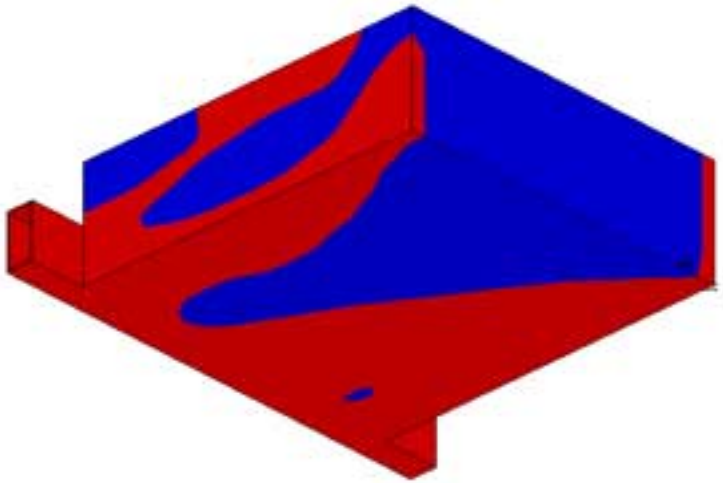
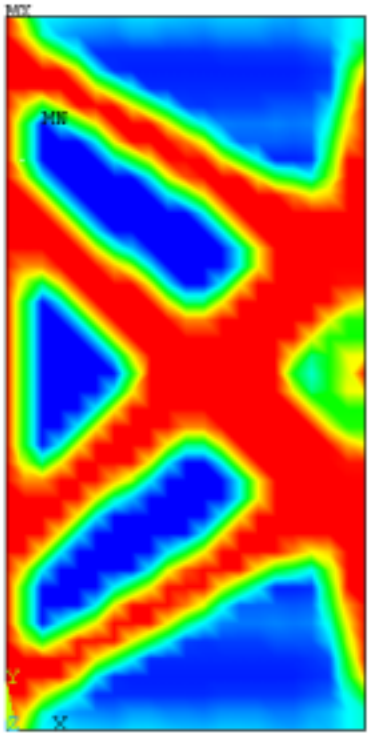
IMPACT
ENGINEERING SOLUTIONS, INC.

# Topology Optimization in ANSYS






Presented by:

**Brian King**

*Analysis Project Engineer*

**IMPACT Engineering Solutions, Inc.**

Tel: 847-599-5635

E-mail: bking@impactengsol.com

IMPACT
ENGINEERING SOLUTIONS, INC.

# About IMPACT…

- Founded in 1987, IMPACT Engineering Solutions Inc. has grown into a market leader, providing technology based engineering services
- Focused on solid modeling since 1993
- Key IMPACT Product Offerings:
  - Design Support
  - Staff Augmentation
  - Product Simulation
  - Professional Services
- **Analysis Services Division** Initiated October 2002
- Gurnee, IL Office Opened in November 2003

IMPACT
ENGINEERING SOLUTIONS, INC.

# Analysis Capabilities and Expertise

- COSMOSWorks Structure/Motion
- Pro/MECHANICA Structure/Motion
- MSC.NASTRAN / NE/Nastran
- FEMAP Pre & Post-Processing
- ANSYS
- CFDesign – CFD Simulation
- LS-DYNA – Drop and Crash Testing
- FE-Fatigue (nCode) Durability Analysis

Extensive combined engineering analysis experience in a variety of materials and industries

# What is Topological Optimization?

- Layout optimization
- Try to find the best use of material for a body
- No optimization parameters need to be defined
  - The material distribution function over a body is the optimization parameter
- The goal (objective function) is to minimize / maximize the energy of structural compliance or maximize the natural frequency while satisfying the constraints specified

# What is Topological Optimization?

- The design variables are pseudo-densities
  - Assigned to each finite element
  - Values range from 0 to 1
    - 0 = material to be taken away
    - 1 = material to be kept

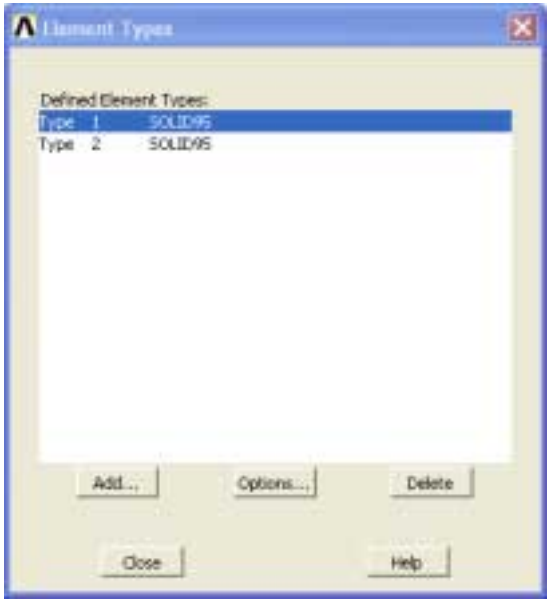
# Main Steps of Optimization Procedure

- Define the structural problem
- Select the element types
- Specify optimized and non-optimized regions
- Define and control the load cases or frequency extraction
- Define and control the optimization process
- Review the results

# Define the structural problem

- Define the problem as you would for any linear elastic analysis
  - Single or multiple load case linear structural static analysis
  - Modal frequency analysis
- Material properties to be defined
  - Young's modulus
  - Poisson's ratio
    - Must be between 0.1 and 0.4
  - Material density if necessary

# Element Types

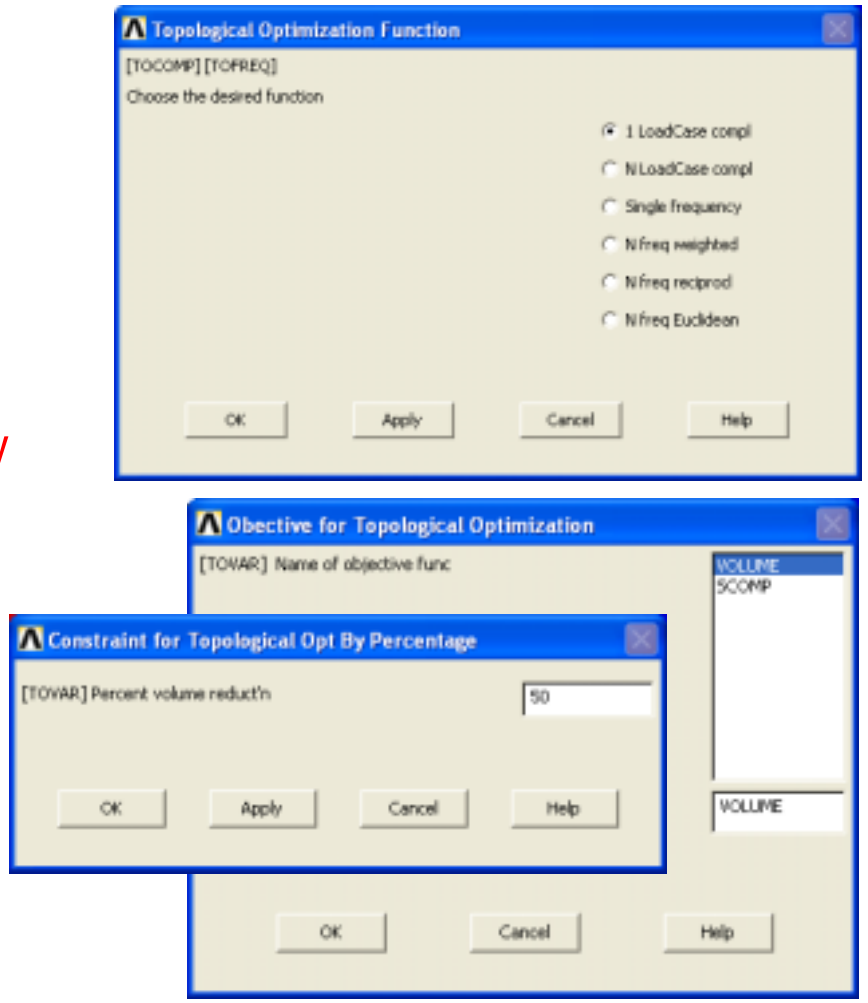



- 2D Planar, 3D Solid, and shell elements are supported
  - 2D Solids: PLANE2, PLANE82
  - 3D Solids: SOLID92, SOLID95
  - Shells: SHELL93
- Only Type 1 elements optimized
  - Used to control which regions of the model to optimize

IMPACT
ENGINEERING SOLUTIONS, INC.

# Primary Commands Used

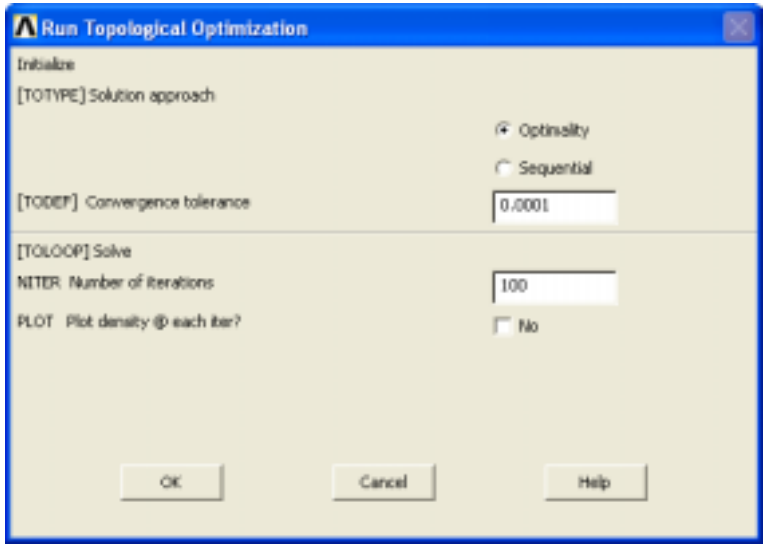
- TOCOMP
  - Defines single or multiple load cases as topological optimization function for linear static problem
- TOFREQ
  - Defines single or mean frequency formulation as the topological optimization function for modal analysis
- TOVAR
  - Specifies objective and constraints



IMPACT
ENGINEERING SOLUTIONS, INC.

# Primary Commands Used

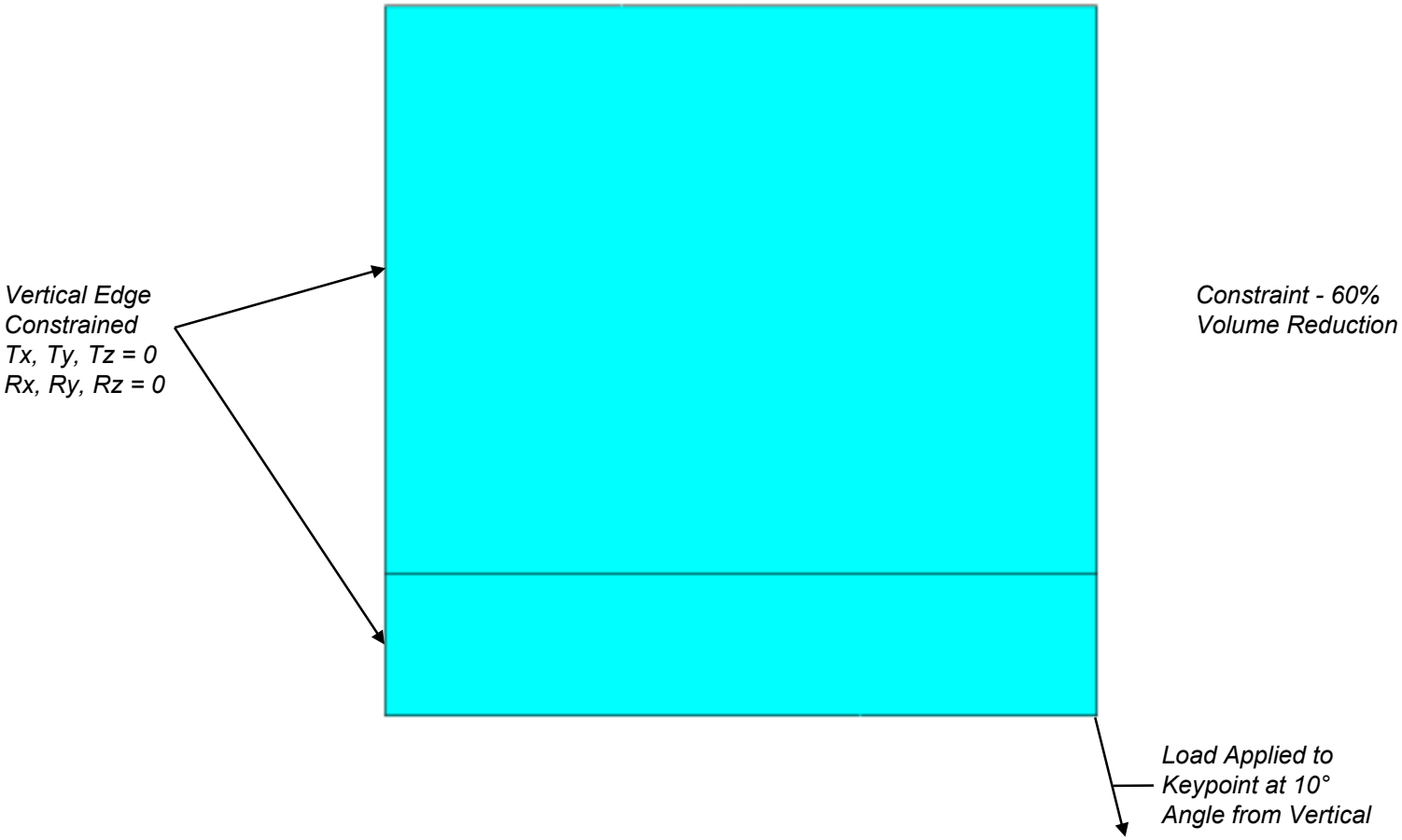
- TOTYPE
  - Specifies solution method for topological optimization
- TODEF
  - defines the accuracy for the solution
- TOLOOP
  - Invokes a macro to solve, postprocess, and plot each iteration
  - Process terminates once convergence is attained or the maximum iteration number is reached
  - Up to 100 iterations allowed



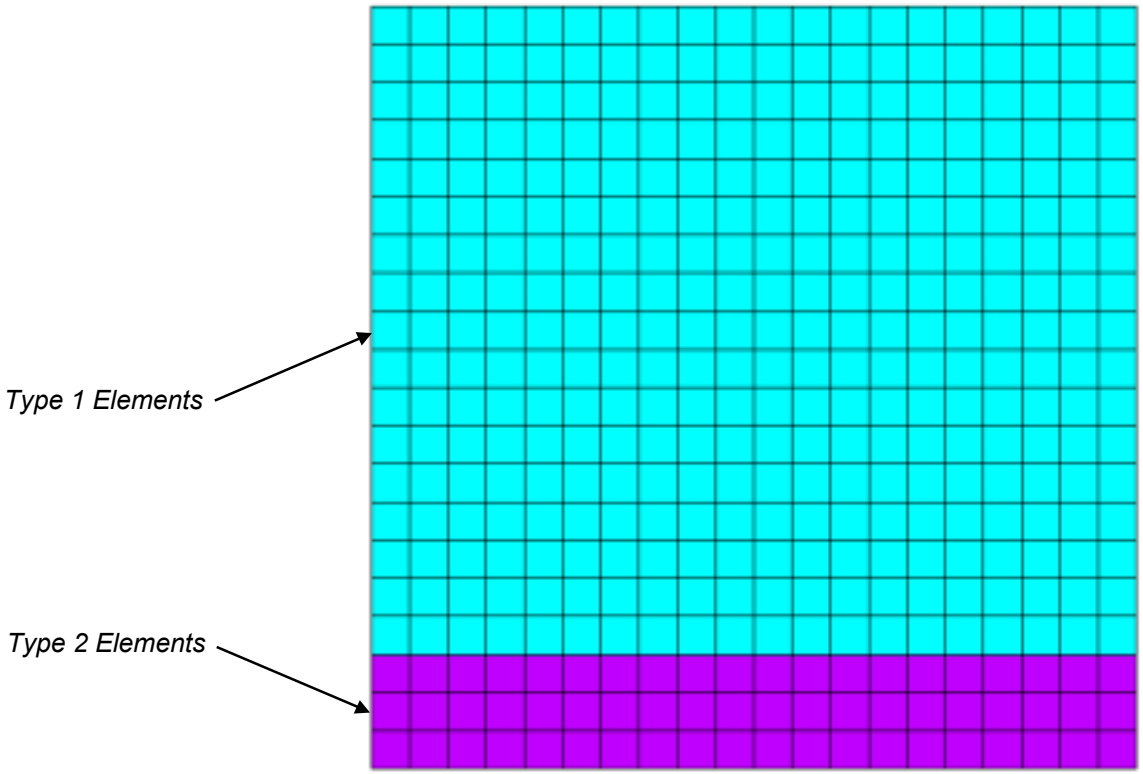
# Workbench vs. ANSYS Optimization

- Solid parts only in Workbench Topology Opt.
- Type 1 and Type 2 elements driven by boundary conditions in Workbench
  - Preprocessing commands can change defaults
- Only Basic Opt from ANSYS is available
  - Single load case
  - Maximize stiffness, reduce volume
  - Preprocessing commands for Advanced Top. Opt.

# Textbook Problem

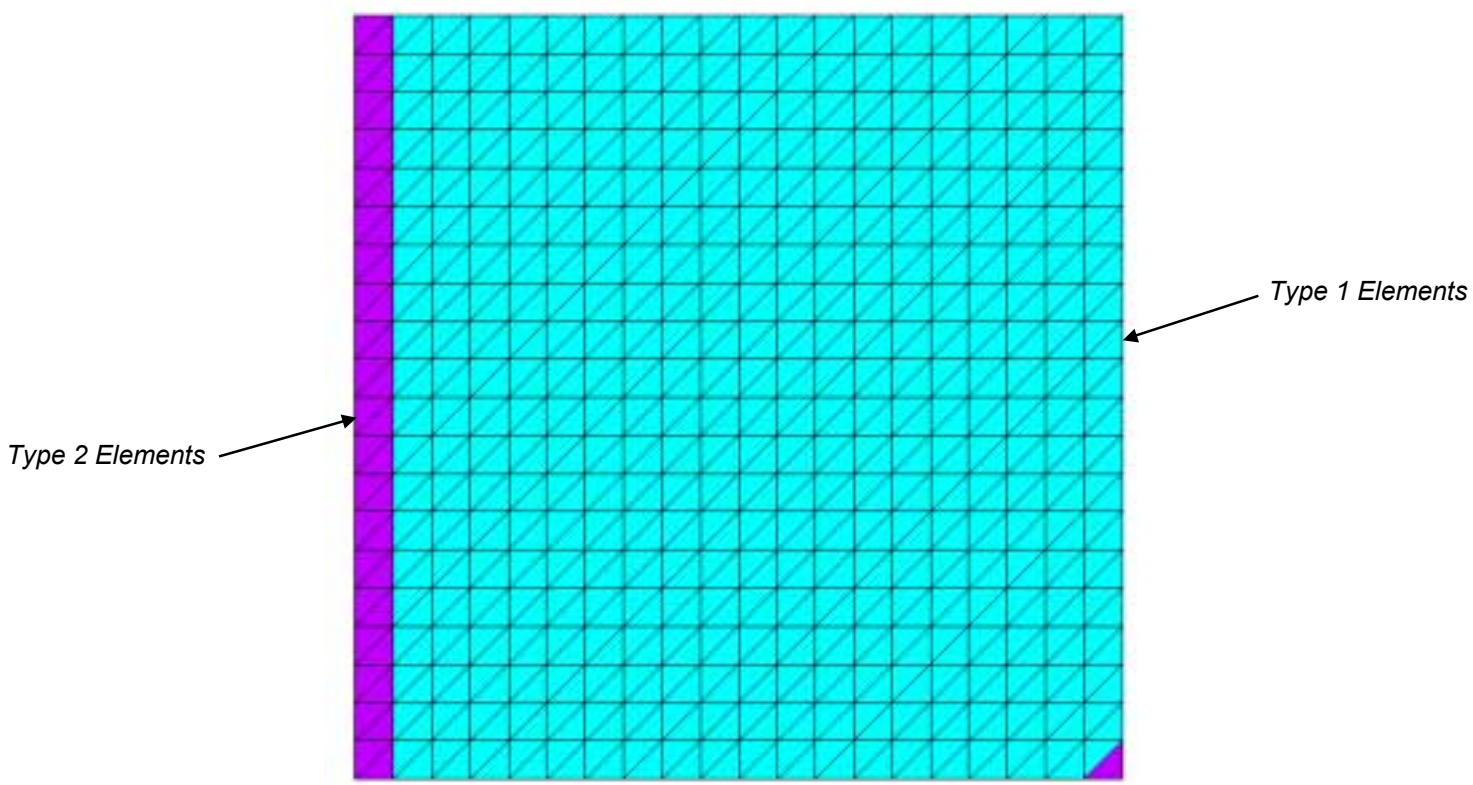


# Textbook Problem



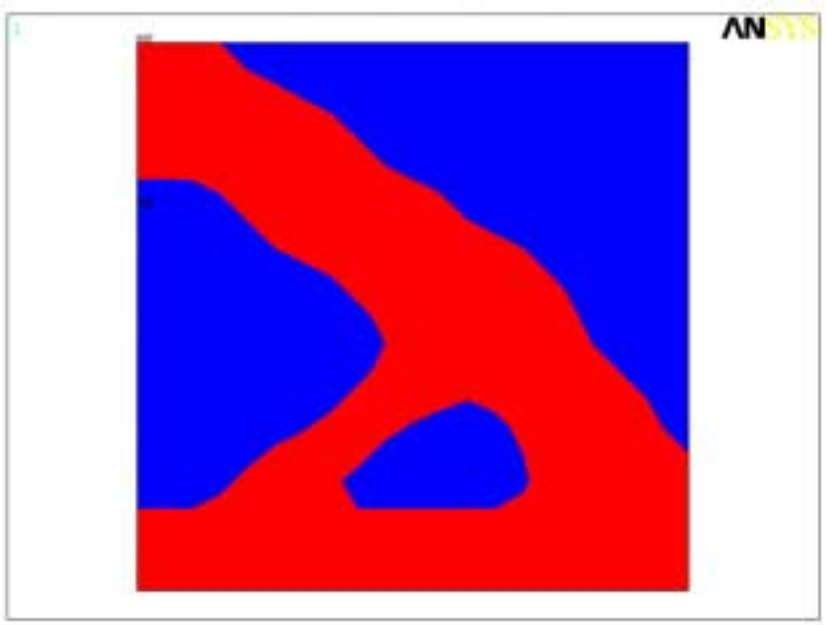

ANSYS Mesh

# Textbook Problem

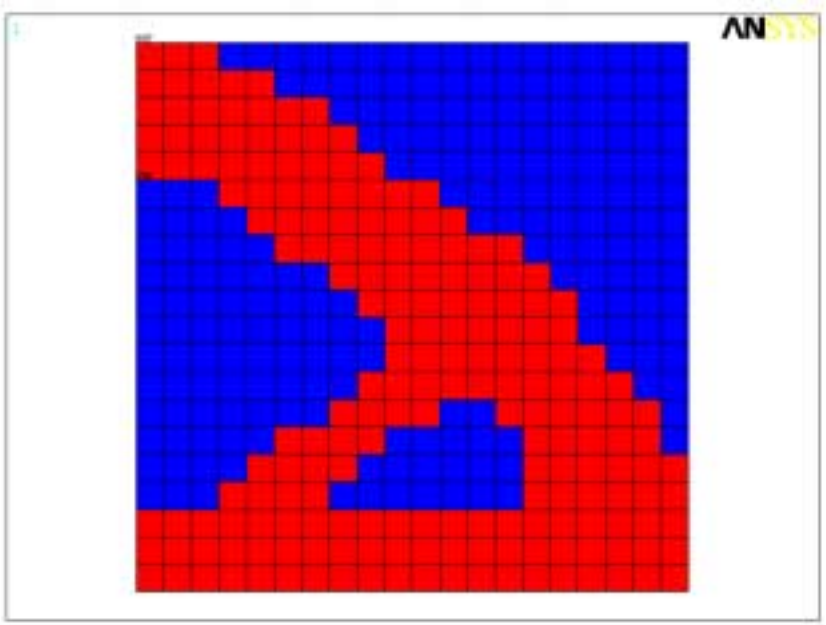



Workbench Default Mesh

# Textbook Problem



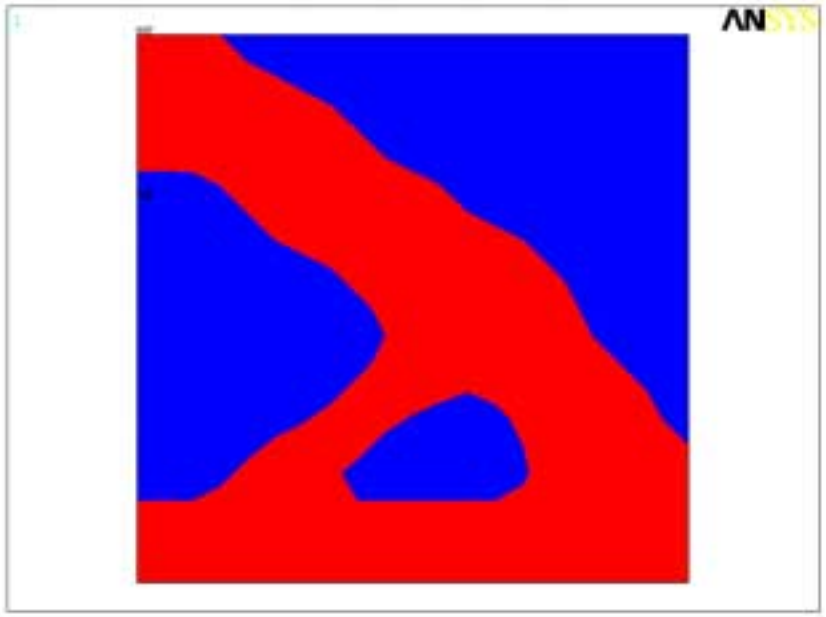
**Density Plot - Averaged**



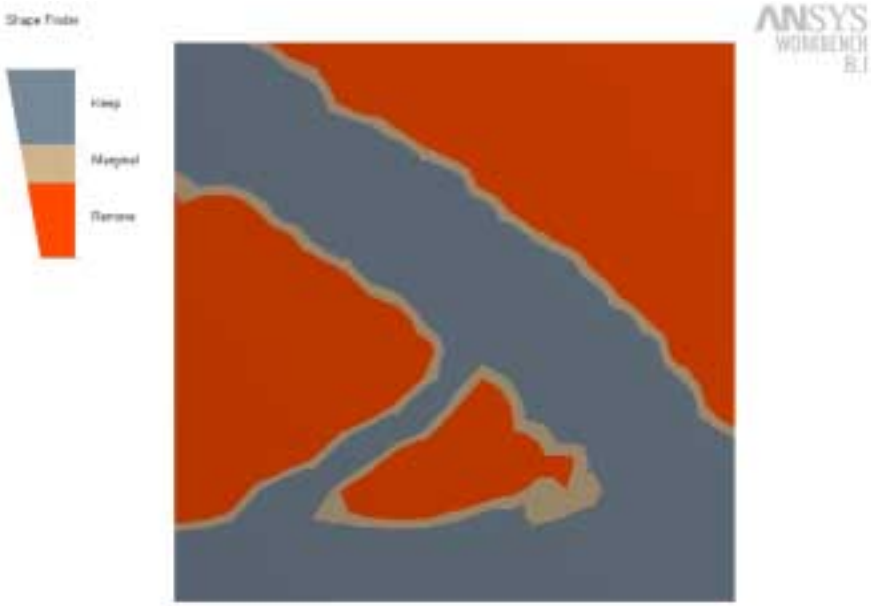
**Density Plot - Unaveraged**

*Blue = pseudo-density 0 < 0.5*
*Red = pseudo- density 0.5 ≥ 1.0*

# Textbook Problem



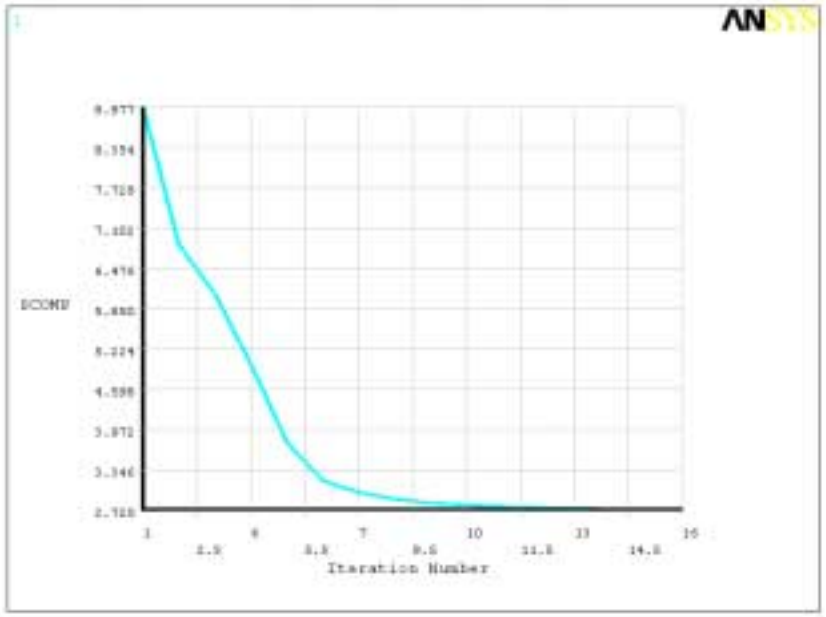
ANSYS Density Plot

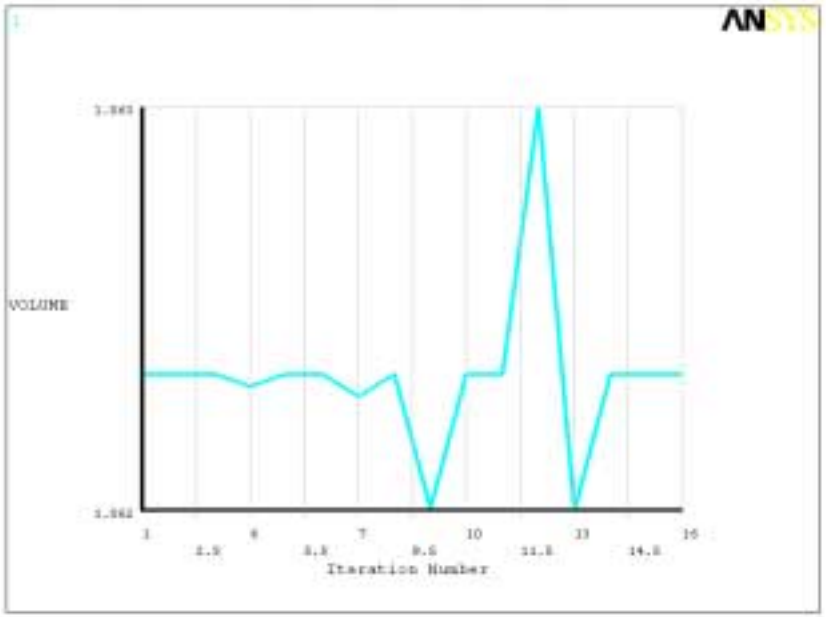


Workbench Density Plot

IMPACT
ENGINEERING SOLUTIONS, INC.

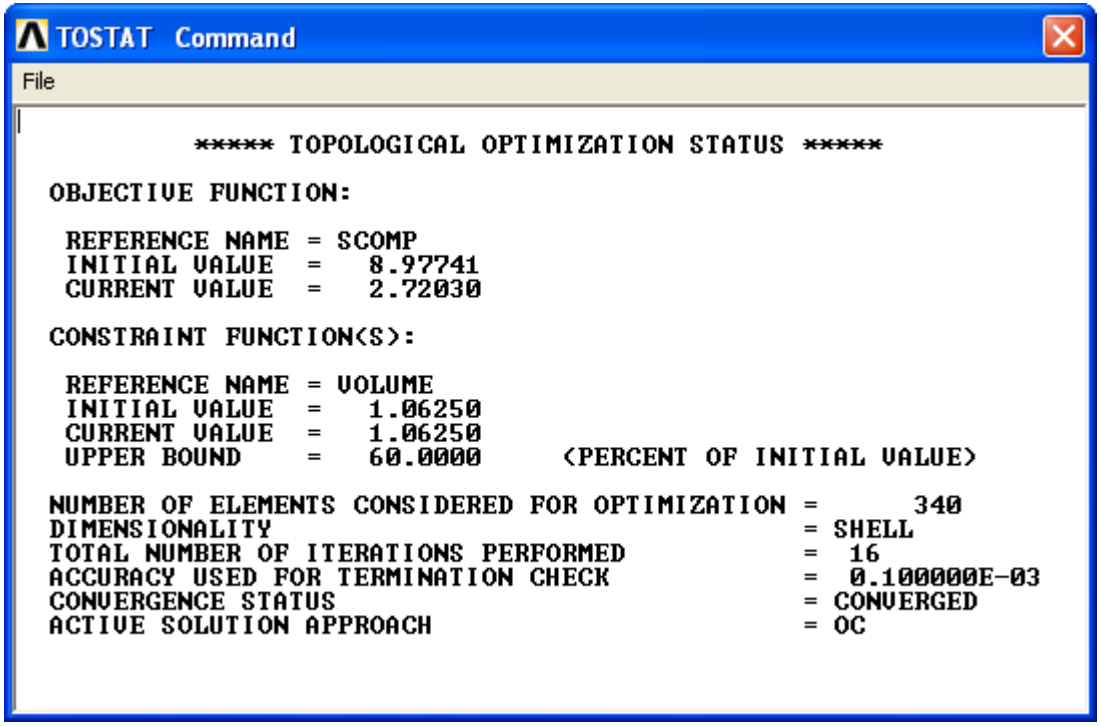
# Textbook Problem



IMPACT
ENGINEERING SOLUTIONS, INC.

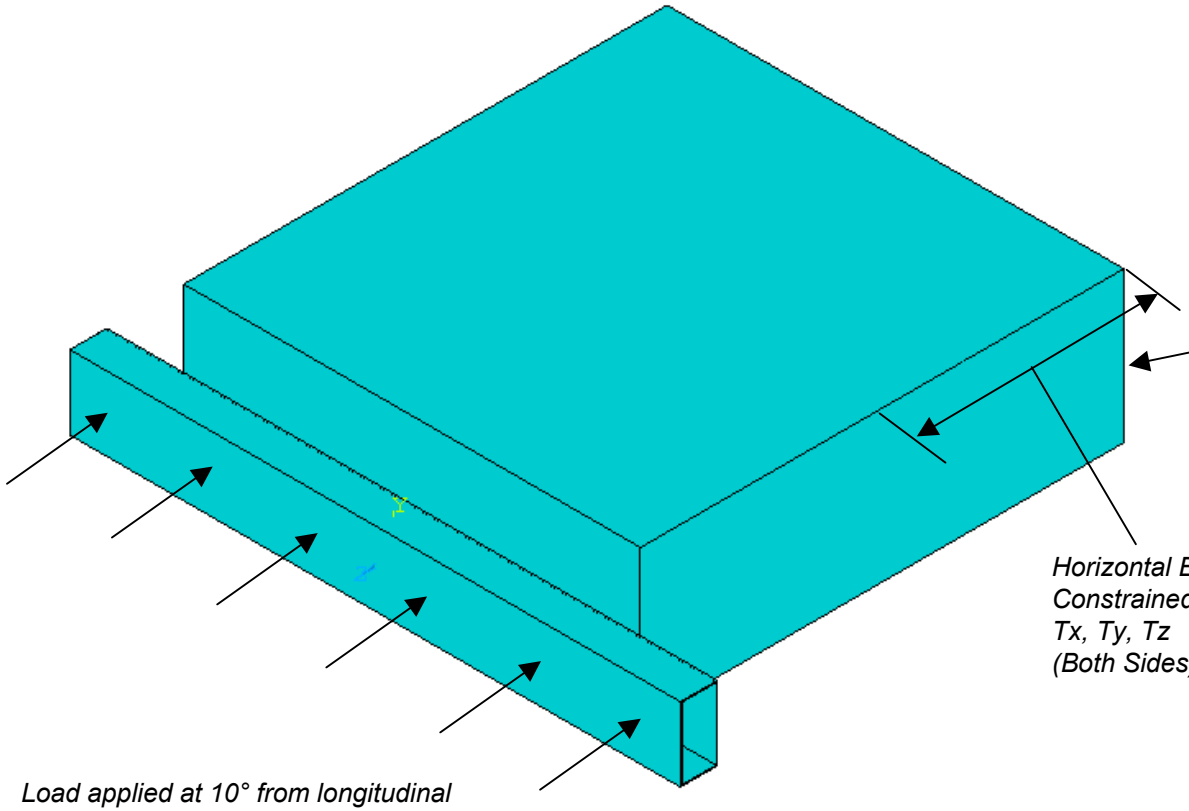
# Textbook Problem

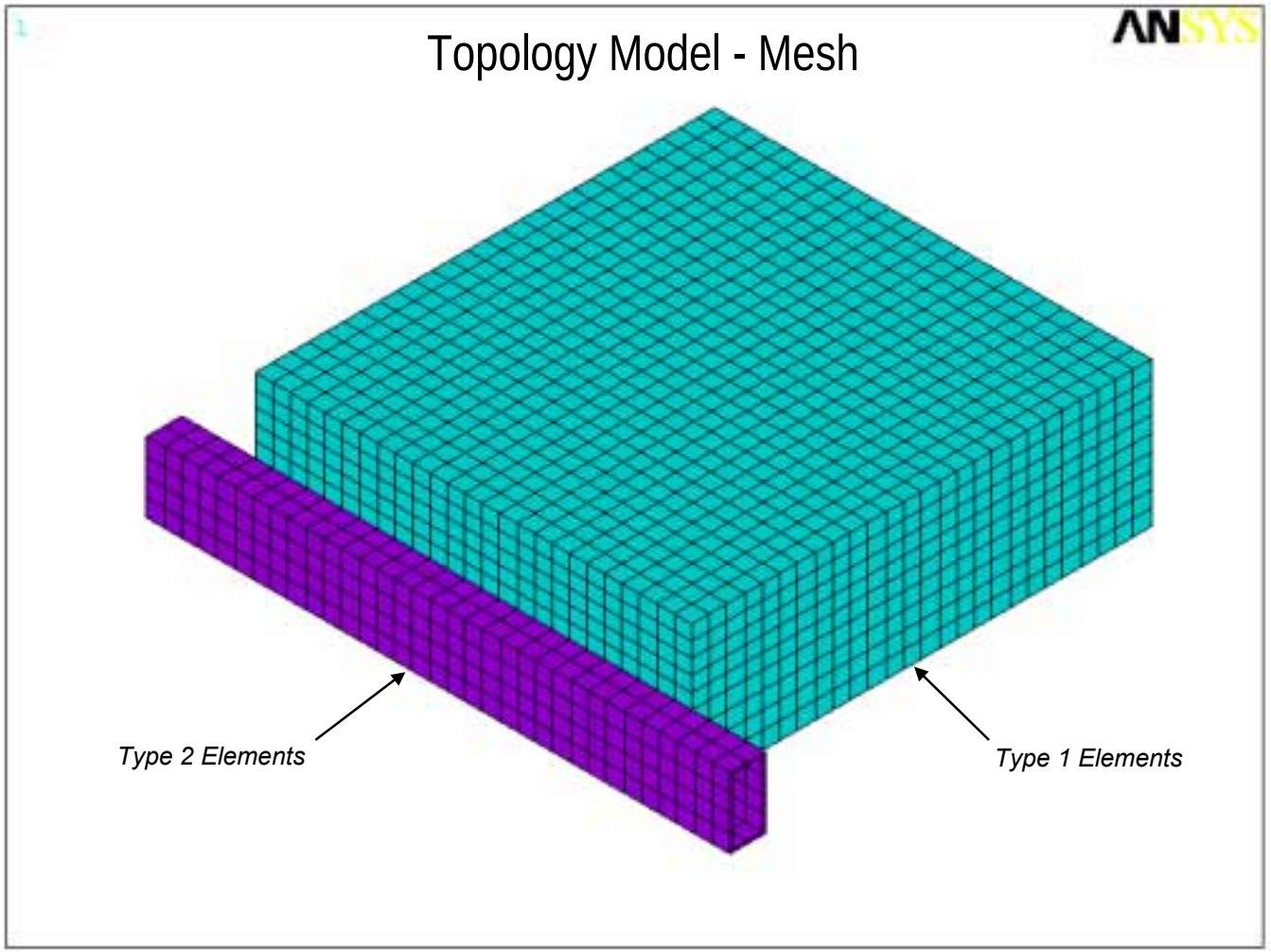


# Case Study – Bumper Project

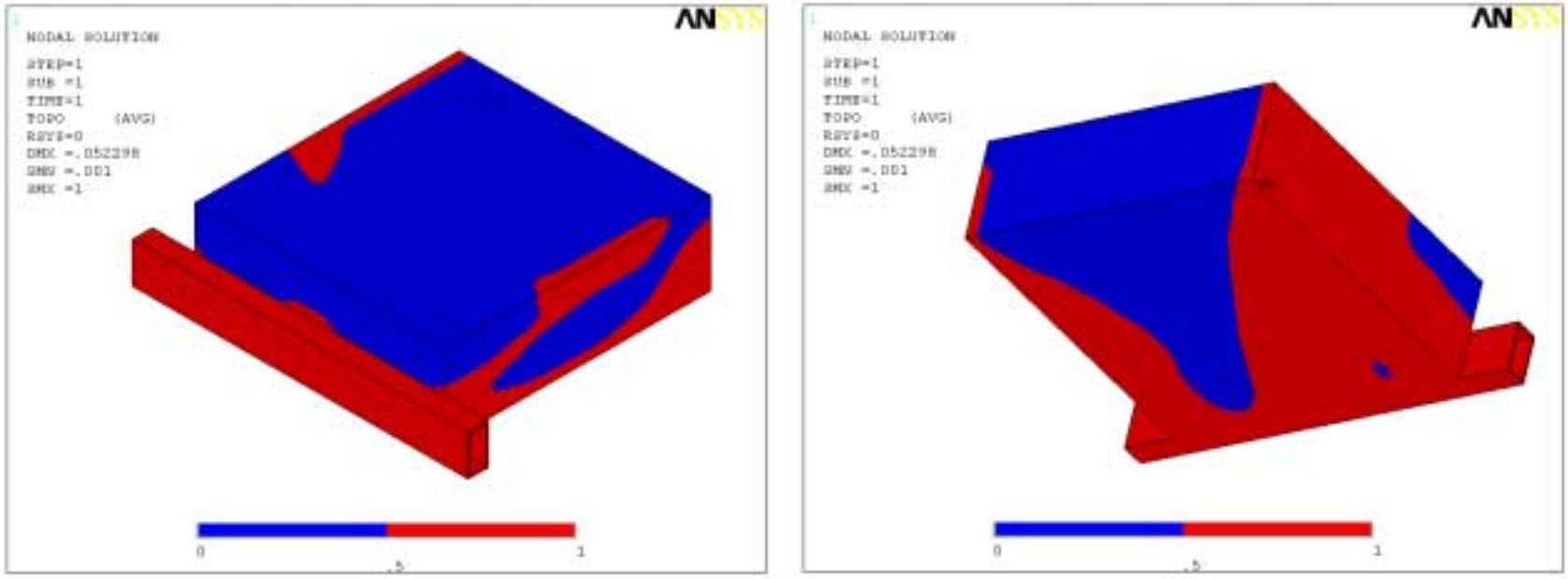
## Topology Model – Problem Definition

ANSYS





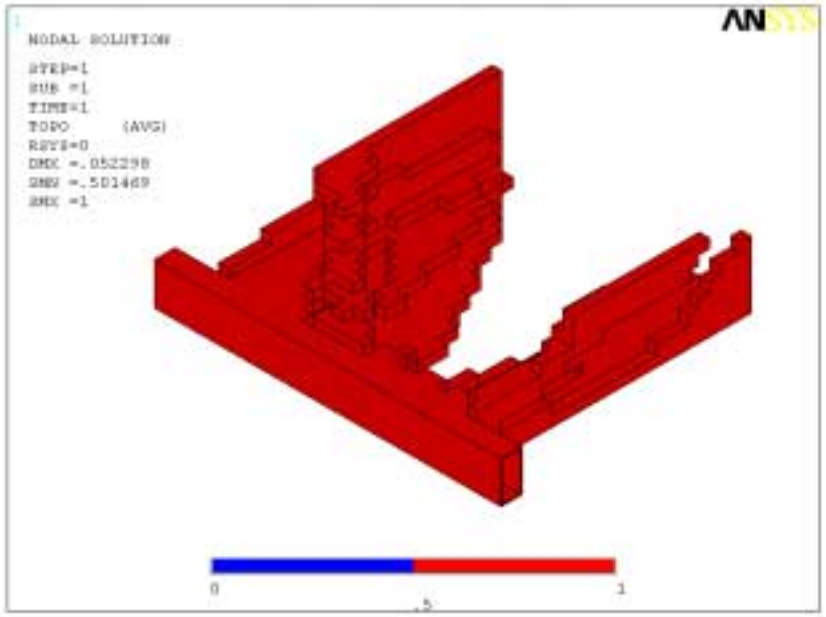
# Density Plots

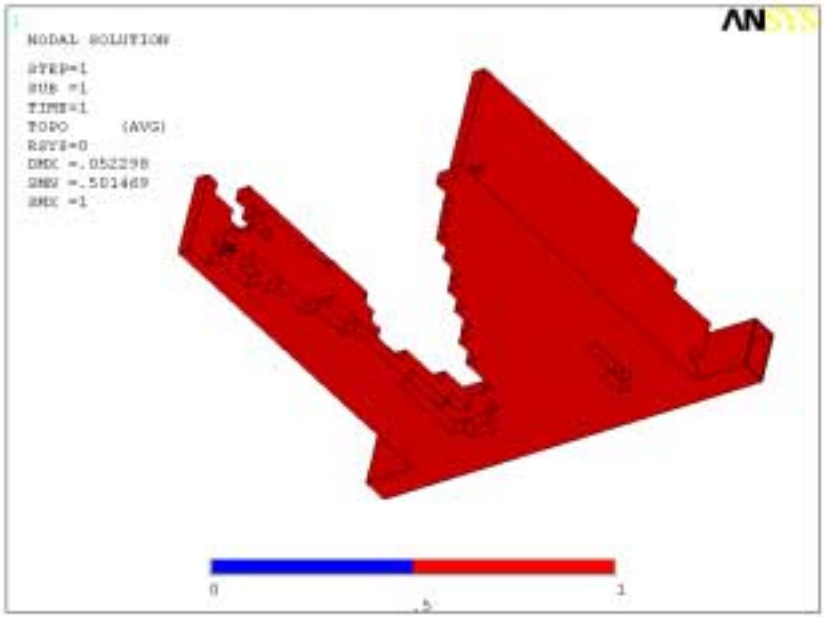



*Blue = pseudo-density 0 < 0.5*
*Red = pseudo- density 0.5 ≥ 1.0*

IMPACT
ENGINEERING SOLUTIONS, INC.

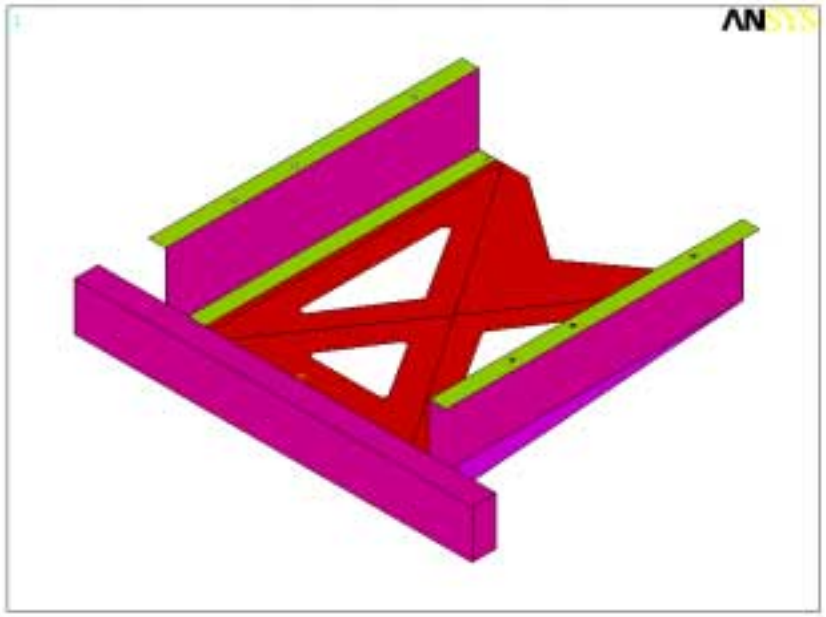
# Density Plots

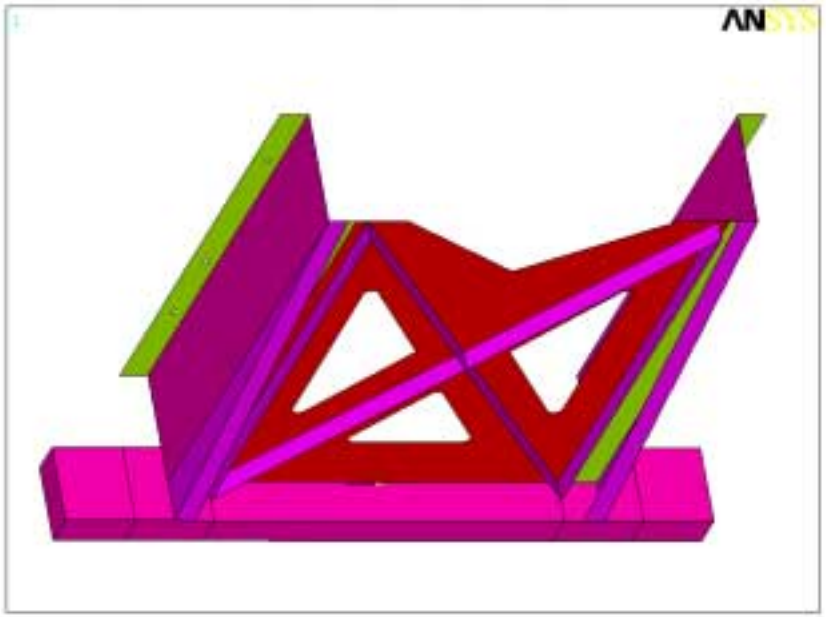



Elements with density 0.5 or greater only

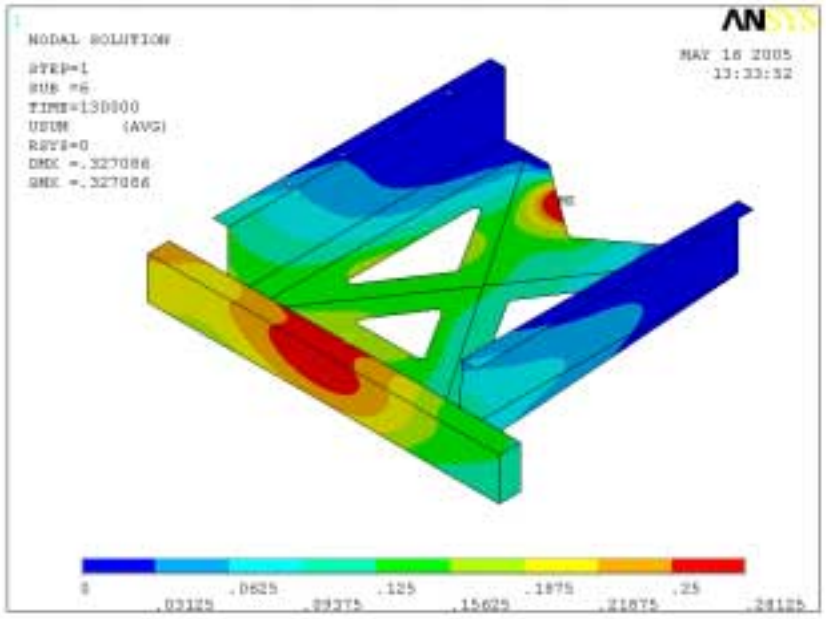
# Model Created Based on Topology Results



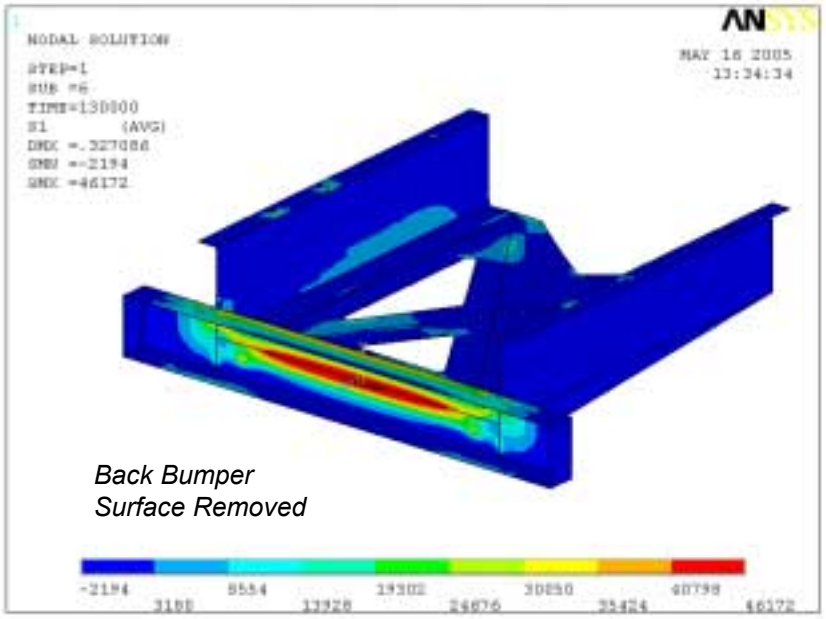


IMPACT
ENGINEERING SOLUTIONS, INC.

# Analysis Results




Displacement (in)



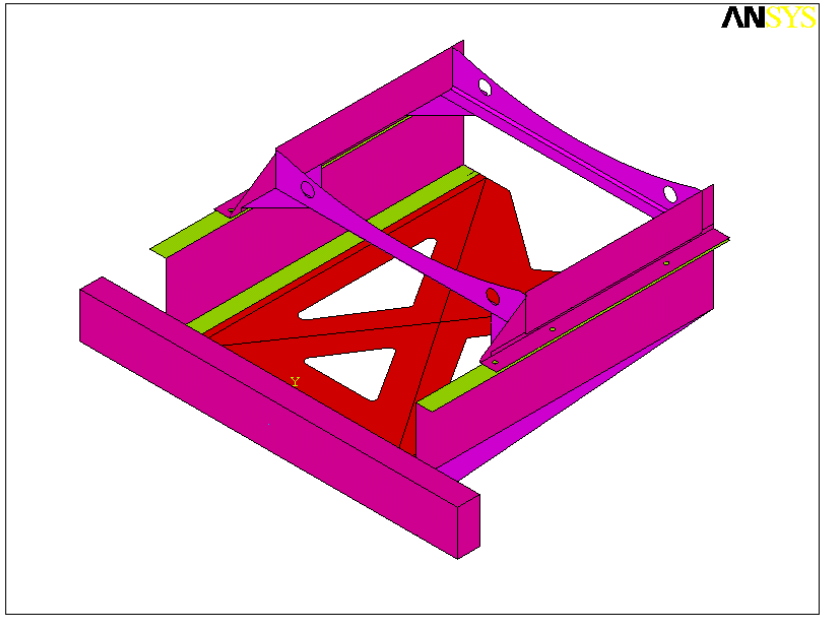

*Back Bumper Surface Removed*

Maximum Principal Stress (psi)

IMPACT
ENGINEERING SOLUTIONS, INC.

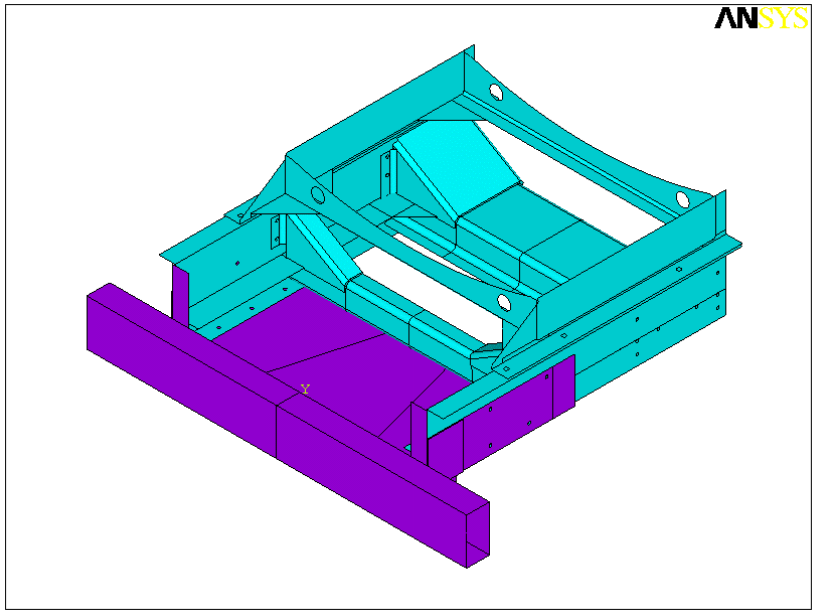
# Comparison to Current Design

New Prototype




*204.5 lbs*

Current Design




*363.5 lbs*

159 lb Difference!

# Hints and Comments

- Results are sensitive to the load configuration
- Results are sensitive to the density of the mesh
- When a large (80% or greater) volume reduction is requested and a very fine mesh is used a truss-like solution may occur
- A linear structural static analysis or a modal analysis must be performed during optimization looping
- Inputs and commands are not saved in the ANSYS database

IMPACT
ENGINEERING SOLUTIONS, INC.

# Questions?

Contact Information:

**Brian King**

Brookfield, WI Office: (262) 317-8118

Gurnee, IL Office: (847) 599-5635

E-mail: bking@impactengsol.com