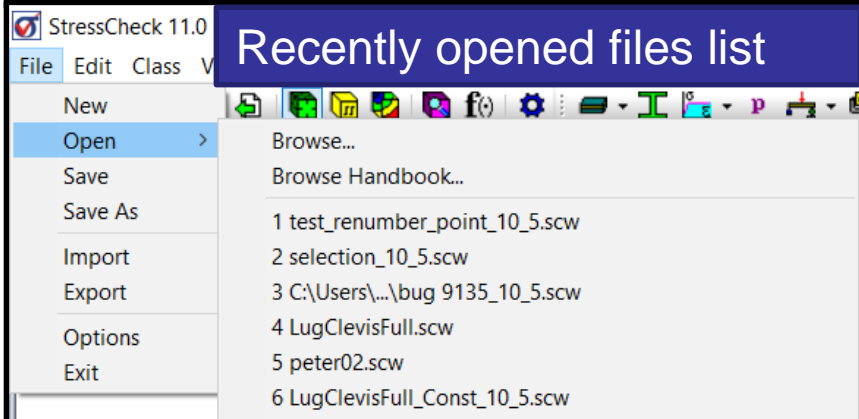
# Upgraded User Interface Experience



STRESSCHECK



Solution progress dialog and progress bar on SC icon



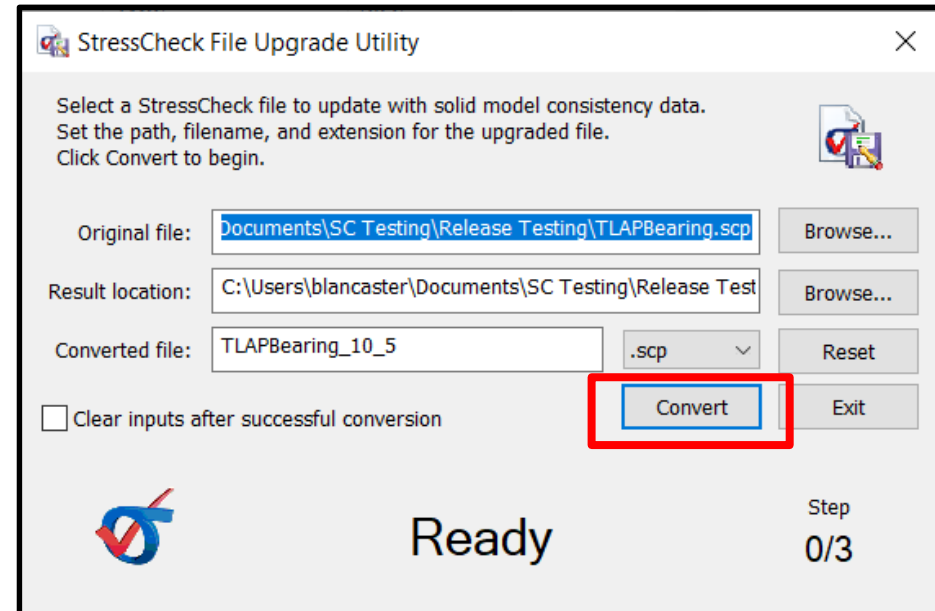
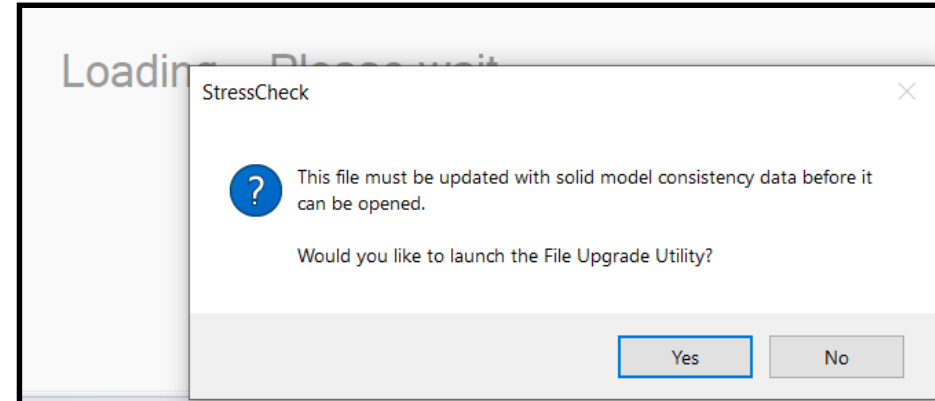
Undo/Redo lists

# Upgraded User Interface Experience



STRESSCHECK

- ❑ If StressCheck workfiles or project files were generated in versions prior to StressCheck v10.5, a semi-automatic file upgrade utility is available to ensure compatibility with the latest solid modeling conventions.
  - Dialog to launch utility will appear if StressCheck detects older file version.
  - Browse to upgrade path and provide converted file name.
  - Click Convert to upgrade original file.
  - Select upgraded file and open.





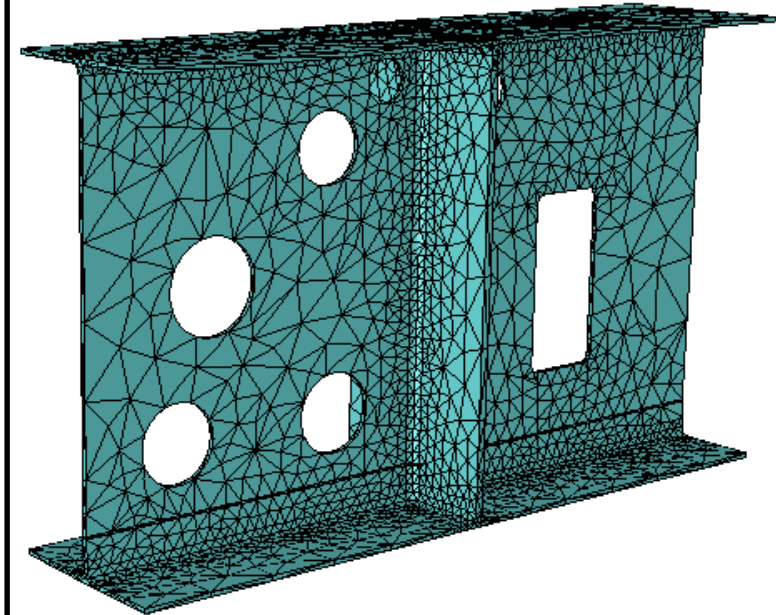
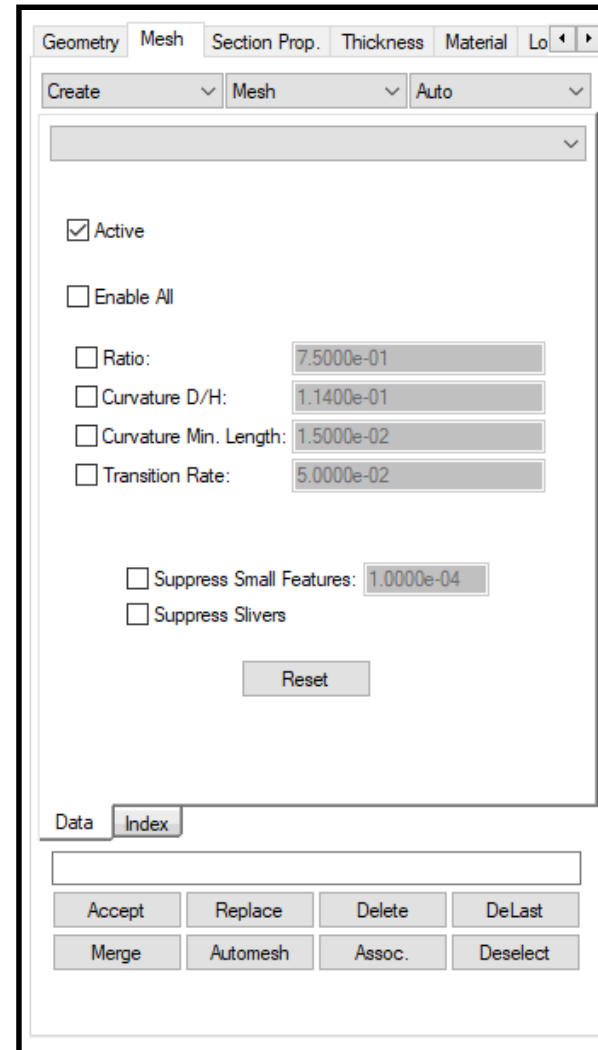
**ENHANCED AUTOMESHING FUNCTIONALITY** <sup>TM</sup>

# Enhanced Automeshing Functionality



STRESSCHECK

- The latest MeshSim automeshing libraries have been implemented and the automesh inputs have been revamped to provide additional clarity and options.
  - Relabeling of input fields.
  - Tooltips for input fields indicating default values and valid ranges.
  - Defaults optimized for complex 3D parts.
  - Small feature meshing and suppression options now available in the GUI.







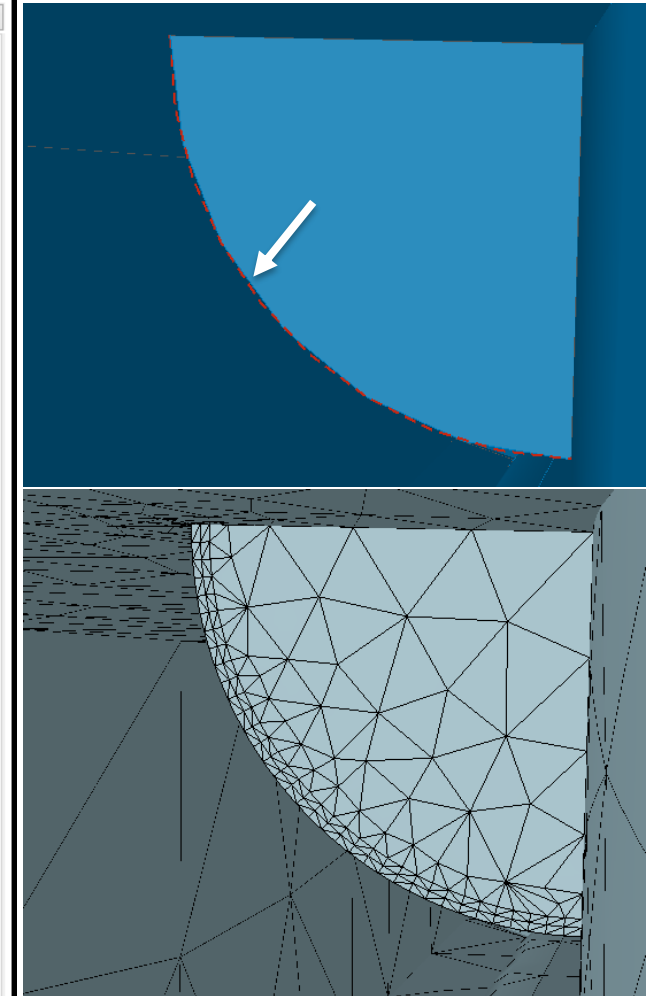
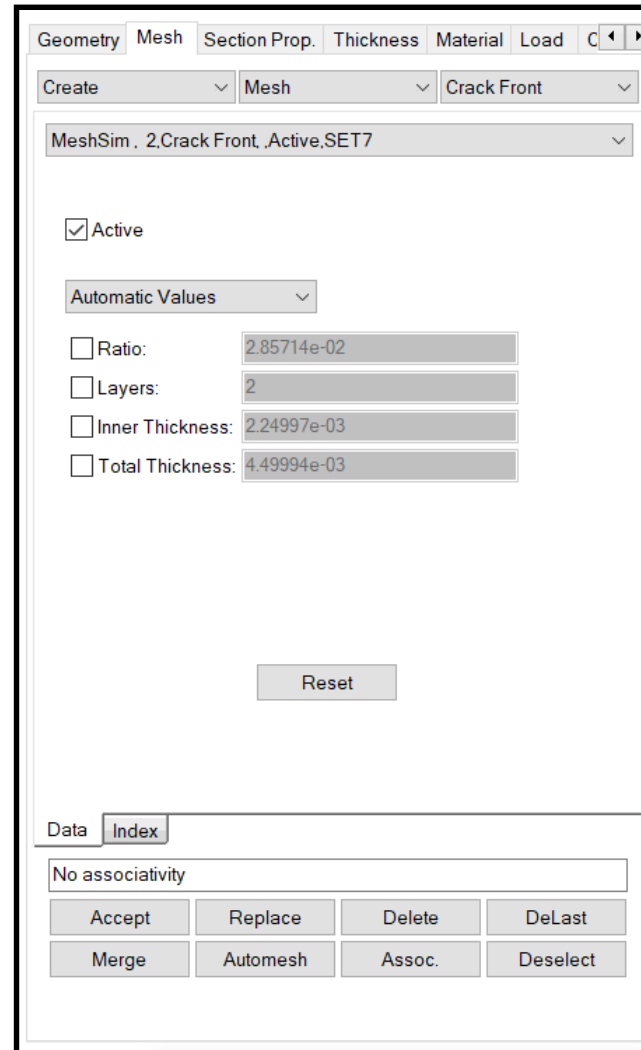
**NEW CRACK FRONT AUTOMESHING METHOD** <sup>TM</sup>

# New Crack Front Automeshing Method



STRESSCHECK®

- The Crack Face and Boundary Layer automeshing methods have been combined into a new and powerful method called Crack Front.
  - Identify crack front curve, and the crack surface is automatically detected.
  - Options for automatic computation of recommended boundary layer refinement, or inputs for user-defined boundary layer refinement.





**AUTOMATIC EXTRACTION RADIUS FOR  
FRACTURE MECHANICS PARAMETERS** <sup>TM</sup>

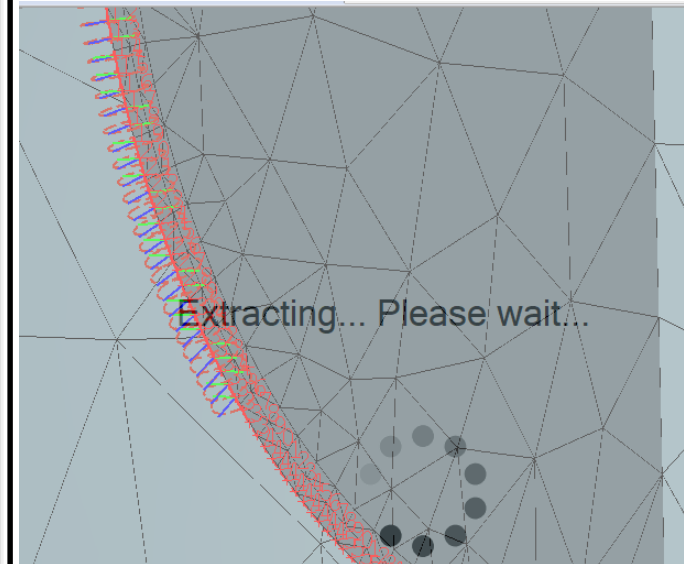
# Automatic Fracture Extraction Radius



STRESSCHECK

- There is a new feature for the automatic computation of the radius of integration for fracture mechanics parameters (SIF/J-integral).
  - If the Radius checkbox is not checked, and the field displays as AUTO, in accordance with best practices StressCheck will automatically compute the radius of integration to be just outside the innermost boundary layer.
  - If the Radius checkbox is checked, the user may provide his or her own custom radius of integration input.

The screenshot shows the 'Fracture' tab in the StressCheck software. The 'Radius' checkbox is checked, and the field next to it is set to 'AUTO'. The dialog box also shows other settings such as 'Func(s): K1, K2', '# of pts: 88', and 'Z-pln:'. The 'Boundary/System' section has 'P1-Min: 0.0', 'P2-Min: 0.0', 'P3-Min: 0.0', 'P1-Max: 1.0', 'P2-Max: 1.0', and 'P3-Max: 1.0'. The 'Location' section has 'X:', 'Y:', and 'Z:' fields. The 'Formula/System' section has 'Sys: SYS2'. The 'Input' section has 'Settings' and 'File' buttons. The 'Accept' and 'Cancel' buttons are at the bottom.





**ENHANCED SOLVER SETTINGS FOR MULTI-BODY CONTACT ANALYSIS**

# Multi-Body Contact Solver Settings



STRESSCHECK

- Additional controls for iteration limit, pressure tolerance and ray projection tolerance have been added to the Linear tab when active contact zones are detected.
  - Iteration limit
  - Pressure tolerance
  - Ray tolerance

The screenshot shows the 'Linear' tab of the solver settings. The 'Contact Settings' section is highlighted with a red box and contains the following values:

Setting	Value
Iteration Limit	40
Pressure Tolerance (%)	0.01
Ray Tolerance (length units)	10000



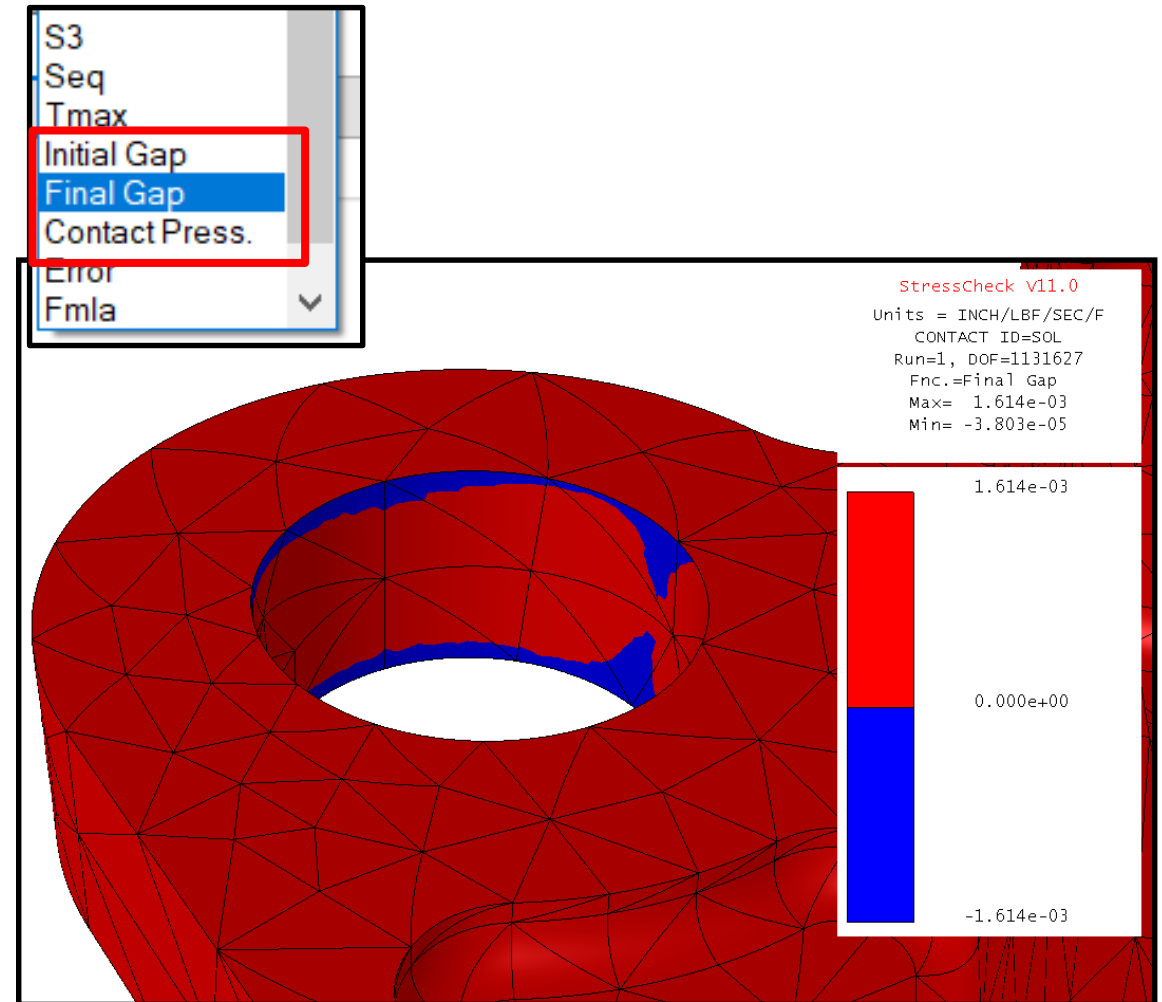
**ENHANCED EXTRACTION FUNCTIONS FOR  
MULTI-BODY CONTACT RESULTS**

# Multi-Body Contact Extraction Functions



STRESSCHECK

- Three new extraction functions are available for post-processing of multi-body contact solutions: initial gap measurement, final gap measurement, and contact pressure
  - The final gap measurement (Final Gap) is especially useful for detecting penetration of contacting parts.
  - Features are available in the Plot tab.







**ENHANCED LOAD/CONSTRAINT EXPERIENCE** <sup>TM</sup>

# Enhanced Load/Constraint Experience



STRESSCHECK

- Several improvements were made to load and constraint assignment management and editing.
  - Editable drop-downs for Load and Constraint ID's for convenient selection of existing ID's for assignment and checking.
  - Improved editing of Load and Constraint ID's to quickly visualize current assignments.

The screenshot displays the StressCheck software interface with several key components highlighted:

- Top Panel:** A menu bar with options: Geometry, Mesh, Section Prop., Thickness, Material, Load. Below it, a dropdown menu is set to 'Any Surface' and 'Traction'.
- Table:** A table with columns: Object, Method, Name, Data. The first row is 'Surface ,Traction ,LOAD1 ,SET1'. Below the table, there is a section for 'ID: LOAD3' with a dropdown menu showing 'LOAD1', 'LOAD2', and 'LOAD3' (highlighted in blue). To the right, 'Set: New set' is shown.
- Direction and System:** 'Direction: XYZ' and 'System: Global' are displayed.
- Buttons:** 'Force/Area' (selected) and 'Force' are at the bottom.
- Bottom Panel:** A 'Simulation' window with 'Input' and 'Solve' tabs. The 'Input' tab shows 'Object: Surface ,Traction ,LOAD3 ,SET3' and 'ID: LOAD3'. Below, there are checkboxes for 'Scale: 1.000e-02', 'Direction: XYZ', and 'System: Global'. At the bottom, there are checkboxes for 'X', 'Y', and 'Z' with corresponding numerical values.
- Left Panel:** An 'Edit Load Cases' dialog box with a table of load cases. The table has columns: Object, Method, ID, Data. The third row is selected: 'Surface Traction LOAD3 SET3'. Above the table are buttons for 'Delete', 'Purge ID', and 'Purge All'. Below the table are '< Previous' and 'Next >' buttons.
- Right Panel:** A 3D visualization of a red cube.
- Bottom Left:** A 'Message Log' window.



TM

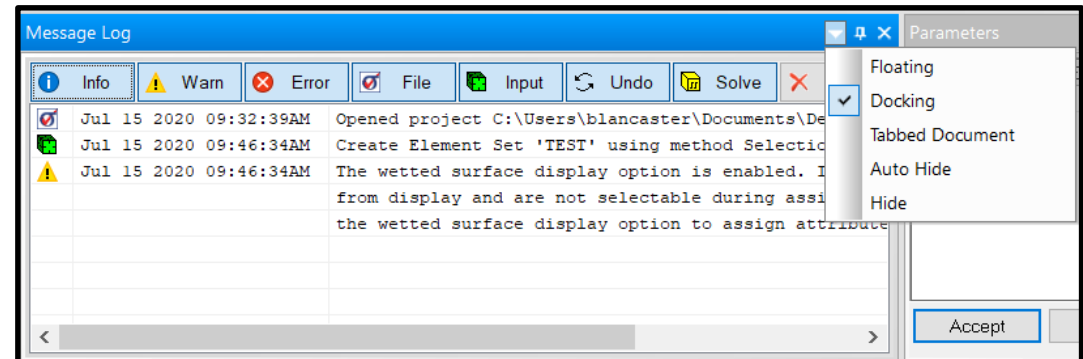
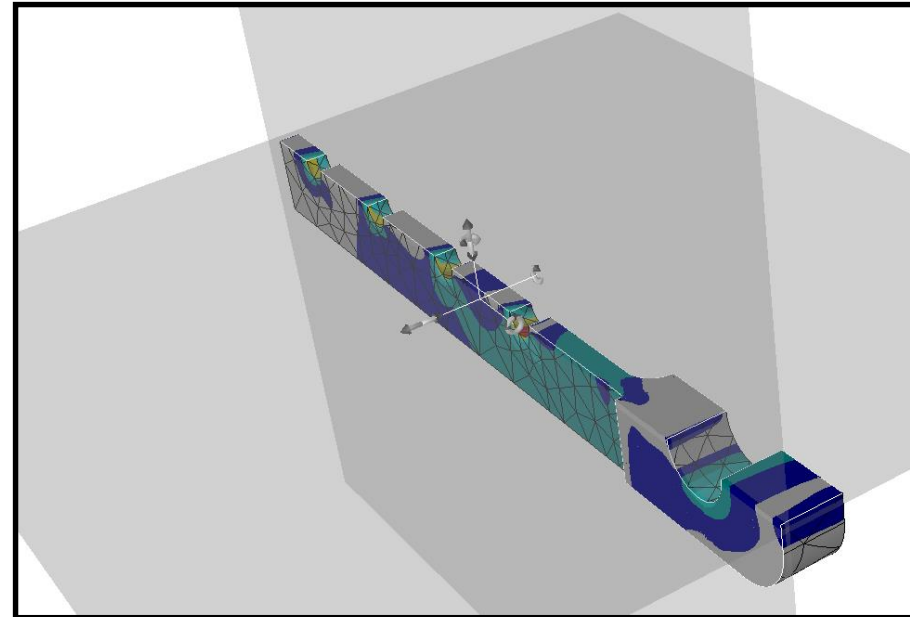
# **ADDITIONAL FEATURES AND ENHANCEMENTS**

# Additional Features and Enhancements



STRESSCHECK®

- ❑ Improved Cutting Plane performance for Contour Plots.
- ❑ New Force/Length option for Spring Coefficients.
- ❑ Improved Mesh Region assignment.
- ❑ New Error and Warning message handler.
- ❑ [Online documentation.](#)





**QUESTIONS OR COMMENTS?**

TM

Contact [support@esrd.com](mailto:support@esrd.com)