

NEi Nastran and Creo Parametric

Providing Reliable Answers for Complex Product Development Questions

How it Works

STEP 1 - CAD Model Import

The Creo (formerly known as Pro/ENGINEER) model is opened in the Pro/MESH application mode.

STEP 2 - Creating the FEA Model

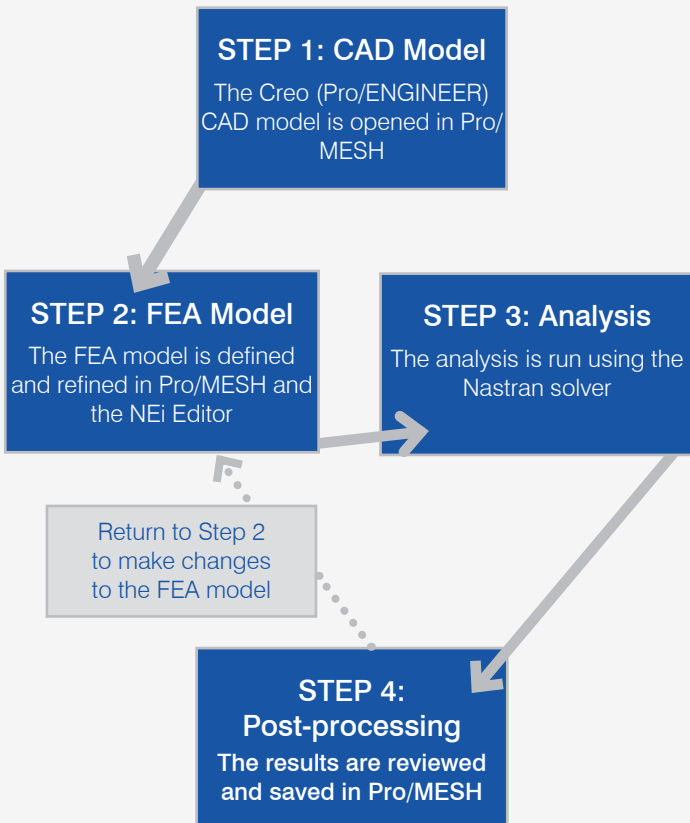
The engineer defines and refines the FEA model in the Pro/MESH environment. For advanced analysis, the engineer can take advantage of the additional capabilities offered by the NEi Editor.

STEP 3 - Running the FEA Analysis

The analysis is run using the Nastran solver.

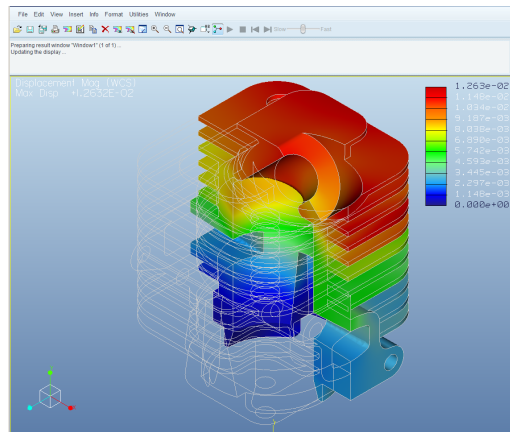
STEP 4 - Viewing and Exporting the Results

Post-processing is accomplished in the Pro/MESH environment.



Benefits of Nastran + Creo

- Powerful Creo (Pro/ENGINEER) CAD modeling and import
- Easy access to Nastran FEA technology
- Ability to provide industry-standard Nastran models and results to partners, customers, and vendors



Benefits of Finite Element Analysis (FEA)

Finite Element Analysis takes a complex problem and breaks it down into a finite number of simple problems. Finite element analysis approximates the behavior of a continuous structure by analyzing a finite number of simple problems known as elements. The finer the mesh, the closer we can approximate the geometry of the structure, the load application, as well as the stress and strain gradients. The strategy of optimizing the mesh size can greatly reduce an analyst's time without compromising on the quality of analysis results.

Some FEA benefits include:

- Reduces the number of prototypes needed
- Increases the number of parameters that can be tested including types of materials, thickness, shape and weight
- Reduces testing time and time-to-market

Interested in learning more?

Contact us:
info@neissoftware.com
<http://www.nenastran.com/Creo>

