

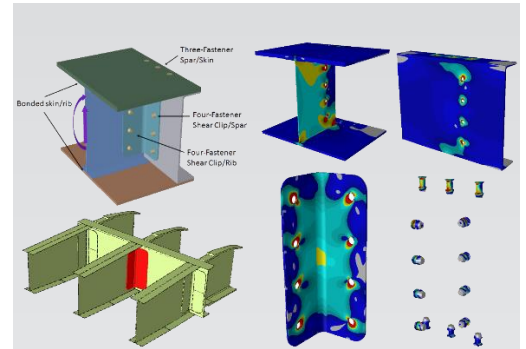


The Premier High Definition Structural Simulation Software

StressCheck is the only finite element software product that was designed to support solution verification. Users can obtain estimates of the error of approximation for any output without re-meshing, making the comparison of model predictions to experimental observations simple and “stress”-free.

Through the use of hierarchic finite element spaces and models, StressCheck provides the means for efficient control of two types of errors: the discretization (approximation) errors and model form (epistemic) errors. As such, there is no need for a finite element library; elements are only needed to represent the shape of the associated CAD geometry, and are independent of the analysis type or theory. The theoretical basis of the algorithms and methods implemented in StressCheck has been thoroughly documented in the technical literature and in textbooks¹.

StressCheck is being extensively used for strength, durability and damage tolerance applications, primarily in the aerospace and defense industries. StressCheck supports Smart Simulation Applications and standard processes. Those applications are expert-designed with built-in safeguards, so that engineers can use them safely and efficiently. Smart Simulation Applications increase productivity and preserve corporate know-how via customizable frameworks.



“StressCheck’s quality control features have given us the tool we need to perform detailed ... bonded joint analysis with confidence that the results are accurate.”

“...StressCheck has greatly improved our ability to accurately predict the fatigue life of joints with interference fit fasteners and cold worked holes”

The Boeing Company



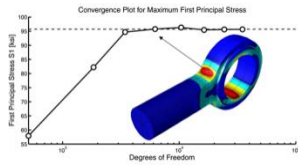
“This software tool...includes an FEM-based handbook format which allows non-experts to utilize models prepared by specialists.”

Lockheed Martin

¹B. Szabó and I. Babuška. Finite Element Analysis: Method, Verification and Validation. 2nd Edition. John Wiley & Sons Inc., Hoboken NJ 2021.

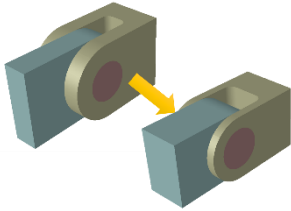


In StressCheck the mathematical model and its numerical approximation are treated separately. This is an essential technical requirement in numerical simulation. In contrast, legacy finite element tools entangle models and their approximations in finite element libraries that were designed to support the obsolete practice of finite element modeling rooted in the 1960s.



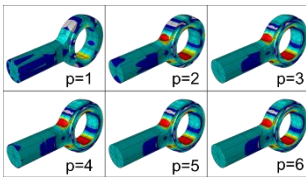
AUTOMATIC SOLUTION VERIFICATION

StressCheck’s powerful FEA implementation allows users to increase the polynomial order of element displacement functions (p-level) automatically and hierarchically, thus increasing solution quality without the need for successive mesh refinements. When three or more increasing p-levels are solved, you may perform simple and efficient convergence assessments for any engineering function at any location in the model.



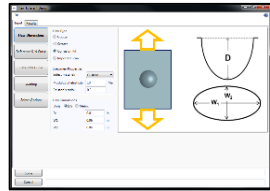
PARAMETRIC MODELING FRAMEWORK

In StressCheck, any attribute of the model (geometry, material properties, loads, constraints, etc.) may be defined parametrically (with variables), and in many cases formulae expressions may be used during pre-processing or results processing. With this framework in place, standardization of design and analysis processes may be achieved. In addition, StressCheck supports native importation of Parasolid transmit files, and the translation of CATIA, STEP, Creo and IGES formats.



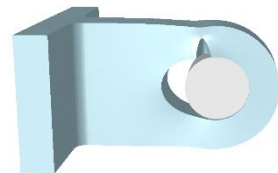
“LIVE” RESULTS PROCESSING

StressCheck has long been known for its superior results processing. A significant advantage is that all extraction results are computed “on the fly” anywhere in the model - you do not have to anticipate where the critical areas will be, nor do you have to predefine any solution output. With StressCheck, you have at your disposal a wide variety of extraction methods that give you virtually every engineering result anywhere in the model- and all come with built-in assessment of accuracy.



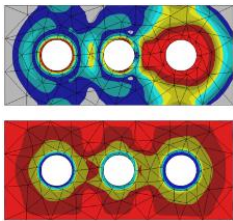
SMART SIMULATION APPLICATIONS

Develop custom Smart Simulation Applications (Sim Apps) tailored for your organization via StressCheck’s powerful and flexible application programming interface (API). From simple Excel VBA/Python scripts designed to automate repetitive tasks and processes, to .NET applications with WPF front ends targeting engineering professionals with no FEA experience or background, the possibilities with StressCheck’s API are endless.



FRACTURE MECHANICS

The unique features in its Fracture Mechanics module clearly set StressCheck apart from other FEA products. You have powerful automatic meshing of embedded cracks combined with superconvergent point-n-click extraction methods to calculate stress intensity factors or energy release rates at any location along the crack front – all with automatic convergence reporting.



COLD WORKED HOLES

Cold worked hole analysis capabilities were developed in response to demand by the aerospace industry to replace existing time consuming and error prone techniques based on traditional methods that are unable to account for the variety of situations that appear in real structural components. With StressCheck you can accurately predict residual stresses due to cold working of complex parts.

Additional features include (but are not limited to): material/geometric nonlinear, modal/buckling, steady-state conduction heat transfer, multi-body contact and laminated composites. For more information on our applications, features and pricing, contact sales@esrd.com.

www.esrd.com



314-744-5021