Global Engineering Solutions

Serving Globally

One Stop Solution for Mechanical Engineering

Design And Engineering Analysis

- Engineers need to analyze various components/designs for a variety of loading, material, manufacturing conditions.
- Come up with a best design without extensive experiments and prototype building.
- FEA is a computational (numerical) modeling method.

What is FEA?

- Finite Element Analysis is a way to simulate various conditions (loading, material, etc.) on a design and determine the Design's response
- Finite Element Method (FEM) divides a design model into smaller "elements" and solves the resulting system of equations
- FEA is used in many industries to conduct modal, structural, harmonic, thermal and other analysis

Why FEA is Needed?

- Reduces the amount of prototype testing
  - Computer simulation allows multiple “what-if” scenarios to be tested quickly and effectively.
- Models designs that are not suitable for prototype testing
  - Example: Surgical implants, such as an artificial knee
- The bottom line
  - Cost savings
  - Time savings… reduces time to market!
  - Create more reliable, better-quality designs

Historical Note

- The finite element method of structural analysis was created by academic and industrial researchers during the 1950s and 1960s

- The underlying theory is over 100 years old, and was the basis for pen-and-paper calculations in the evaluation of suspension bridges and steam boilers

Three Phases of Finite Element Analysis

- Preprocessing:
  - Discretizing the solution domain into finite elements
  - Assuming a solution that approximates the behavior of an element
  - Developing equations for an element
  - Assembling the elements to present the entire problem
- Solution:
  - Solving a system of algebraic equations simultaneously to obtain nodal values of primary variables, e.g., displacements
- Post-processing:
  - Obtaining information on elemental values of secondary variables, e.g., strains, stresses and forces

Various Applications

- Different Software's are available in market for different type of analysis and application:
  - Static Analysis: Ansys, Abaqus, Nastran (Linear)
  - Crash Analysis: LS-Dyna, Nastran
  - Impact Analysis: ANSYS, LS-Dyna
  - Fatigue Analysis: FEMFAT, N-Code
  - Multi Body Dynamics: Adams
  - Meshing/Solver: Hyper mesh, ANSA, SimExpert

- Among above software's, ANSYS is user-friendly due to following reasons:
  - One Umbrella for all type of analysis
  - Couples Field Analysis
  - Geometry Access in Design Modeler
  - Easy for post processing

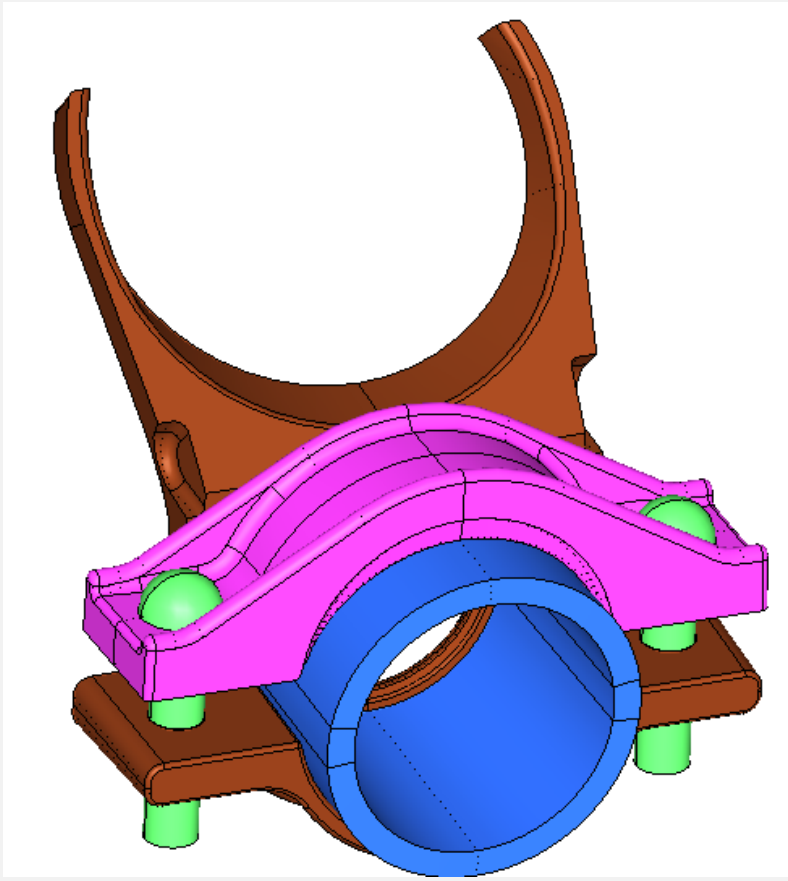
About ANSYS

ANSYS is a complete FEA software package used by engineers worldwide in virtually all fields of engineering:

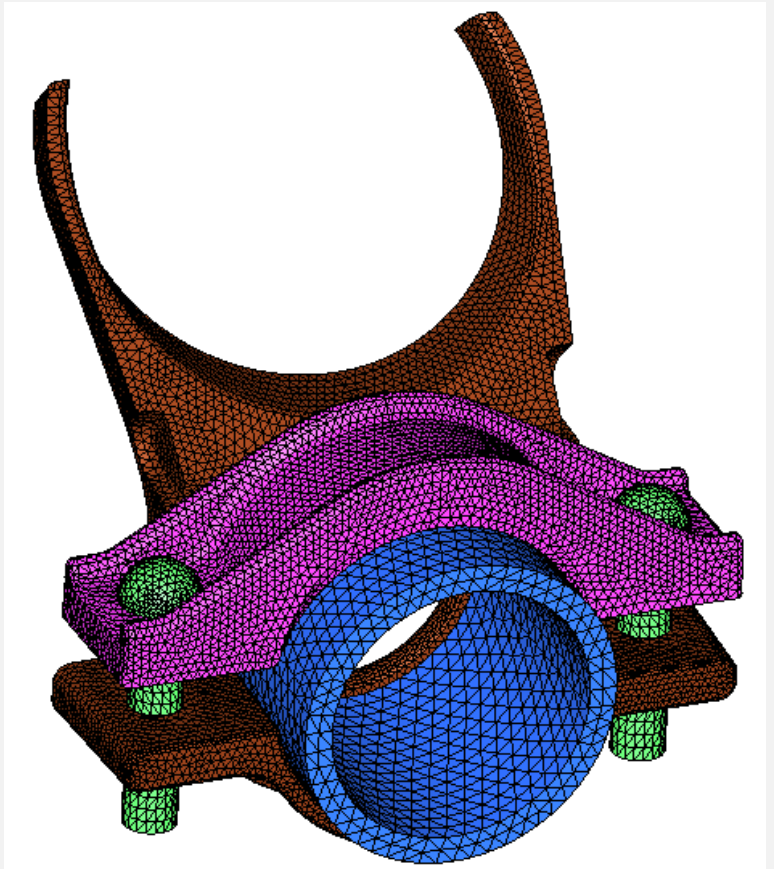
- **Structural**
- **Thermal**
- **Fluid, including CFD (Computational Fluid Dynamics)**
- **Electrical**
- **Electromagnetics**

Case Study: Pipe Clamp

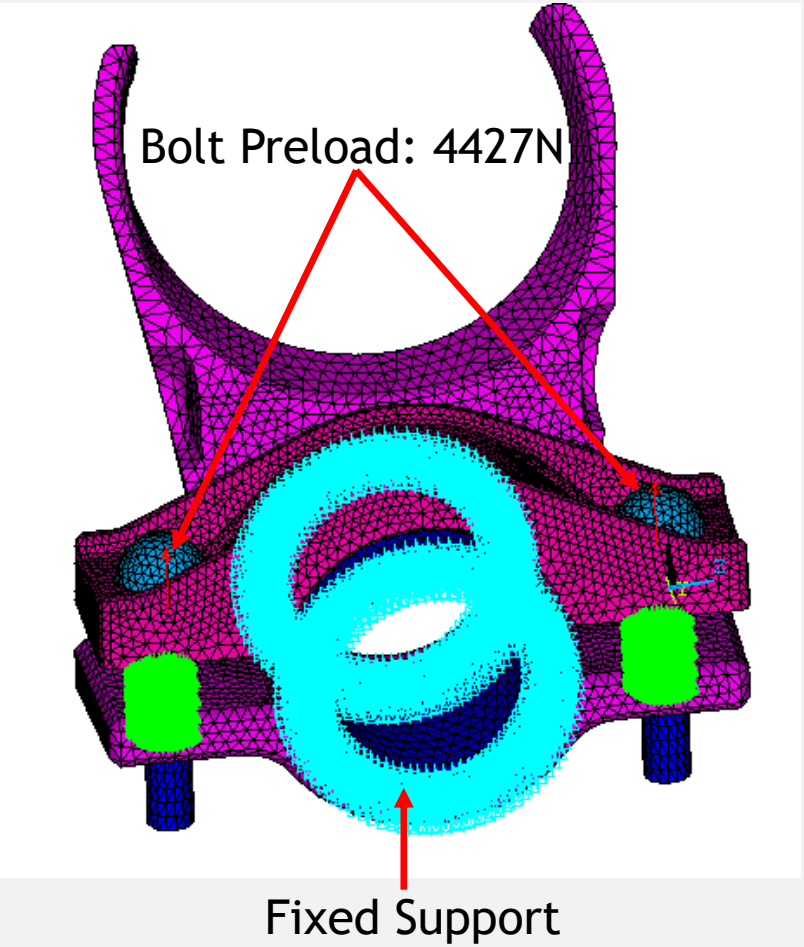
CAD Model



Meshed Model

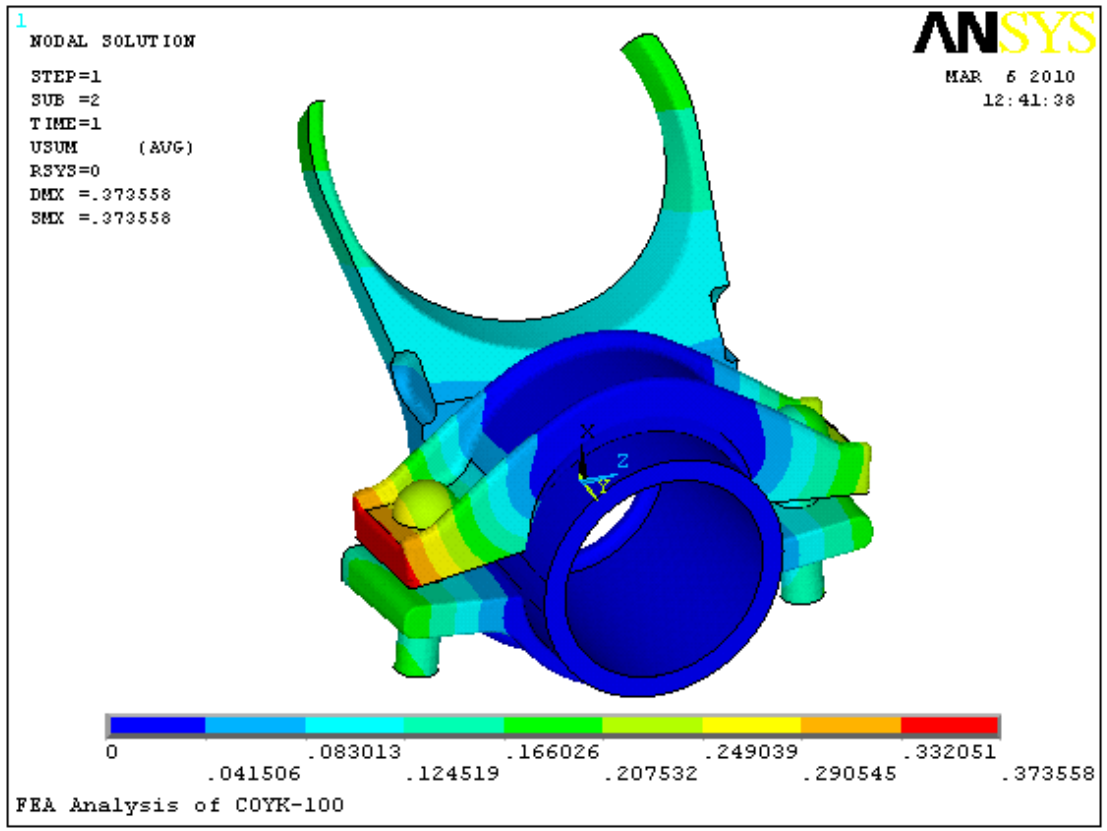


Boundary Conditions

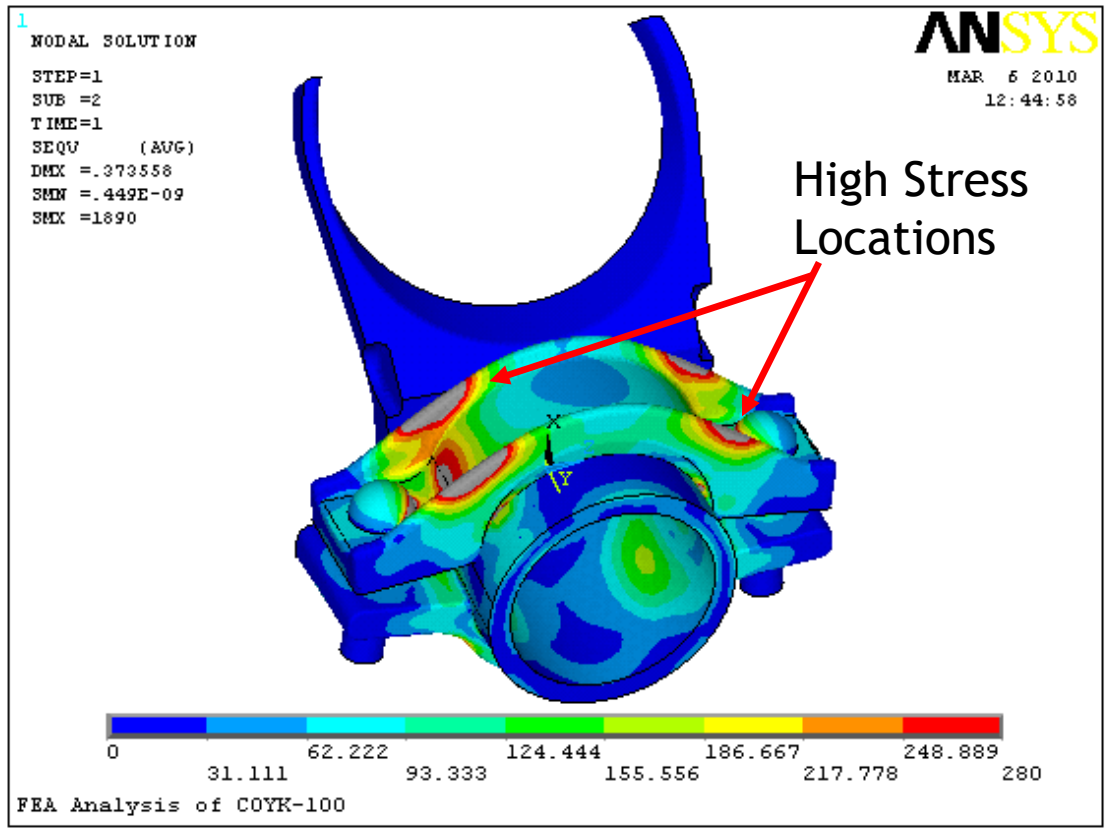


Case Study: Pipe Clamp

Deflection Plot (mm): Assembly

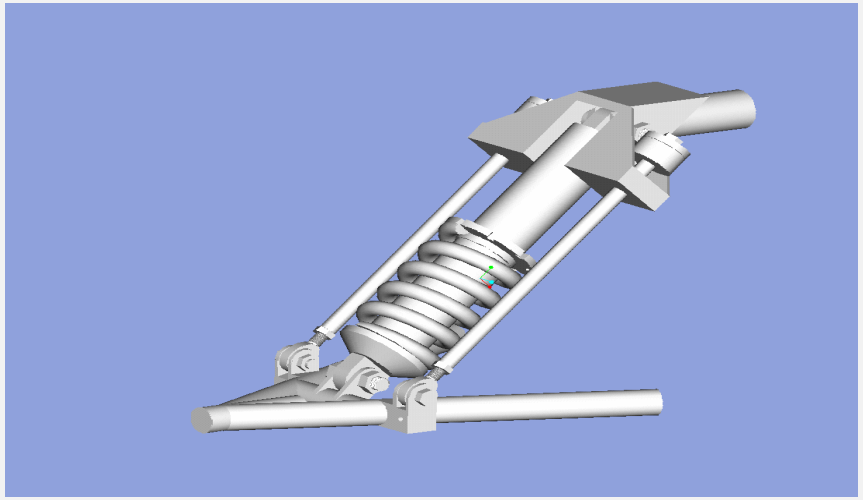


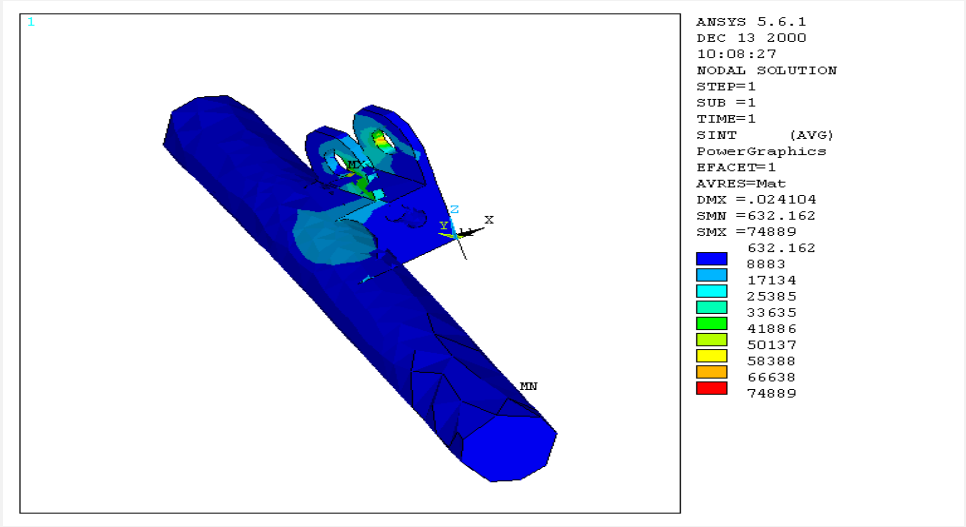
Equi. Stress Plot(MPa): Assembly

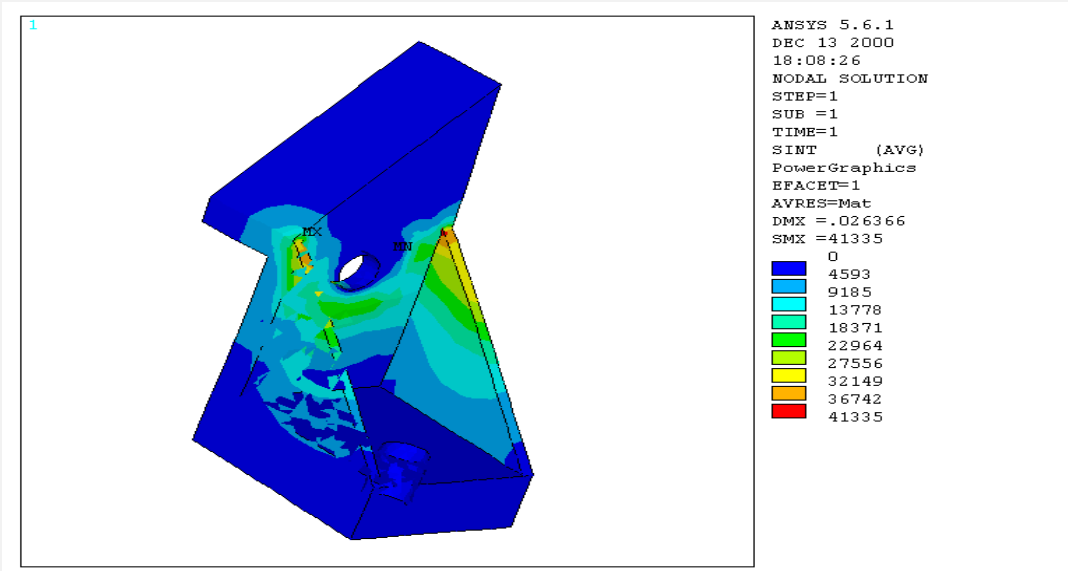


Yield Strength: 110MPa

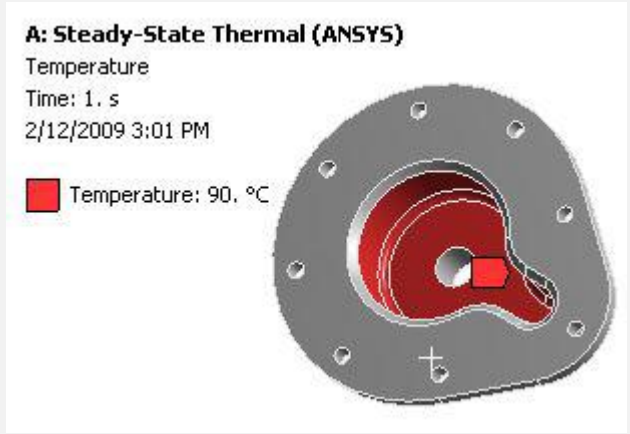
Case Study: Electric Car Shock Absorber

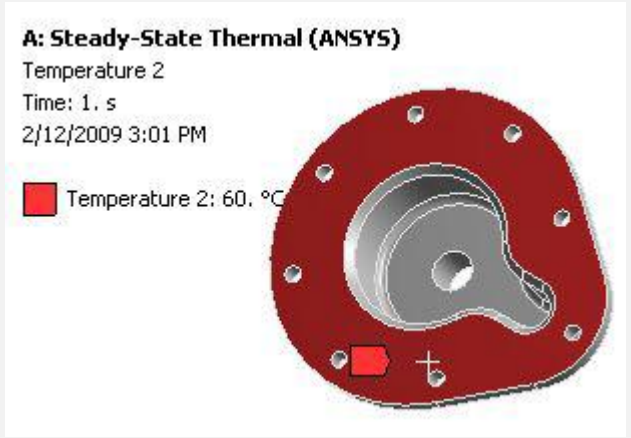




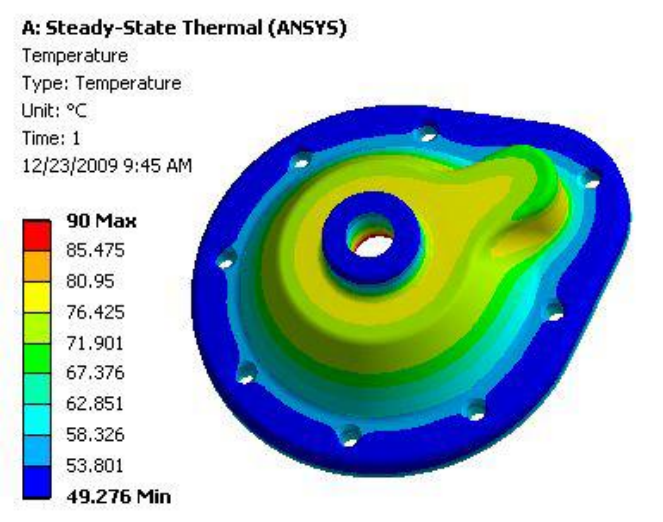


Case Study: Cap Thermal Analysis

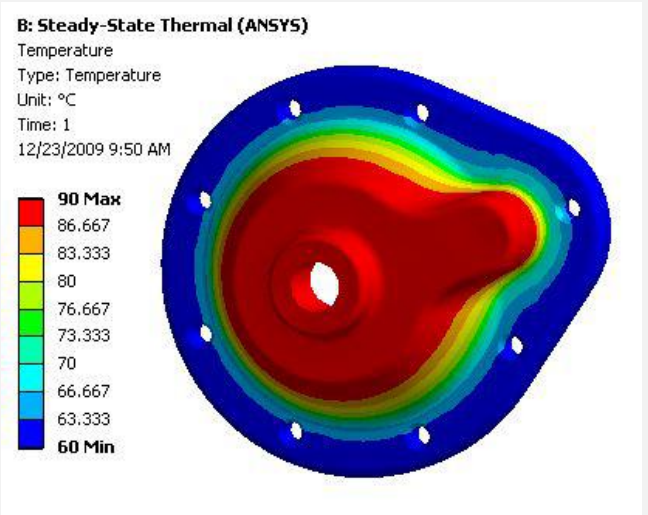




A. Results for polyethylene model.



B. Results for aluminum alloy model



FOR MORE DETAILS VISIT US @

www.vidsunglobalengineering.com

Global Engineering Solutions

Serving Globally