StressCheck v10.3 Training Outline

# Introduction to StressCheck Workshop

Day 1 Objectives:

1) Familiarization with the StressCheck (SC) graphic user interface (GUI).
2) SC file management available using I/O options. Procedures for saving and retrieving SC project files (*.scp) and SC work files (*.scw).
3) SC associative meshing and application of boundary conditions (pre-processing).
4) SC quality assurance procedures (pre- and post- processing).
5) Hands-on training:
   - Building 2D models (geometry, mesh, thickness, material properties, etc).
   - Applying boundary conditions (loads & constraints).
   - Creating solution settings and executing a linear analysis.
   - Performing post-processing operations to obtain results and verify quality.
   - Extruding 2D models into fully 3D models.

**Session I (4 hrs) Introduction to StressCheck, 2D (Planar) Modeling**

- ❑ Demo: 3D Step-Down Test Specimen with Live Results Processing
- ❑ Lecture: "Introduction to StressCheck: Strengths, Differentiators, and FEM Implementation" (Lecture Intro to StressCheck.pptx).
- ❑ Discussion: Discuss standard file I/O protocol (StressCheck projects) and how to perform the basic operations of starting and ending a StressCheck session.
- ❑ Discussion: Walk-through of the SC user interface with Handbook model, Eyebolt3D. Start SC and review the basic components of the GUI. Demonstrate parametric modelling, basic post-processing and AOM help.
- ❑ Exercise 1: Build and solve the 2D planar elasticity model "Rectangular Plate with a Central Hole" (pg 29-41). Save the project file as *TrainPlanar.scp*.
- ❑ Lecture: "StressCheck Post-Processing Adventures" (Lecture Post Processing.pptx).
- ❑ Exercise 2: Perform post-processing procedures for quality assurance: global error estimate, local convergence, contour plots, min/max, and point extraction (pg 41-48).

**Session II (4 hrs) Parametric Model Creation, Boundary Conditions, Extrusion**

- ❑ Discussion: Discuss parametric model creation and how to create parameters and settings.
- ❑ Exercise 3: Perform model-editing operations (pg 48-53). Use a parameter for the radius.
- ❑ Exercise 4: Start a new project, and construct the 2D planar elasticity model "Attachment Lug" with a hand built mesh (pg 53-55). Save work file as *LugWireFrame.scw* for later use. Perform point extraction around a hole to obtain location of maximum stress. Repeat the problem using the automesh option (pg 55-57).
- ❑ Lecture: "StressCheck 2D and 3D Boundary Conditions" (Lecture Boundary Conditions.pptx).

- Discussion: Explore the Extrusion option in StressCheck and learn how to properly constrain an extruded model (pg 58-59). Discuss rigid body constraints.
- Exercise 5: (Optional) Retrieve the *TrainPlanar.scp* project and switch reference from Planar to Extrude. Solve and post-process. Compare results with 2D model (pg 59-63).

Day 2 Objectives:

1) Solid Modelling and CAD Importation
2) Formula Definition and Usage
3) Automatic Meshing
4) Global-Local
5) Multi-Body Contact
6) Hands-on training:
   - Building 3D geometry using Boolean operations
   - Defining and using formulae
   - Automatic meshing parameters and their influence
   - Importing and automeshing 3D geometry for global-local analysis
   - Multi-body contact analysis overview and best practices

**Session III (4 hrs) 3D Solid Modeling, CAD Import/Export,**

- Discussion: "Introduction to StressCheck Modeling Terminology" (Lecture Solid Modeling.pptx). Object types, modelling operations, associativity, etc.
- Exercise 6: Start SC and build the "Rectangular Plate w/ Central Hole" model in 3D elasticity using the solid modeling option (pg 63-74). Compare hand & automesh results.
- Discussion: "Use of H-Discretization in Planar and 3D" (Optional, Lecture Use of H-Discretization in Planar and 3D.pptx)
- Exercise 7: Begin a new project and build the "3D Lug" model (pg 75-79). Save project file as *3DLug.scp*. Export Parasolid file as *3DLug.x_t*.
- Exercise 8: Import the *3DLug.x_t* file into StressCheck, and modify the geometry by Boolean-unioning a parametric cylinder to the solid (Exercise8.pptx). Save work file as *AutoMesh.scw*.

**Session IV (4 hrs) Formulae, Automeshing, Global-Local, Multi-Body Contact**

- Exercise 9: Create a parametric bending moment by formula on *3DLug.scp*. (Optional, Exercise9.pptx)
- Discussion: Discuss the influences of automatic meshing parameters on global and local refinement. (Lecture Automesh Parameters.pptx).
- Exercise 10: Open the *AutoMesh.scw* file and automesh body by using the default global mesh record. Study the influence of the D/H, Ratio and Transition parameters for this geometry. Optional: remove the Unioned cylinder by DeLast, and apply local mesh refinement (Curve and Boundary Layer) in the bore.
- Supplement: Explore global-local analysis best practices using imported point loads and automeshed 3D geometry. (3D Solid Part Analysis using StressCheck.pptx)

- Lecture: “Mechanical Contact/Fastened Connections” (Lecture Mechanical Contact.pptx, Optional, Lecture Fastened Connections.pptx).
- Discussion: Helpful Hints for optimization of contact problems (Lecture Optimization of Contact Problem.pptx).
- Supplement: Explore multi-body contact analysis best practices using an imported CAD assembly. (StressCheck 3D Contact Analysis with Knuckle Joint.pptx)
- Discussion: Geometric vs. Isoparametric mapping and when Geometric mapping is required (contact analysis) (Optional, Lecture Geometric vs. Isoparametric Mapping.pptx).