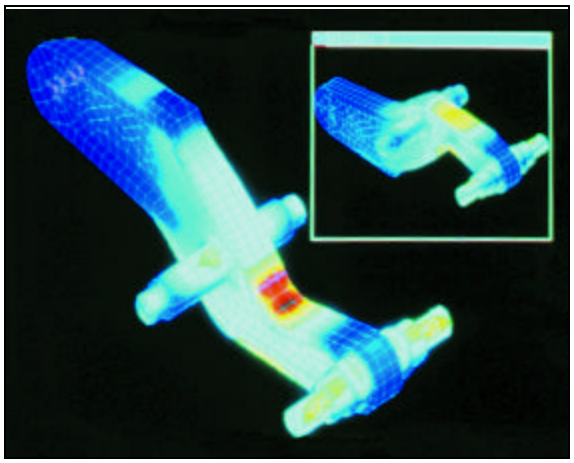
STAR



Spring-assisted seat suspension under maximum load condition (courtesy of Milsco Manufacturing Co.)

STAR is COSMOS/M's basic linear static analysis module with state-of-the-art analysis and solver capabilities. Expert FEA users and novices alike can use the extensive capabilities of this module to perform stress analysis of simple models or complex 3D assemblies. The program produces displacements, strains, stresses, forces, and error estimates as results of the analysis under a variety of loading conditions. STAR offers a fully integrated pre- and postprocessor, allowing you to assess how your designs will perform in their operating environments, quickly and affordably.

Easy and Flexible Linear Static Analysis

With STAR, you can address 2D and 3D linear static problems. The program's large element library and its many other modeling features allow you to set up your designs for quick and accurate analyses. Among the extensive capabilities of STAR is the support of various types of constraint equations, and bonding of surfaces with incompatible meshes. STAR also supports other features such as contact, geometric stiffening, P-type elements and adaptivity, asymmetric loading, and much more.

To solve problems quickly and efficiently, STAR utilizes two new robust solvers. The Direct Sparse solver handles large problems very accurately in a fraction of the time needed for the standard direct solution procedures. The new Iterative solver is capable of solving problems of several million degrees of freedom with superior performance and with minimum computing resources.

STAR can be used as a stand-alone system with GEOSTAR, COSMOS/M's powerful pre- and postprocessor, or in combination with any other COSMOS/M modules to give you an affordable solution.

Overall, the combination of capabilities offered by STAR gives you an economical and powerful tool to test your designs in all the operating environments they will encounter.

SPECIFICATIONS

STAR

DESCRIPTION

STAR performs 2D and 3D linear static analysis.

USER INTERFACE

- On-line help and pull down menus
- Command icons and multiple windows
- Element, material and section libraries
- Data checking of material and geometric properties, element connectivity

ANALYSIS MODELING

- Parametric input for material properties and loads
- Stress stiffening and softening effects
- Cyclic symmetry, substructuring
- Analysis coupling with COSMOS/M thermal, electromagnetic, fluid flow, fatigue, advanced dynamics modules and FFE Thermal
- Submodeling
- ASME stress evaluation
- J-Integral evaluation
- Inertia relief
- Contact problems and friction

ELEMENT LIBRARY

- 2D and 3D P- elements
- Isoparametric 2D elements (plane stress, plane strain, and axisymmetric)
- Trusses (Bars) and Beams
- Rigid Bars
- Curved isoparametric plates & shells
- Shells (thin and thick)
- Elastic pipes
- Isoparametric solids
- General stiffness
- Gap/contact
- Spring and boundary
- Masses
- Composite: shells and solids with interlaminar shear
- Fluid (fluid structure interaction)
- Crack

MATERIAL PROPERTIES

- Types include: isotropic, orthotropic, anisotropic and layered composites
- Temperature dependent material properties and failure theories for statics
- Layered composite shells

BOUNDARY CONDITIONS

- Prescribed displacements and rotations
- Automatic rigid connection between solids and shell elements
- General multipoint constraints
- Constraint equation capability
- Constraint of different geometric entities together by bonding

LOADS

- Multiple mechanical and thermal load cases
- Types include: concentrated, pressure, beam loading, temperatures, acceleration, centrifugal and asymmetric (Forces and pressures can be applied in any coordinate systems)

RESULTS

OUTPUT VALUES

- Displacements and rotations
- Stresses: normal, shear, principal, von Mises, stress intensity and energy error norm
- Strains: normal, shear, and equivalent
- Total strain energy and density
- Reaction forces
- Beam stress resultants
- Mass, moment of inertia and center of gravity
- Foundation pressure
- Failure index
- Grid force balance
- Fracture Mechanic's j-integral
- Estimated error in displacements when direct solvers (skyline or sparse) are used

DISPLAYS

- Hidden line and shaded image views
- Types: stress, strain, deformed shape, displacement, axial force, shear force, torsional, bending moments, energy, failure index and energy error norm
- Graphics: color-filled and color line contour, vector, isoplane, section plots, force and moment diagrams and X-Y plots
- Animation: stress, and deformation, Web-ready AVI file formats
- Convergence plots: number of iterations vs. energy error norm, maximum von Mises stress, maximum resultant displacement,percentage of error or DOF
- Tabular data reports

NUMERICAL TECHNIQUES

- Skyline column solver
- Direct sparse matrix solver
- Iterative PCG solver
- FFE solver

SYSTEM REQUIREMENTS

- Windows 9X or NT systems (for Unix platforms, please inquire)
- 64 MB RAM minimum, 128 MB RAM recommended
- 200 MB disk space minimum
- CD-ROM drive

OPTIONS

- Maintenance: hotline support, program updates and QA reports
- Training: Introductory and Advanced
- FEA Translators: NASTRAN®, ANSYS®, PATRAN®, I-DEAS® and SINDA®
- COSMOS/M Database Utility for accessing the COSMOS/M database
- AISC code checking
- Windows NT & EWS network option

COSMOS/M

STRUCTURAL RESEARCH & ANALYSIS CORP.
Developers of COSMOS/M Applications
12121 Wilshire Blvd.
7th Fl., Los Angeles, CA 90025
310 207-2800 Fax: 310 207-2774
www.cosmosm.com