Femap Version 10.2 What's New

October 2010

SIEMENS

Femap 10.2

What's New Overview

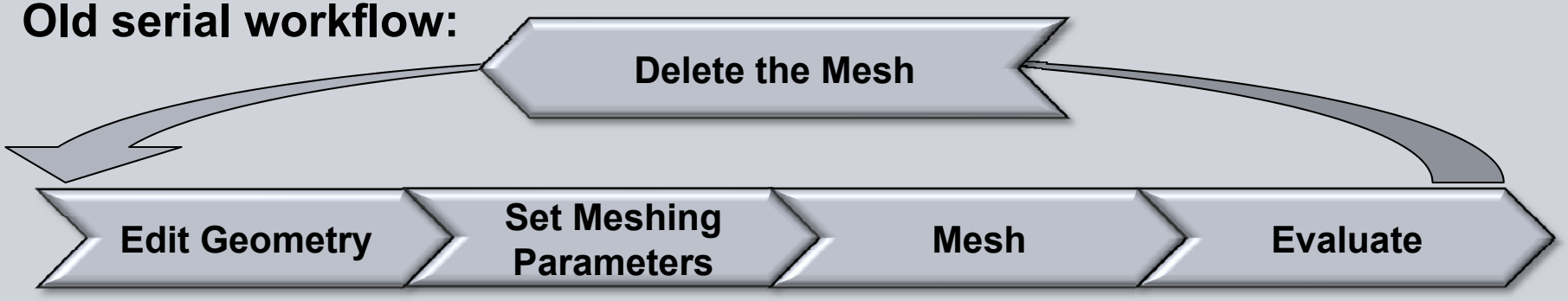
- Meshing Toolbox
- Postprocessing
- Topology Optimization
- NX Nastran 7.1
- General Nastran
- Customer Driven Updates
- Thermal and Flow Solver Updates

Meshing Toolbox

More Efficient Interactive Workflow

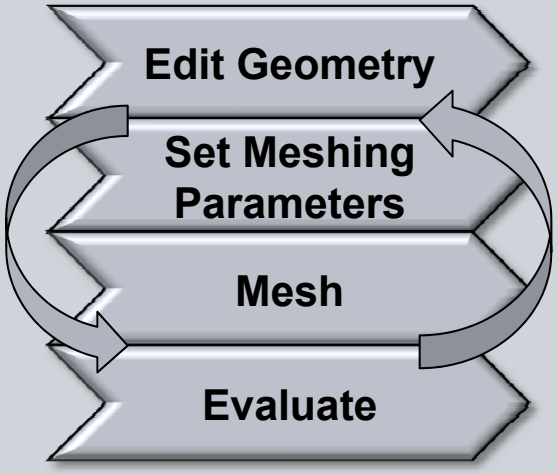
SIEMENS

Old serial workflow:



New interactive workflow:

- Interoperable tools – more efficient modeling and meshing process
- Immediate visual feedback on mesh and model quality
- Improves productivity
- Generates models faster
- Creates more accurate models



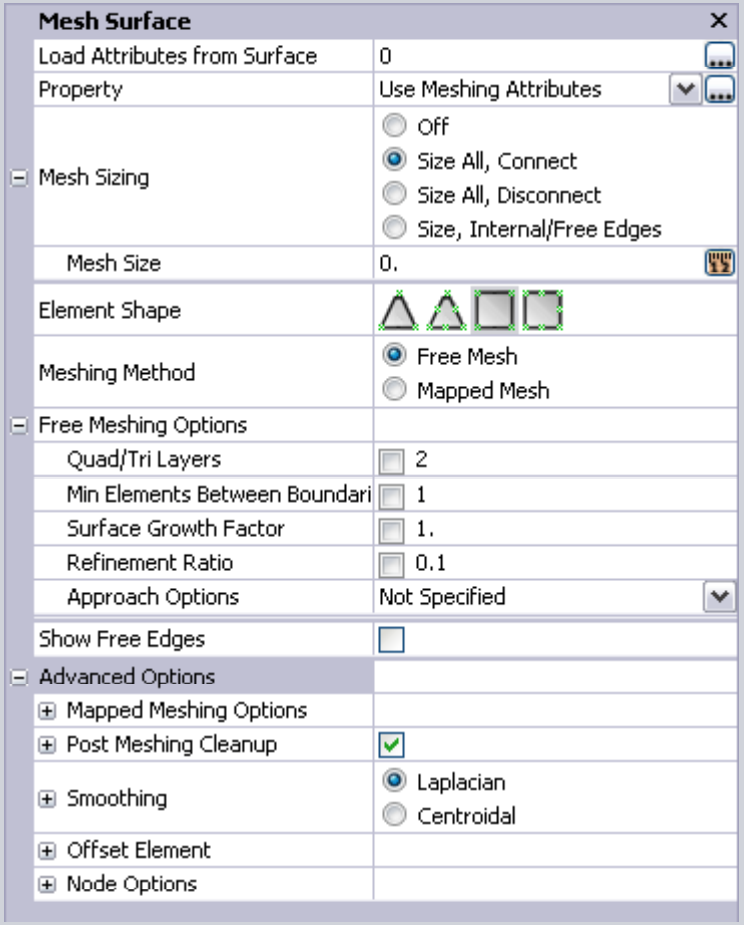
Meshing Toolbox

Mesh Surface Toolbox

SIEMENS

New Mesh Surface Toolbox

- Used for Shell Meshing only
- Will apply Mesh Sizing, and mesh attributes on the fly then mesh the surfaces
- Method for automatically applying 3 Corner / 4 Corner mapped meshing attributes based on the surface geometry
- Surface meshing options such as Quad Layers around boundary and number of elements between boundaries can also be set



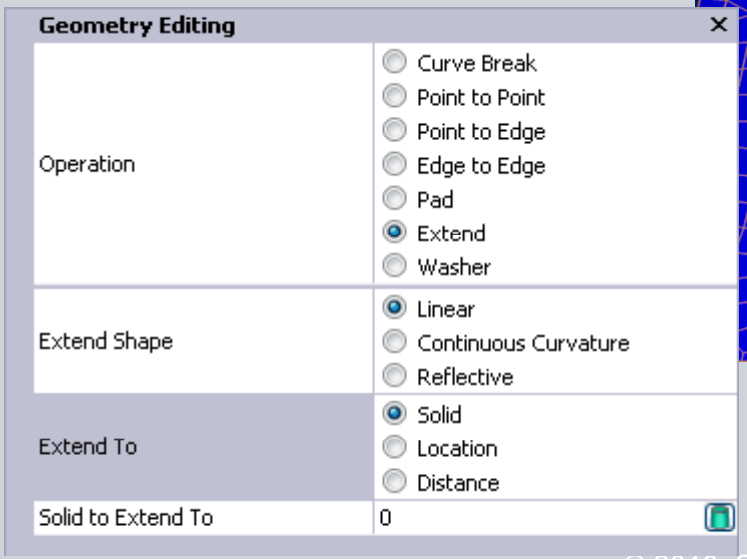
Meshing Toolbox

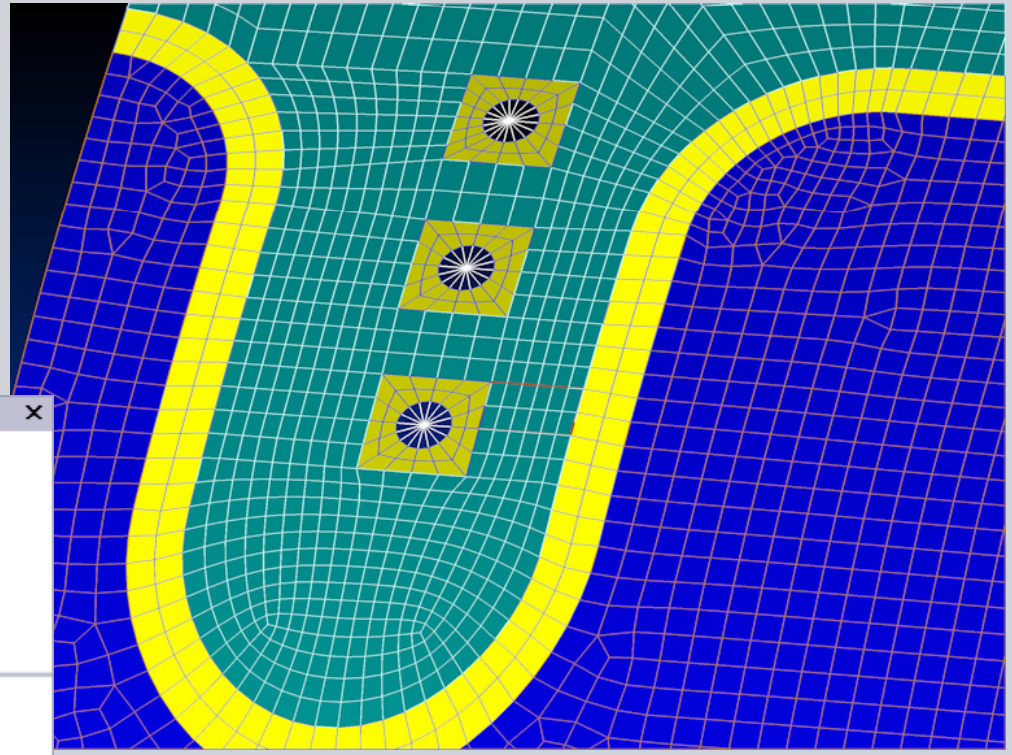
Geometry Editing Toolbox

SIEMENS

Geometry Editing Toolbox

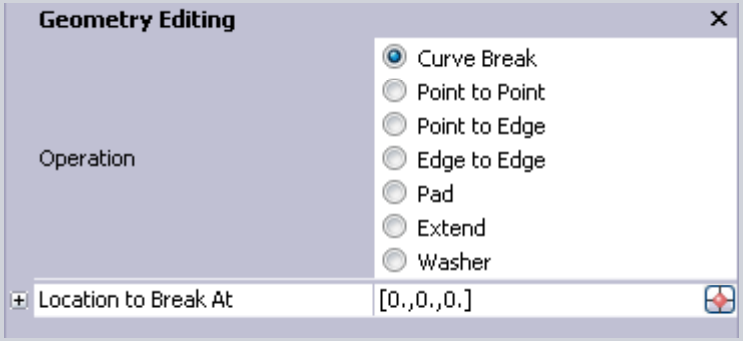
- Point to Point
- Point to Edge
- Edge to Edge
- Pad
- Washer
- Extend Surface



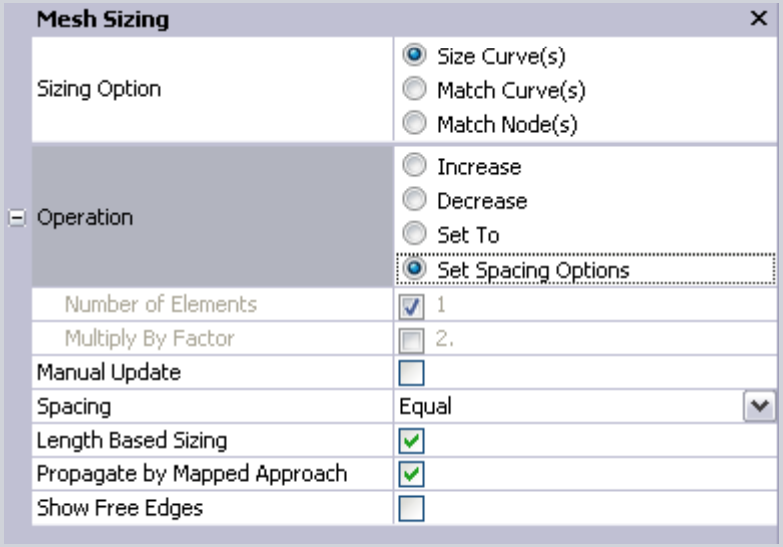


Meshing Toolbox

Geometry Editing
- Curve Break



Mesh Sizing toolbox additions
- Match nodes
- Match multi-curves
- Increase or decrease node count by factor in addition to a number



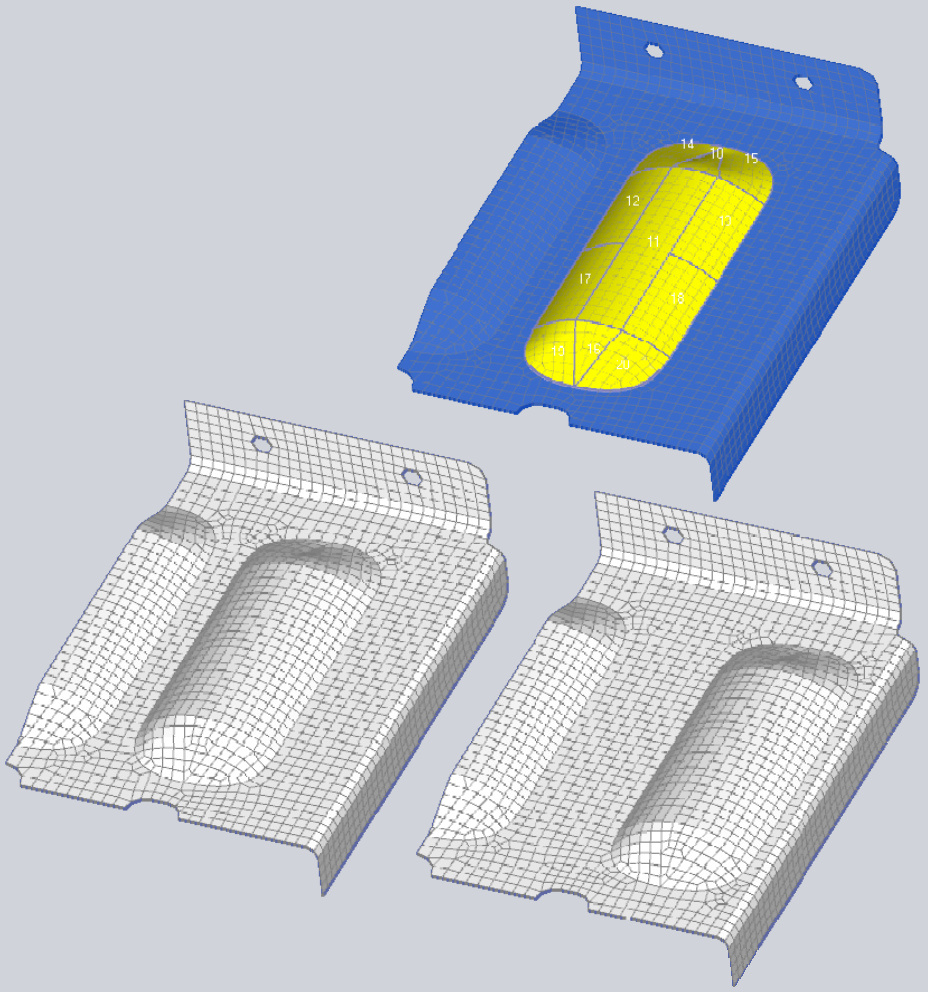
Meshing Toolbox
Feature Editing

SIEMENS

Move and Rotate Surfaces to update Geometry and Mesh

- Selected Surfaces can be moved or rotated
- Mesh is updated automatically
- Overall topology must be maintained
- In this example, a group of 9 surfaces is translated

- Note – this is not Synchronous Technology – user chooses the surfaces to be moved

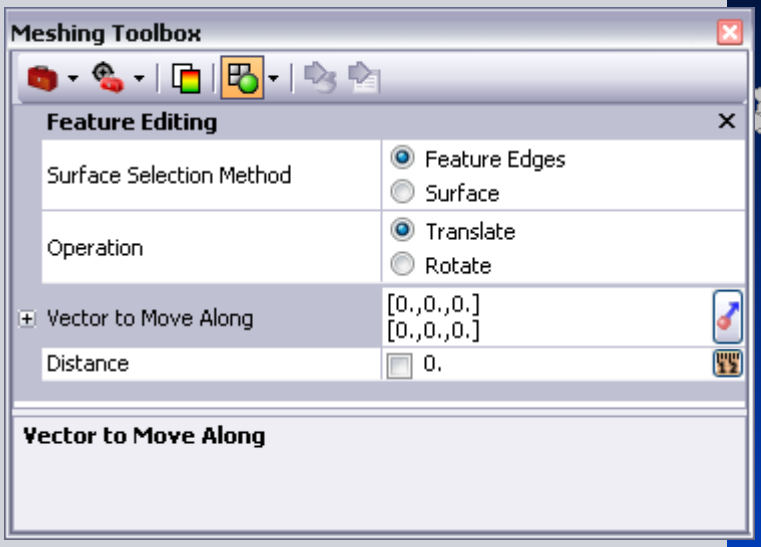


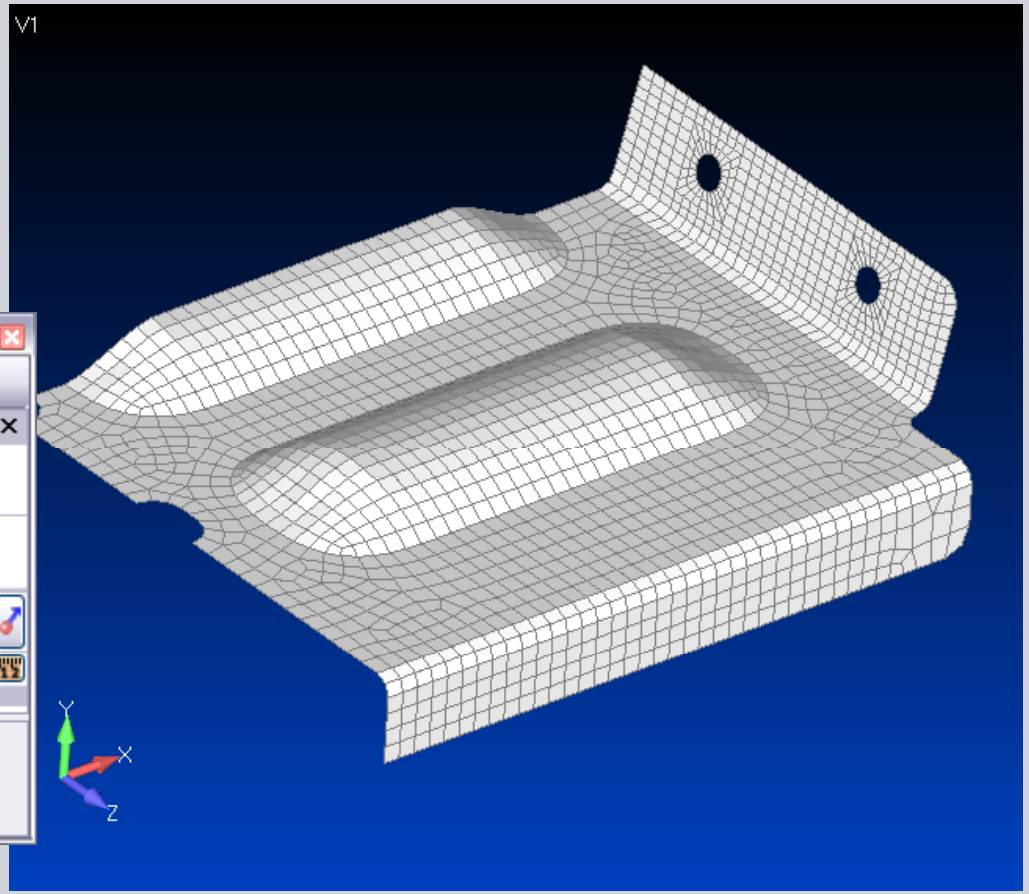
Meshing Toolbox
Feature Editing

SIEMENS

On the same model:

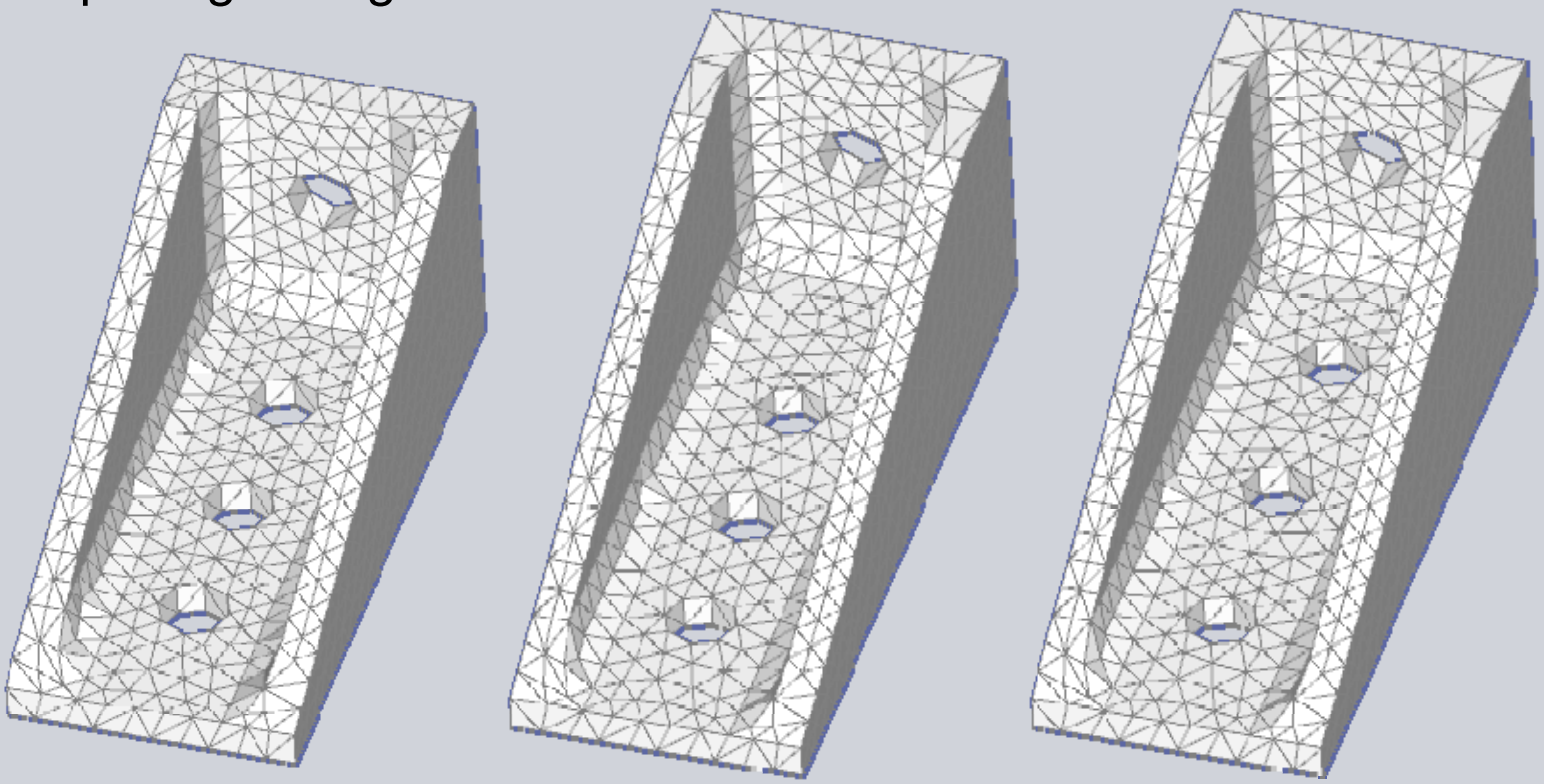
- One flange is translated
- The other flange is rotated through15 degrees





Solids example

- Part stretched
- Hole spacing changed



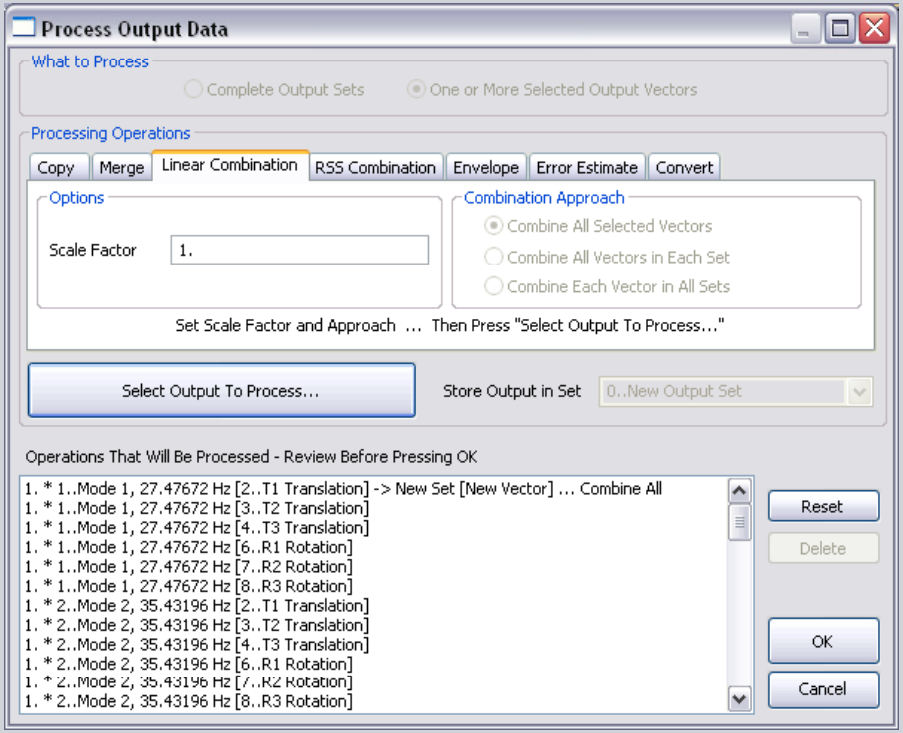
SIEMENS

Processing of Output Data

Complete overhaul of the Output Processing Function

- New user-interface
- Expanded options
- Significant performance improvement

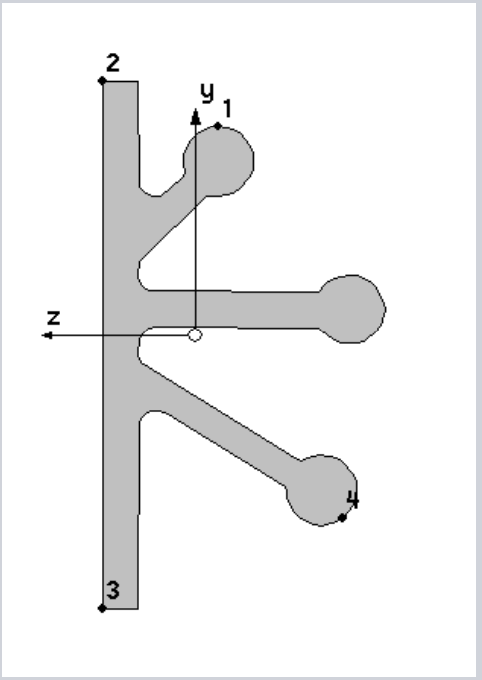
- Considerable speed increase in Model – Output – Process
  - up to 100x faster
- New Model Output Delete functionality
- Algorithm improvement with Undo
- Instantaneous without Undo

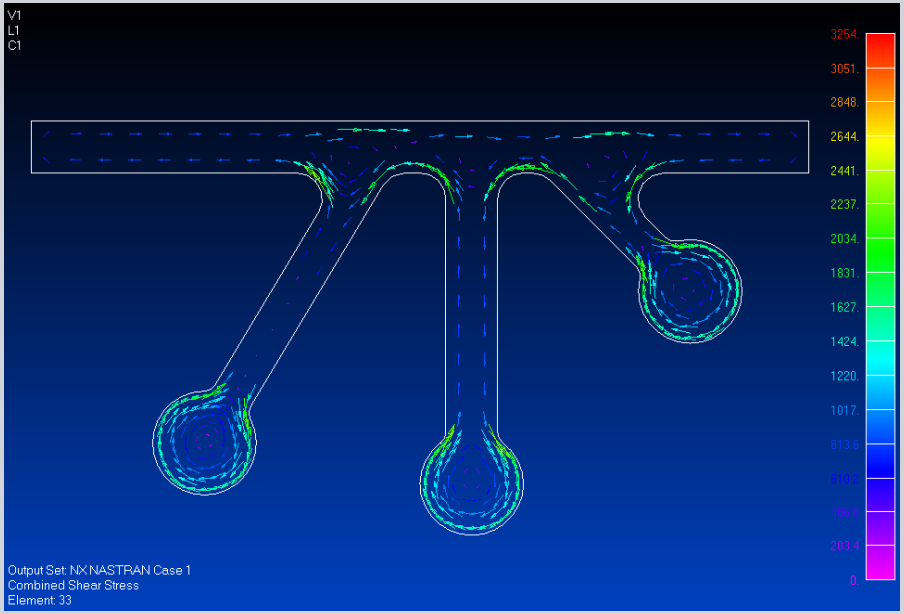


SIEMENS

Beam Postprocessing

- Nastran provides four stress recovery locations – fully configurable in Femap
- Drawback, the highest stress, as well as details about shear distribution, cannot be readily determined or visualized and the points must be defined ahead of the analysis

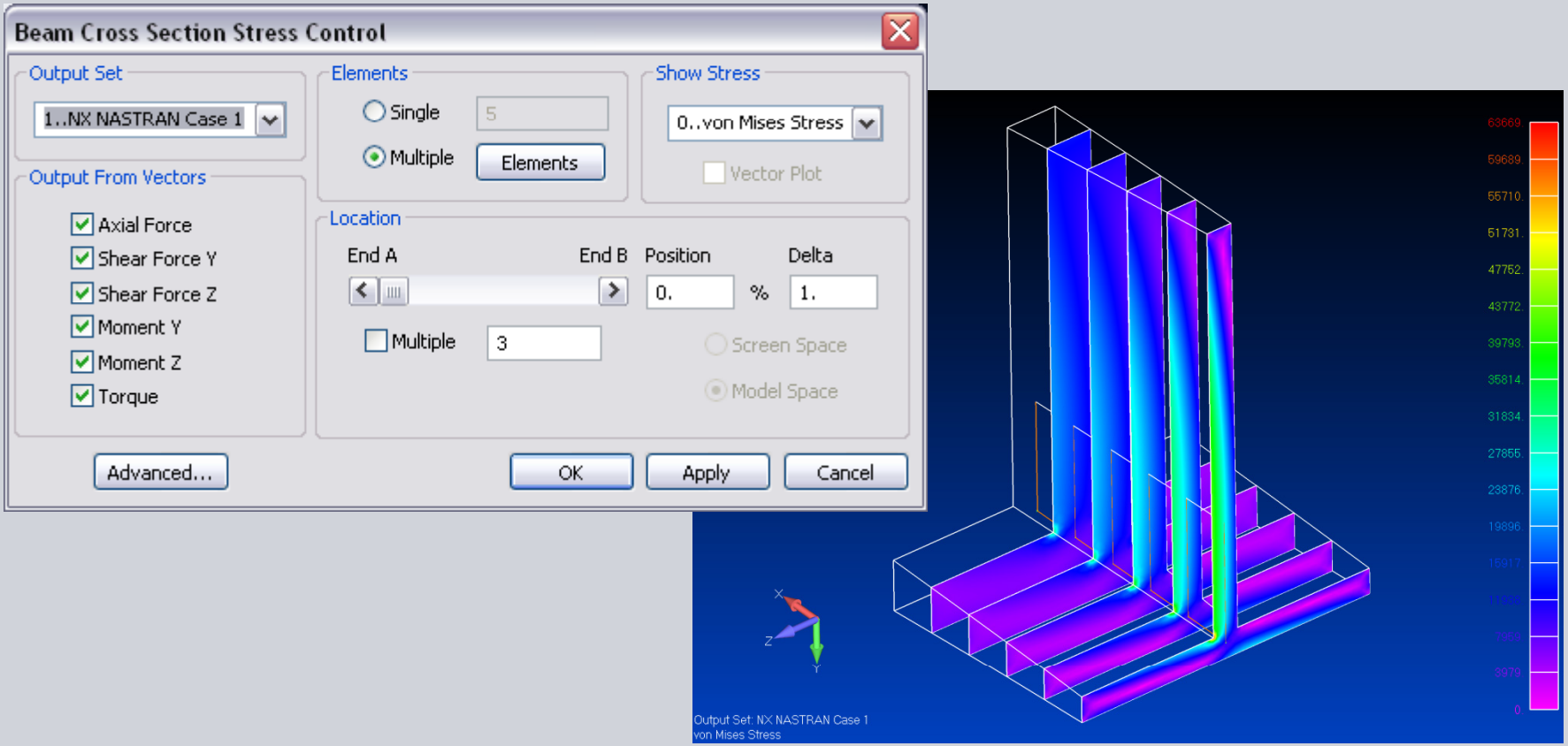




SIEMENS

Beam Postprocessing

Using forces recovered from any FEA solver – axial, shear, bending

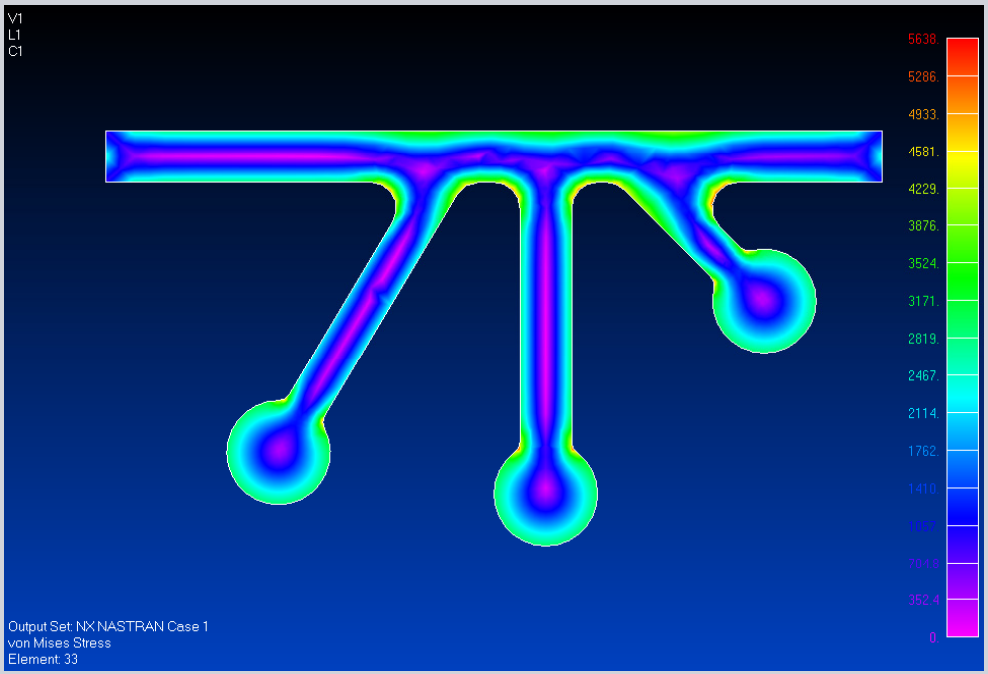


SIEMENS

Beam Postprocessing

Beam postprocessing flexibility

- Calculate stresses such as von Mises
- Visualize complete distribution of stress
- Visualize shear flows
- Analyst control over force components used to calculate stress, i.e. visualize stresses due to axial loading only, bending only, shear only, etc.

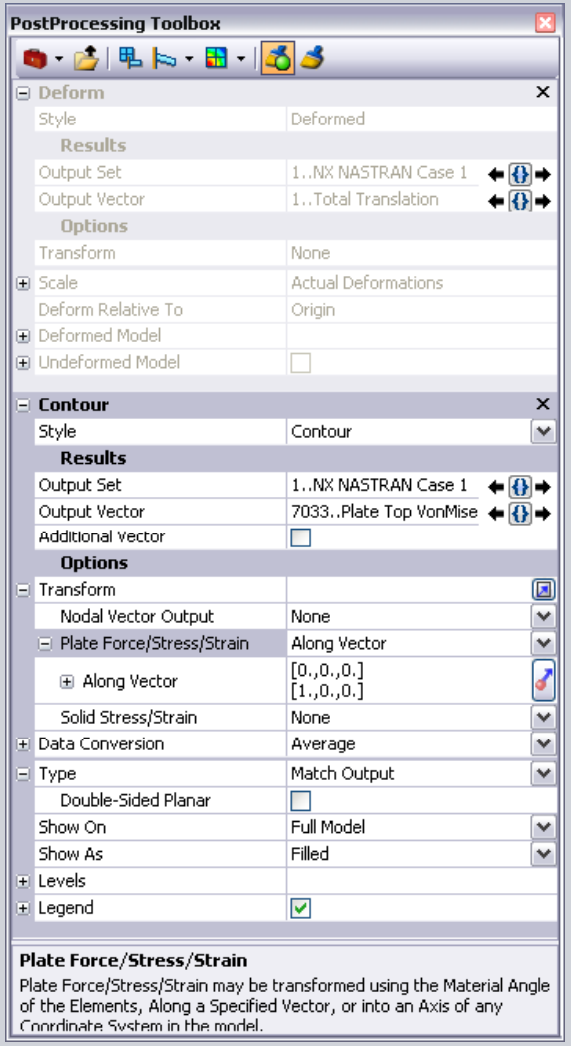


SIEMENS

Postprocessing Toolbox

Consolidated access to the most popular postprocessing View Options in a single Femap toolbox

- Quickly change postprocessing options interactively
- Reduce training and learning time by providing high-level, highly visible access to detailed postprocessing options



Topology Optimization

Femap with NX Nastran Add-On Module

SIEMENS

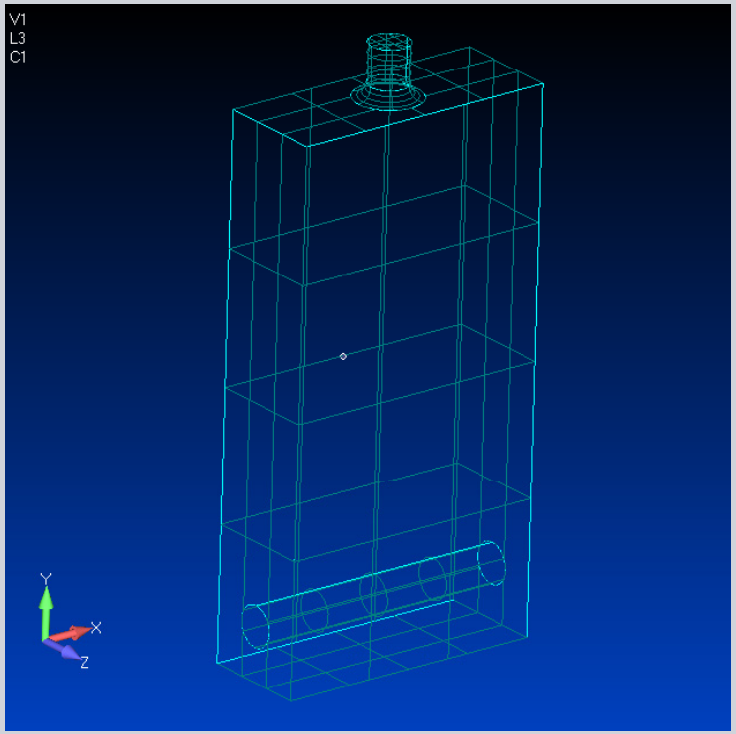
Add-on module on top of Femap with NX Nastran

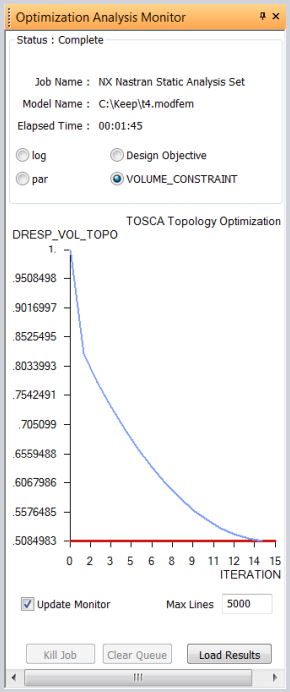
- Mass reduction
- Maximize and tunes stiffness
- Minimize weight
- Based on TOSCA Structure by FE DESIGN, Germany

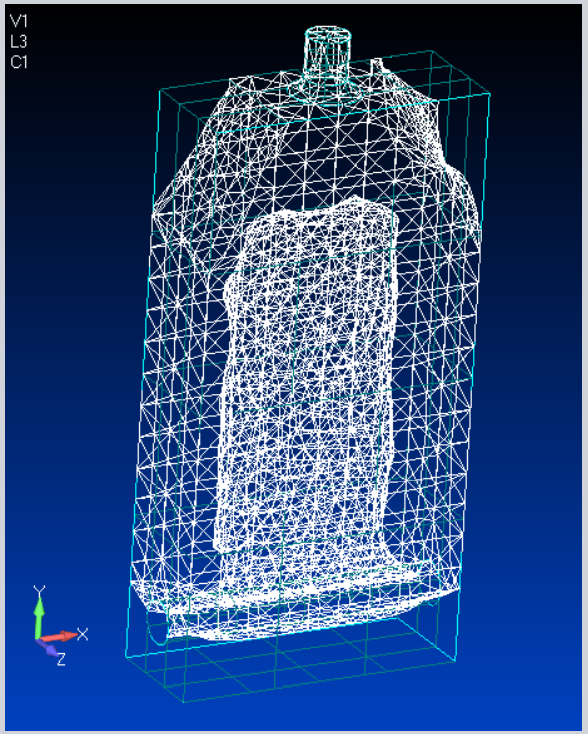
Topology Optimization
Torque Link Example

SIEMENS

Basic geometry run through topology optimization. Axial Loading, +/-Y Torque Loading





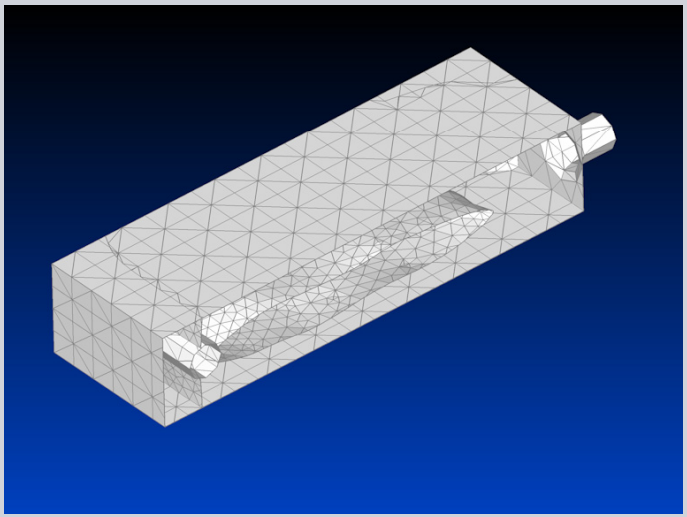


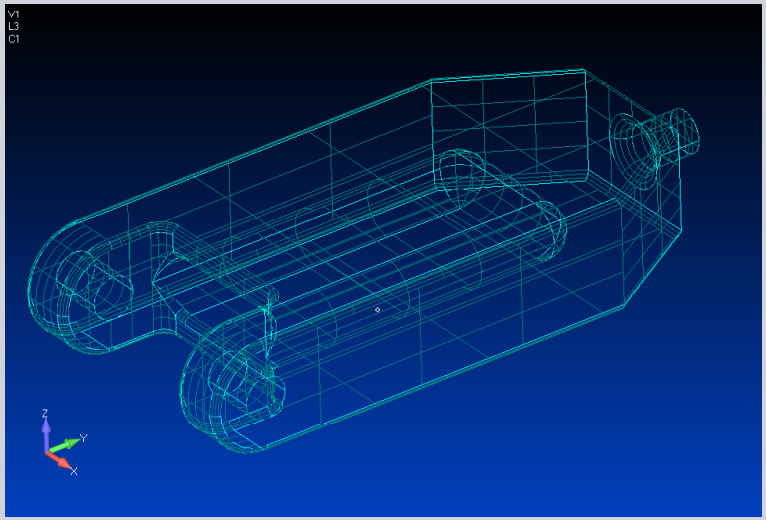
Topology Optimization
Torque Link Example

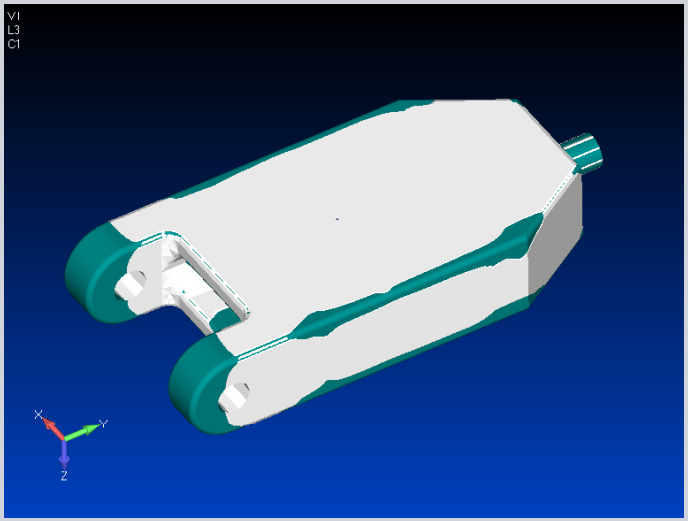
SIEMENS

Using this information, the design is updated as shown

- Initial Weight: 27.8 pounds
- First Pass: 18.5 pounds
- Another run on the optimized design shows where some more material can still be removed







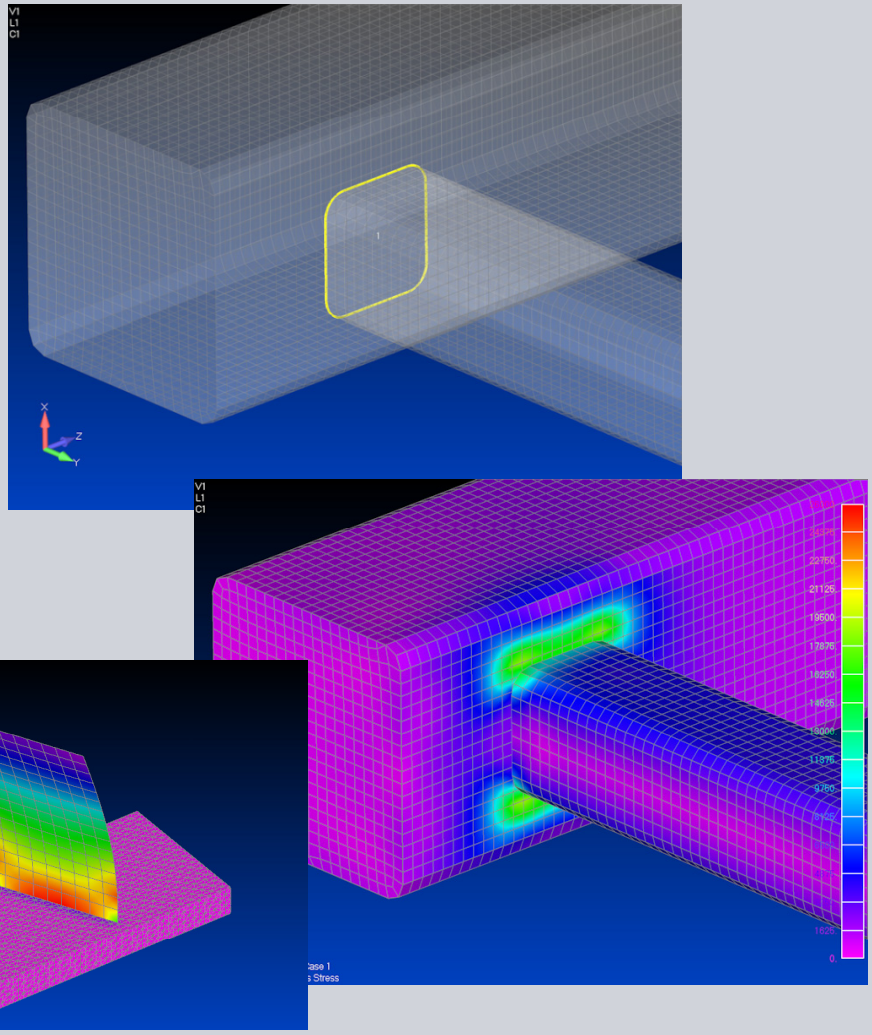
NX Nastran 7.1

Edge to Face Glue

SIEMENS

Edge to Face Glued Connection

- Connect edges of shells to faces of other shells or solid models
- Auto Connection and Setup just like solid connections in Femap
- Stiffness of the connection can be tuned for dynamic model correlation
- Handles moment transfer between shell and solid elements



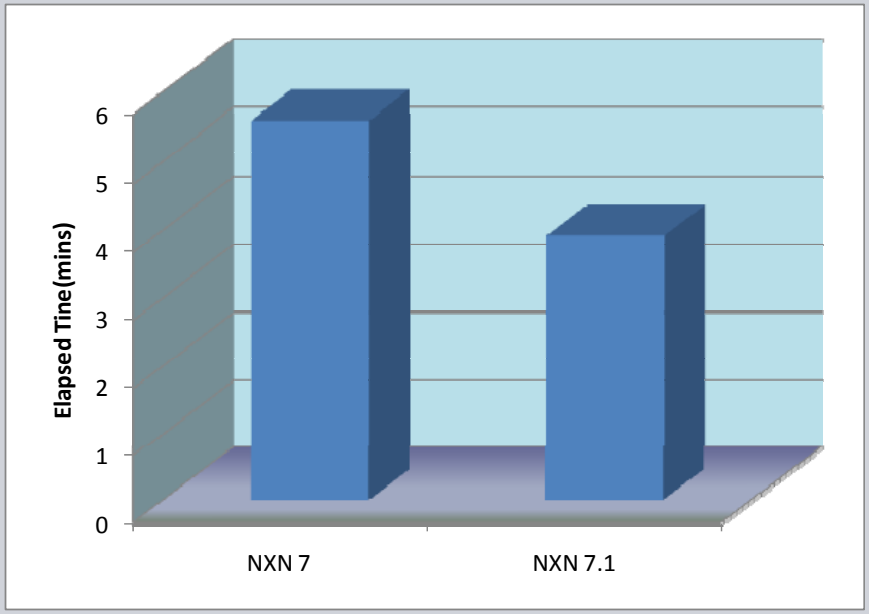
NX Nastran 7.1

Bolt Preload with Contact Performance

SIEMENS

Bolt preload with contact performance improvement

- Bolt preload solution steps:
  1. Solve for strains from preload
  2. Solve for applied load + preload strains
- Previously, if contact present both steps started with all elements in contact
- Now step 2 starts with elements in contact at the end of step 1 – bolt preload converges much faster



NX Nastran 7.1

Connectivity Improvements

SIEMENS

Glued contact definition by property ID

- Solid element glued contact regions can now be defined by property ID
- More flexible contact region creation

Performance improvement

- Models with a large number of contact and/or glued faces run more efficiently

Contact support for linear buckling

- Contact now extends to buckling solutions
- Expanded range of buckling solutions

Orthotropic materials for solids

- New material property for solid elements – MAT11 & MATT11
- More flexibility for defining orthotropic materials

Solid element support in SOL 106 nonlinear solution sequence

- Penta and Hexa large displacement effects (Tetra already supported)
- Previously treated as small displacements

Report best design

- Best optimized design is usually, but not always, the last iteration
- New messaging in F06 identifies the best design

Optimum solution bulk model data export

- Complete bulk data model file with updated design variables exported to the punch file

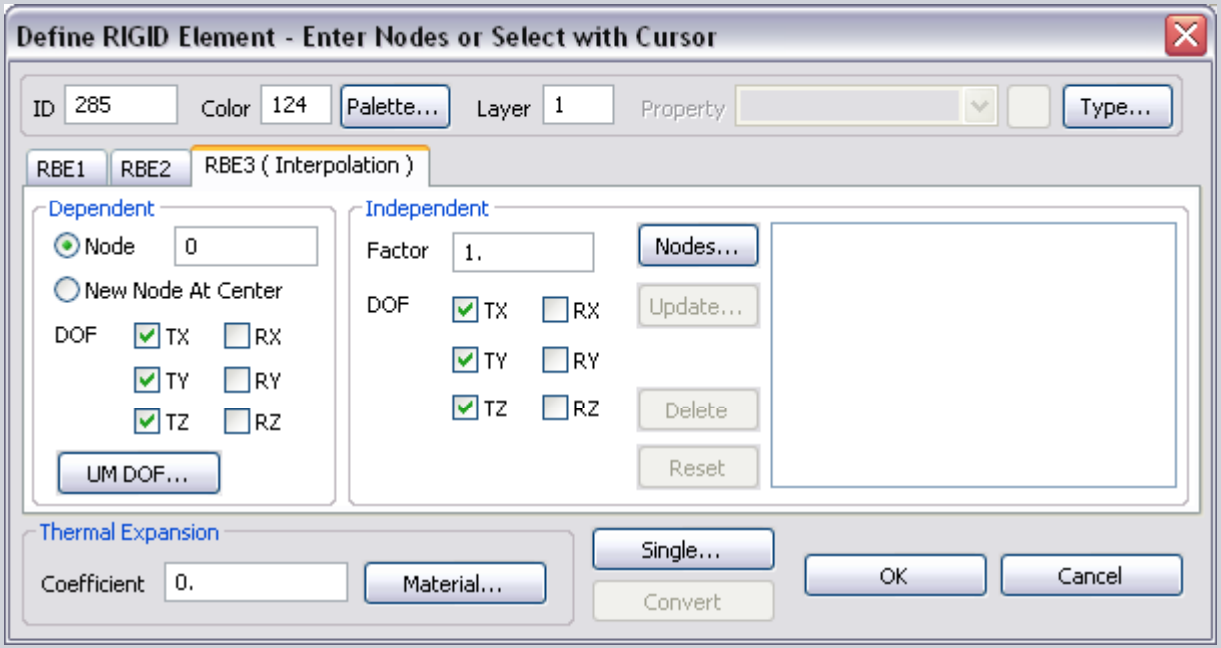
New elements in Advanced Nonlinear

- Plane stress elements
  - CPLSTS3, CPLSTS4, CPLSTS6 connectivity entries
  - PPLANE property entry
- Plane strain elements
  - CPLSTN3, CPLSTN4, CPLSTN6 connectivity entries
  - PPLANE property entry
- 2D edge-to-edge contact

SIEMENS

General NASTRAN Enhancements

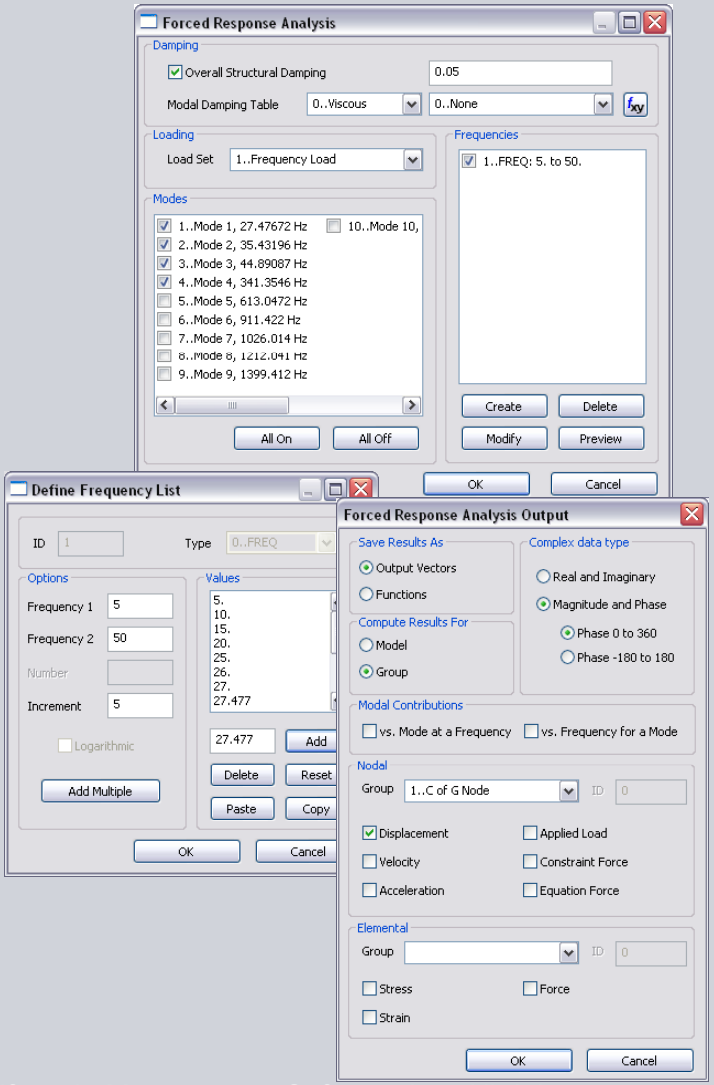
- Added read support for GROUNDCHECK, WEIGHTCHECK
- Added support for reading FREQ1, FREQ2, FREQ3, FREQ4 cards
- Added r/w support for RBE1
- Added support for RBE3 UM DOF



Femap Forced Frequency Response Analysis

- Fast, easy to use, interactive frequency response analysis right inside Femap

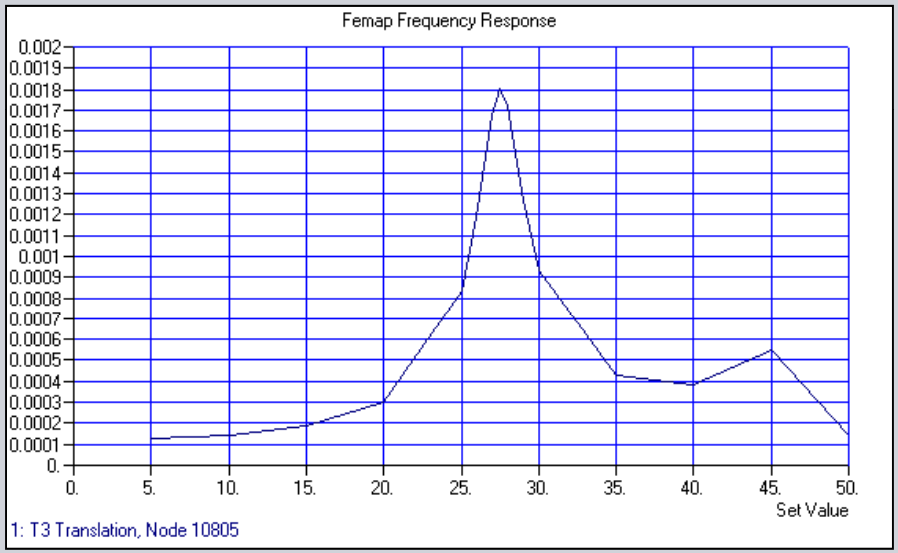
- Run a modal analysis using any solver and recover mode shapes, stresses, etc.
- Apply load and load function
  - Nodal and elemental loads
- Specify output area of interest
  - Group of entities or whole model
- Specify modes of interest to be included in analysis
- Specify frequency range of interest
- Submit analysis – Femap calculates sinusoidal response to forces, moments, and elemental pressure loads directly



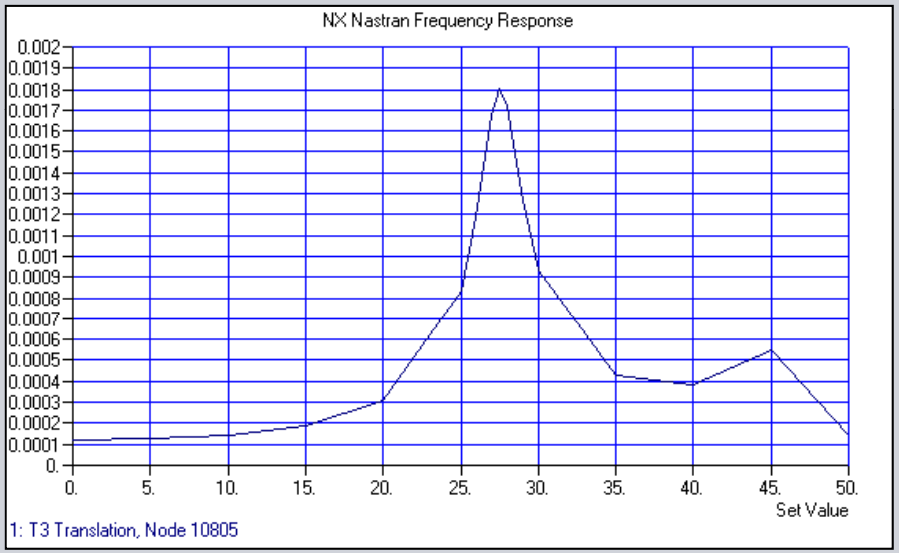
SIEMENS

Femap Forced Frequency Response Analysis

Frequency response analysis calculated by Femap



Frequency response analysis calculated by NX Nastran

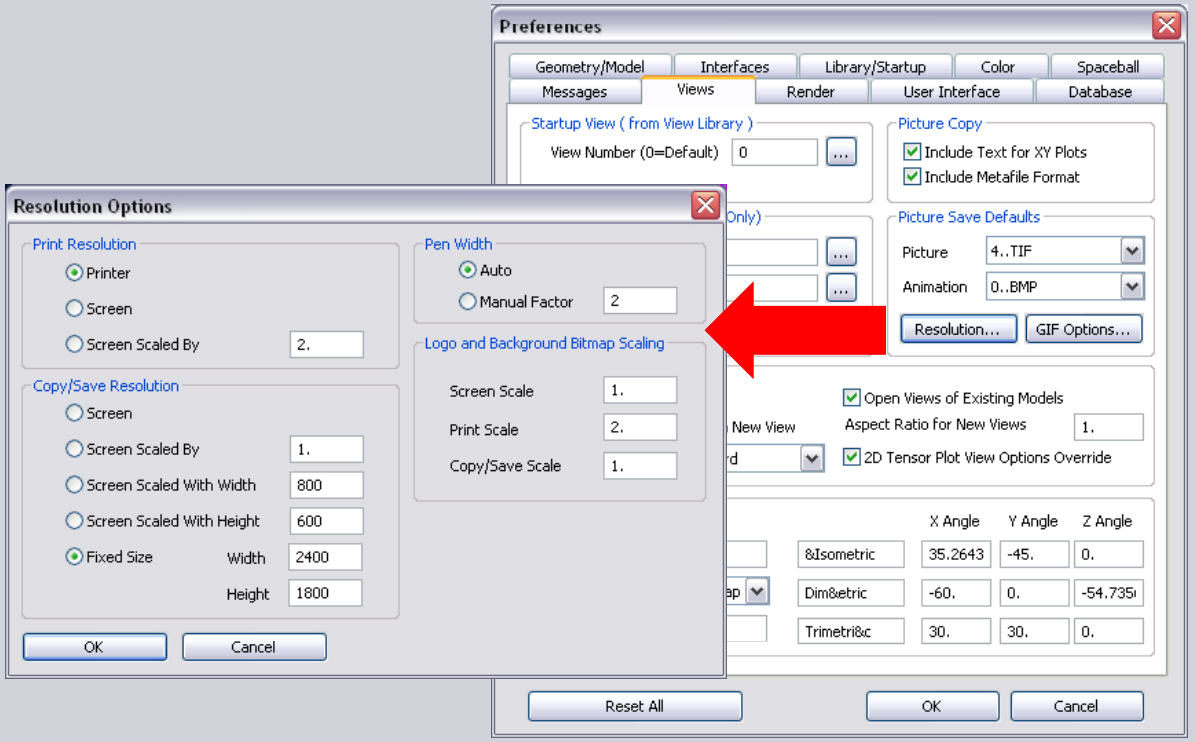


SIEMENS

Graphical Output

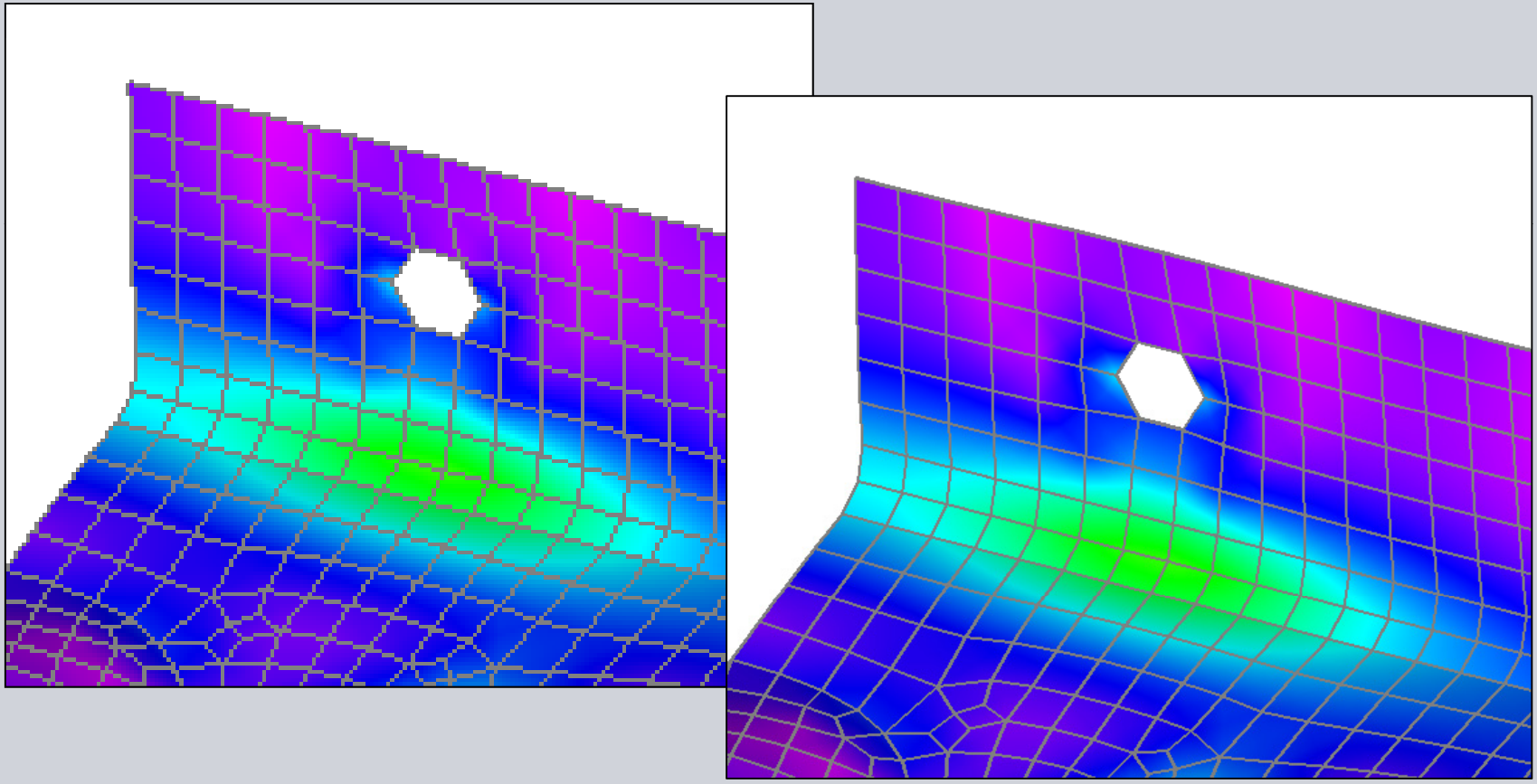
More control over Femap graphics screen capture image allows much higher image resolution and greatly improved picture quality

- Printing can now query the printer and send a full resolution image
- Copying / Saving you can specify the required resolution or scale



SIEMENS

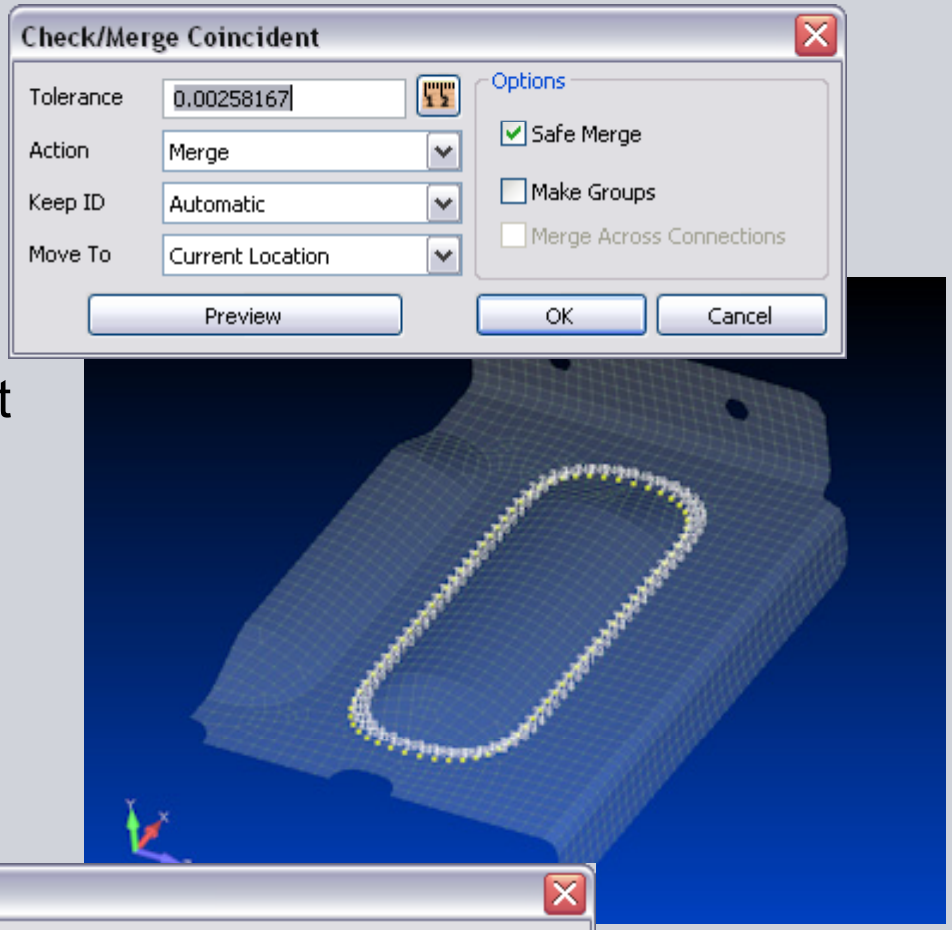
Graphical Output



SIEMENS

Updated Node Merge

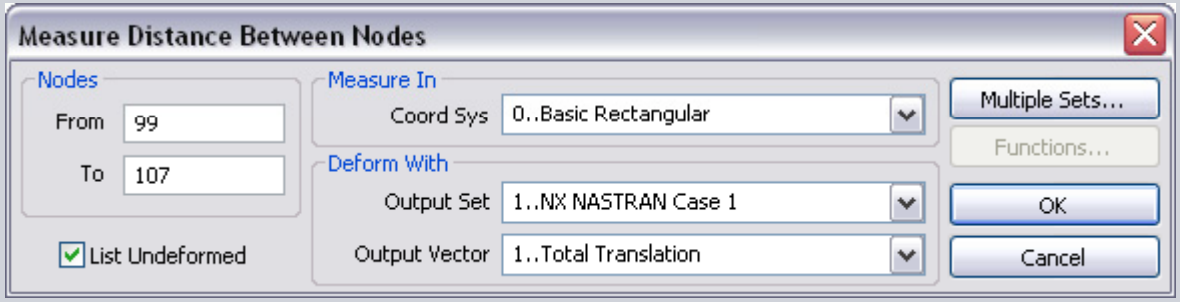
- Simplified User Interface
- Options for Merged Node Location (at Lower ID, at Higher ID, at MidPoint) or original location
- Prevents Over-Merging/Element Corruption with Large Merge Tolerances
- Preview now Interactive, can change tolerance and preview again
- Simplified Listing

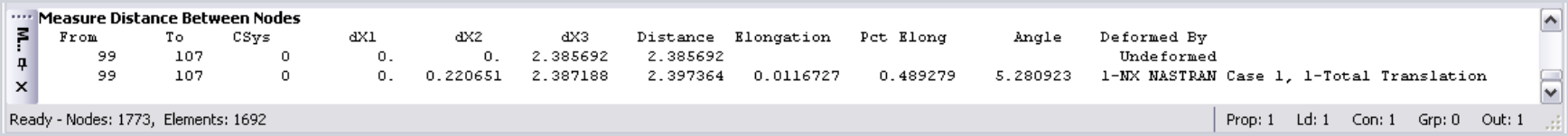


New Distance Measuring Tools

Measure distances between nodes in your FEA model

- User Selected Coordinate System
- Includes the effects of deformation

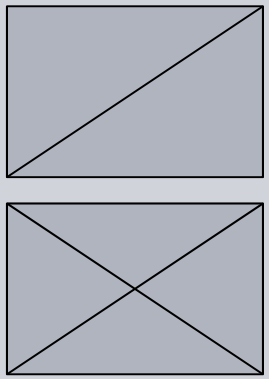


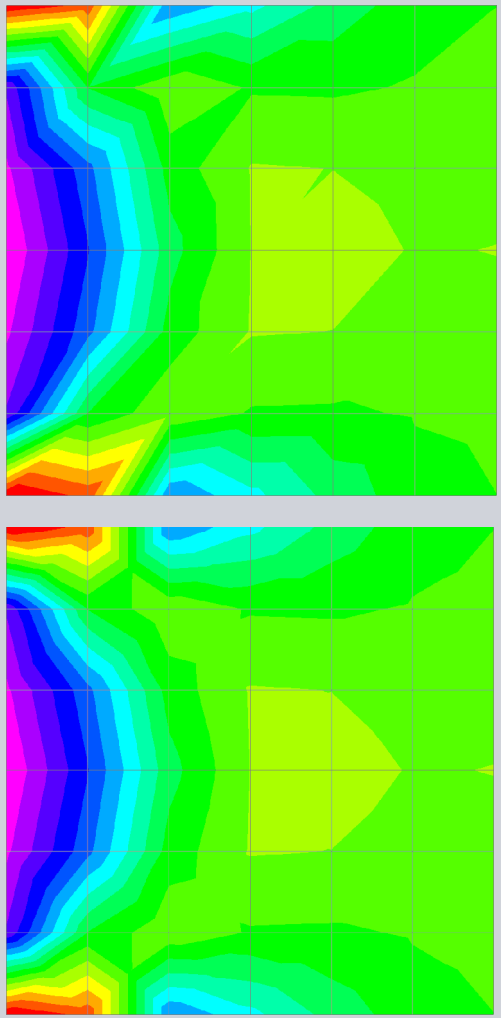


SIEMENS

New Contour Algorithm

- Previously quad elements were split into two triangles for contouring
  - Symmetry not guaranteed and dependent on the way quads are split
- New algorithm splits quads into four triangles
  - Higher quality contour plot that assures symmetry

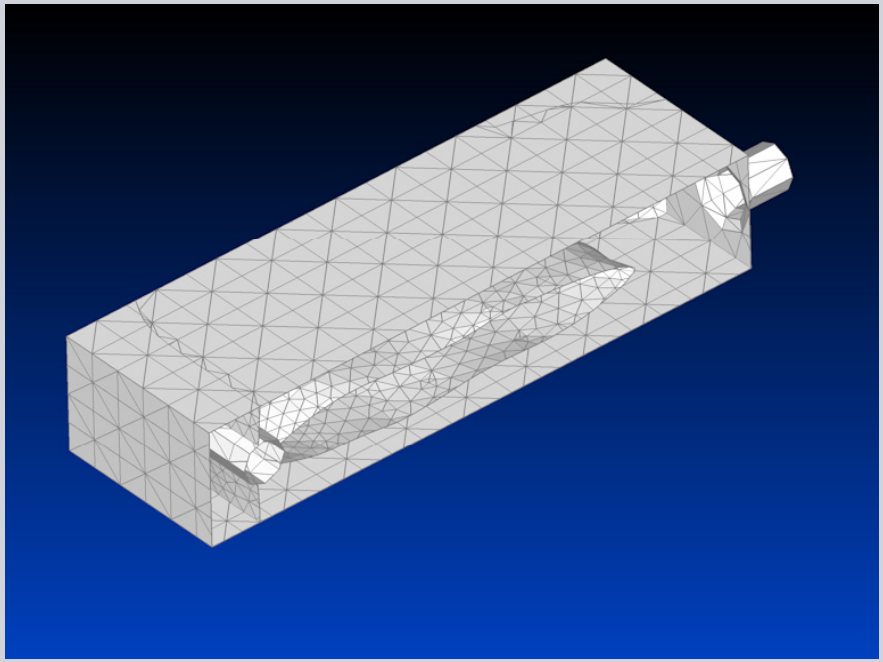




SIEMENS

Quick Model Clipping Plane

- Geometry or FEA model can be clipped using OpenGL
- Dynamic control (Alt-Mouse Wheel)
- Change side (+/-)
- Standard Femap Plane dialog box



SIEMENS

Thermal and Flow Solver Updates

- Consistent model unit systems
  - Model Units Standard option defines the unit system that applies consistent units for all unit types
  - Nine unit systems are available based on SI or Imperial units
- New flow results
  - Acoustic power density: acoustic noise generated by the simulated turbulent flow
  - Vorticity: vector data equal to the curl (rotor) of the velocity field
- Satellite-sun/planet vector display
  - Satellite-sun and satellite-planet vectors are now displayed in the Orbit Visualizer as well as the spacecraft triad

SIEMENS

Q and A