

Information about finite element (FE) program package FEMAP with Nastran

FEMAP with Nastran is a general-purpose software package, which uses Finite Element Method (FEM) to evaluate mechanical design strength ability under different loading conditions.

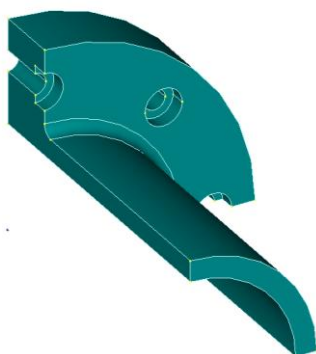
The package consists of two FE programs:

- **FEMAP** for model definition and evaluation
- **Nastran Solver** for FEA solution.

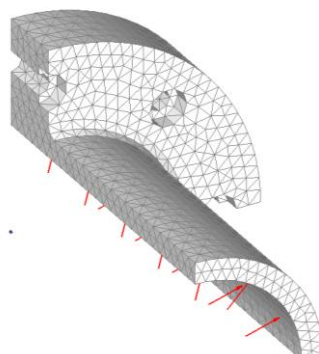
FEMAP program uses for FE model definition and results evaluation. The program creates model geometry by itself or, to save the modeling time, imports it from different CAD systems (e.g. AutoCAD, Mechanical Desktop, Inventor, NX, Solid Edge, Solid Works, CATIA, Pro/E and I-DEAS). Due to **FEMAP** flexibility the program can be used as a converter of finite element models between different FE applications like: NEiNastran, NX Nastran, ANSYS, ABAQUS, COSMOS, MSC.Nastran and other.

Nastran Solver is a general purpose FEA solver, which uses during mechanical design development and manufacturing. NEiNastran as well as other latest Nastrans is based on the popular NASTRAN software but many modern innovations make NEiNastran very easy and user friendly. Program is written from scratch using FORTRAN90 and C/C++ languages for Windows NT and PC environment. NEiNastran **Engine** has interface to among others FEMAP and PATRAN.

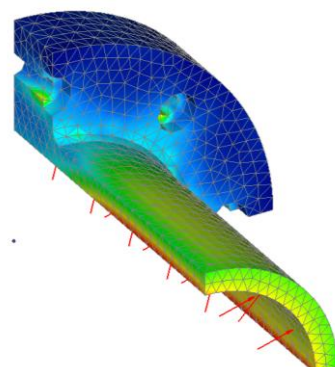
Nastran solves linear and non-linear, static and dynamics, steady state and transient heat transfer analyses. According our customers opinion NEiNastran is the best Nastran on the FEA market today. Very easy to understand and use due to very clear structure and documentation with user-friendly analysis Editor.



Model Geometry



FEA Model



Solution Results