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

❑ Lecture: "Numerical Simulation by the FEM Using StressCheck, Part I" (Lecture Intro to FEM I.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. Start SC and review the basic components of the GUI. Demonstrate 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: "Post-Processing Hints and Tips" (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, Extrusion Modeling

❑ 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.

❑ Discussion: Discuss Edit vs. Move To with respect to objects (Lecture Edit vs Move To.pptx).

❑ 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: "Boundary Conditions Hints and Tips" (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: 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 Modeling Operations
2) CAD Importing and Modifications
3) Formula Definition and Usage
4) Automatic Meshing
5) Multi-Body Contact
6) Hands-on training:
   - Building 3D geometry using Boolean operations
   - Importing and exporting CAD geometry
   - Defining and using formulae
   - Automatic meshing parameters and their influence
   - Multi-body contact analysis overview

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

□ Discussion: "Introduction to Solid Modeling in StressCheck" (Lecture Solid Modeling.pptx).

□ 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" (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.*

□ Discussion: Importing and Exporting CAD in StressCheck.

□ 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, Automatic Meshing, Multi-Body Contact

□ Discussion: Formula Definition and Usage in StressCheck. (Creating a Formula Bending Moment.pptx)

□ Exercise 9: Create a parametric bending moment by formula on *3DLug.scp*. (Optional, Exercise9.pptx)

□ Discussion: Discuss the influences of automatic meshing parameters (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.

- ❑ Discussion: Geometric vs. Isoparametric mapping and when Geometric mapping is required (contact analysis) (Optional, Lecture Geometric vs. Isoparametric Mapping.pptx).

- ❑ Lecture: "Mechanical Contact/Fastened Connections" (Lecture Mechanical Contact.pptx, Lecture Fastened Connections.pptx).

- ❑ Exercise 11: 2D Multi-body contact problem: Plate with filled hole in tension (pp 79-89). Construct and solve this problem, ensuring proper load transfer and stress distributions. Optional: Use h-Discretization to increase number of elements in plate.

- ❑ Discussion: Helpful Hints for optimization of contact problems (Optional, Lecture Optimization of Contact Problem.pptx).

- ❑ Exercise 12: 3D Multi-body contact problem: Lug-Clevis-Pin (pp 89-97). Load, modify and solve this problem and perform quality assurance procedures.