Introduction to Finite Element Analysis with Simulation Mechanical

Sualp Ozel PE
Product Manager

Class summary

This class will cover the basics of finite element analysis (FEA) and explore examples (including linear stress, modal-to-heat transfer, and dynamic simulations) in order to gain additional insight and improve product design for automotive, oil and gas, and industrial machinery industries.

Key learning objectives

At the end of this class, you will be able to:

- Understand the basics of finite element analysis
- Learn how to generate a finite element mesh
- Learn how to set up a problem
- Learn how to post process results

Agenda

- Session Approach
- Finite Element Analysis (FEA) Overview
- FEA Parameters
- FEA Best Practices
- FEA Software Introduction
- Analysis Walkthrough

Session Approach

This session is not just theoretical information… there's just not enough time to teach everyone matrix algebra

$$F = -kx$$

$$\Delta L = \frac{F}{EA} L = \frac{\sigma}{E} L.$$

$$G \stackrel{\text{def}}{=} \frac{\tau_{xy}}{\gamma_{xy}} = \frac{F/A}{\Delta x / l} = \frac{Fl}{A \Delta x}$$

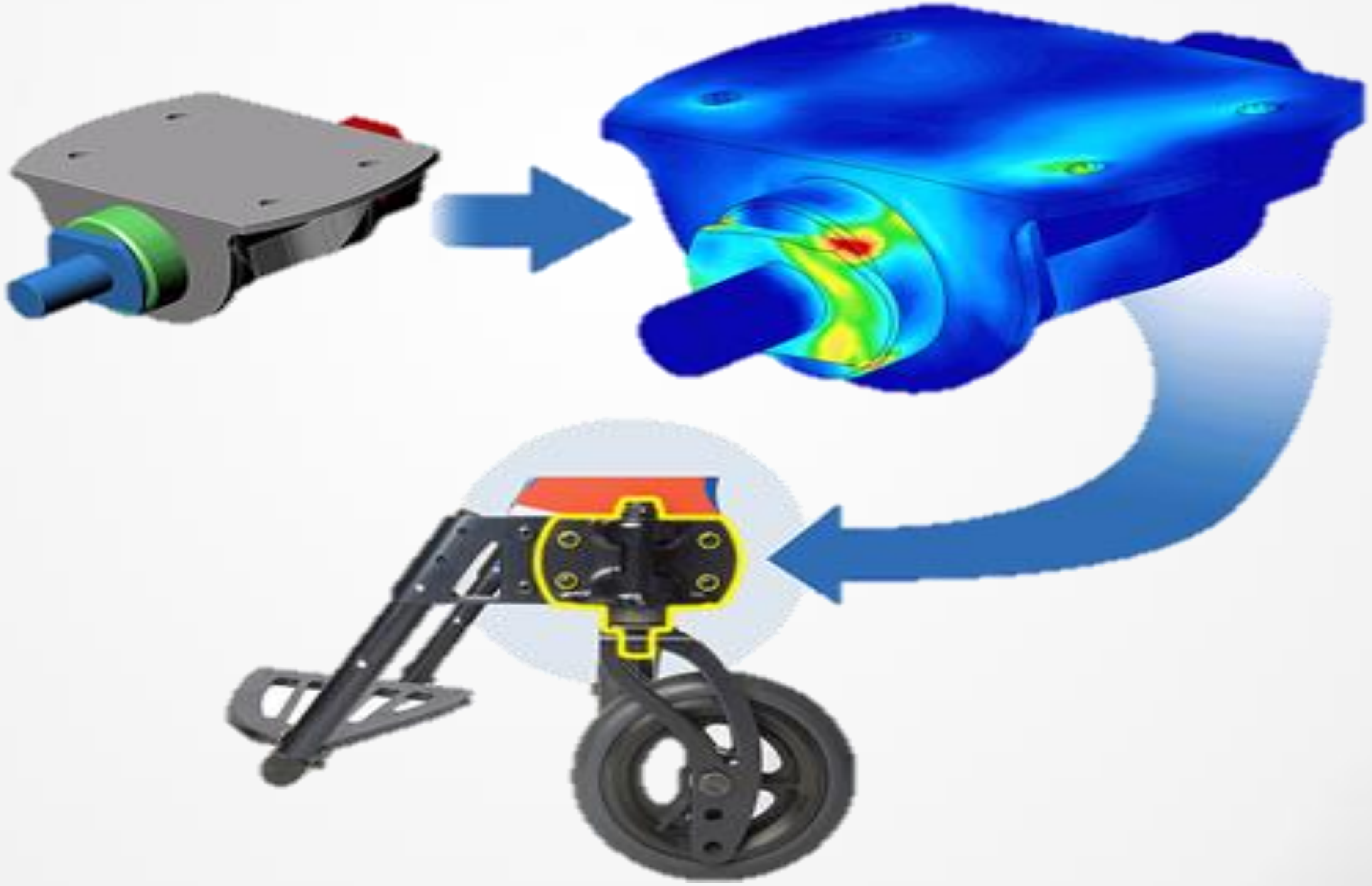
$$-\sum_{k=1}^{n} u_k \phi(v_k, v_j) = \sum_{k=1}^{n} f_k \int v_k v_j dx \text{ for } j = 1, \ldots, n. \quad (4)$$

$$\nu = -\frac{d\varepsilon_{\mathrm{trans}}}{d\varepsilon_{\mathrm{axial}}} = -\frac{d\varepsilon_{\mathrm{y}}}{d\varepsilon_{\mathrm{x}}} = -\frac{d\varepsilon_{\mathrm{z}}}{d\varepsilon_{\mathrm{x}}}$$

$$\begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ 2\varepsilon_{23} \\ 2\varepsilon_{31} \\ 2\varepsilon_{12} \end{bmatrix} = \begin{bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{23} \\ \gamma_{31} \\ \gamma_{12} \end{bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2(1+\nu) & 0 & 0 \\ 0 & 0 & 0 & 0 & 2(1+\nu) & 0 \\ 0 & 0 & 0 & 0 & 0 & 2(1+\nu) \end{bmatrix} \begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{23} \\ \sigma_{31} \\ \sigma_{12} \end{bmatrix}$$

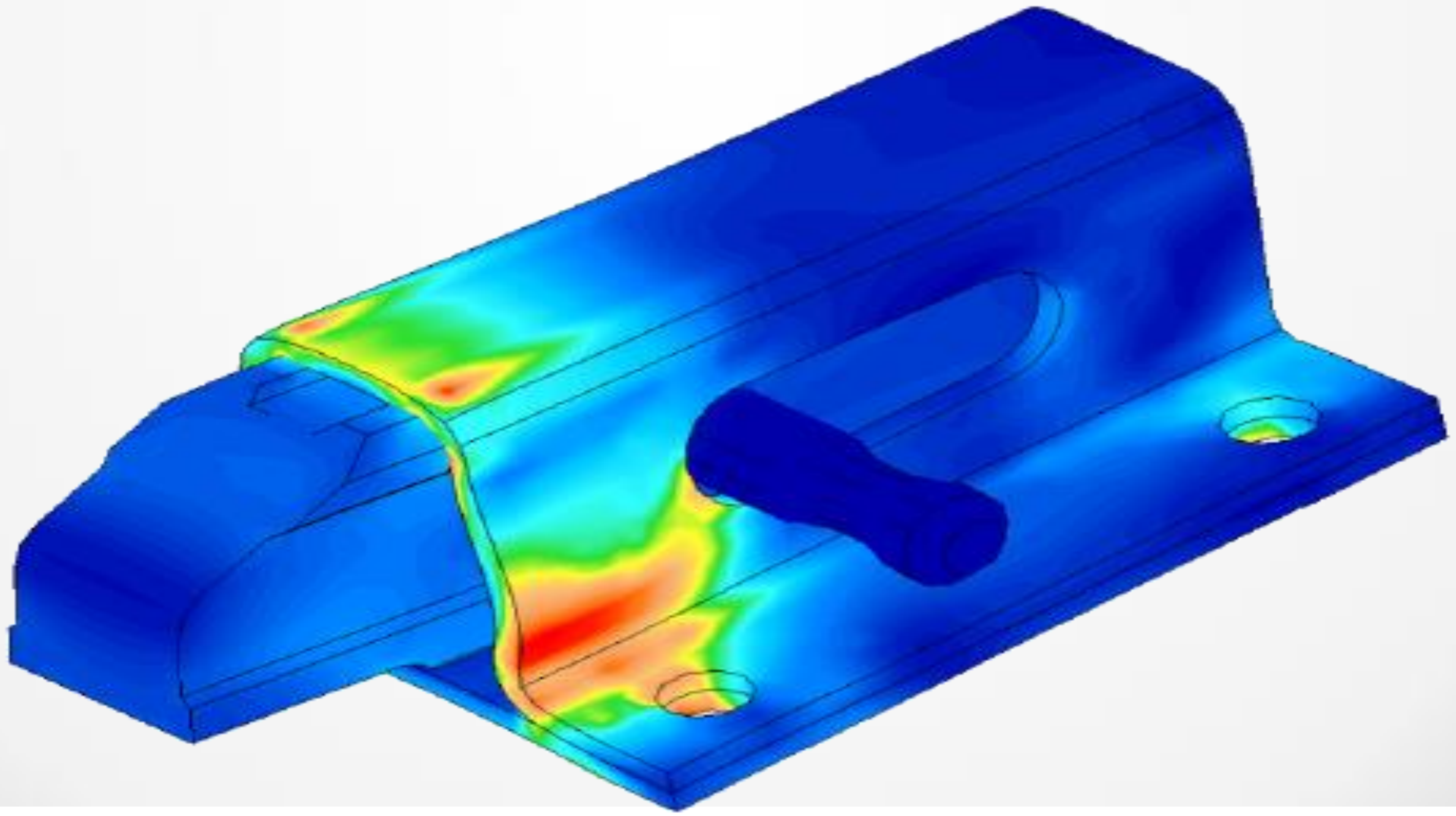
Session Approach

- We are going to apply and leverage modern technology to gain insights into the use cases and capabilities



FEA Description

Finite Element Analysis (FEA) is a computerized method for predicting how a real-world object will react to forces, vibration, heat, etc. to determine whether or not it will function as planned



FEA Benefits

- Predict Product Performance
- Reduce Raw Materials
- Ensure Optimal Design
- Verification
- Reduce Manual Testing and Prototypes
- Test What If Scenarios
- Shorten Design Cycle

Who Uses Simulation?

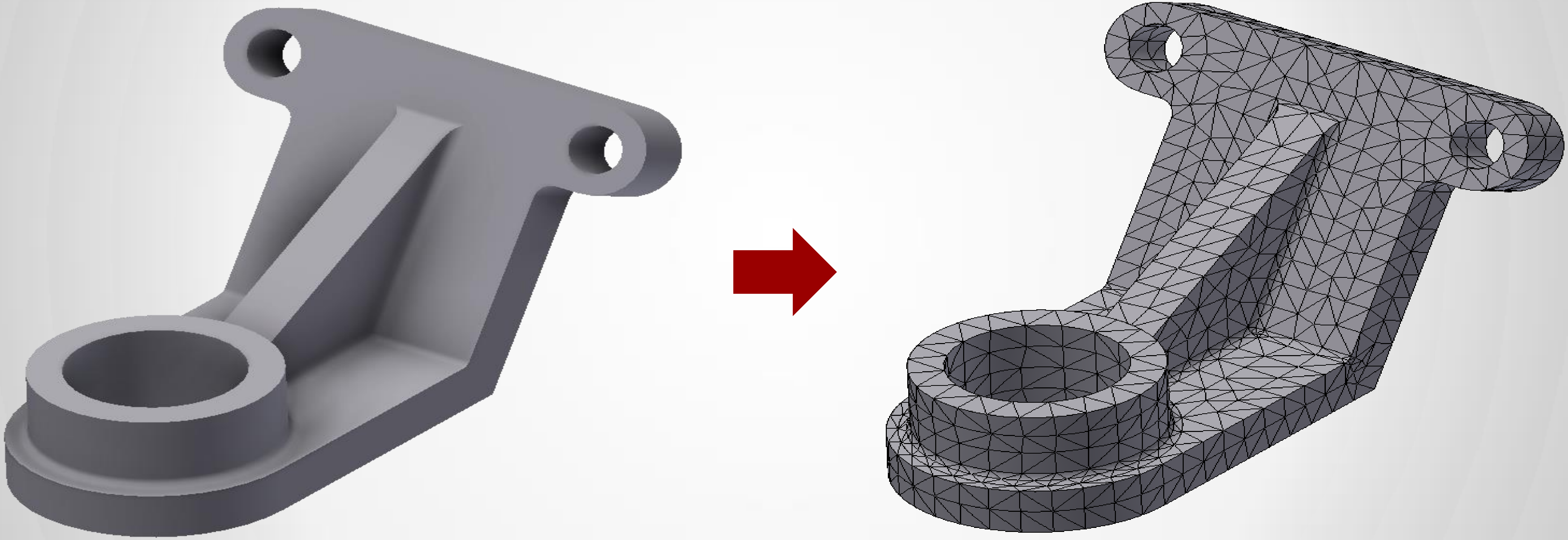
The Engineer

The Analyst

FEA Process Overview

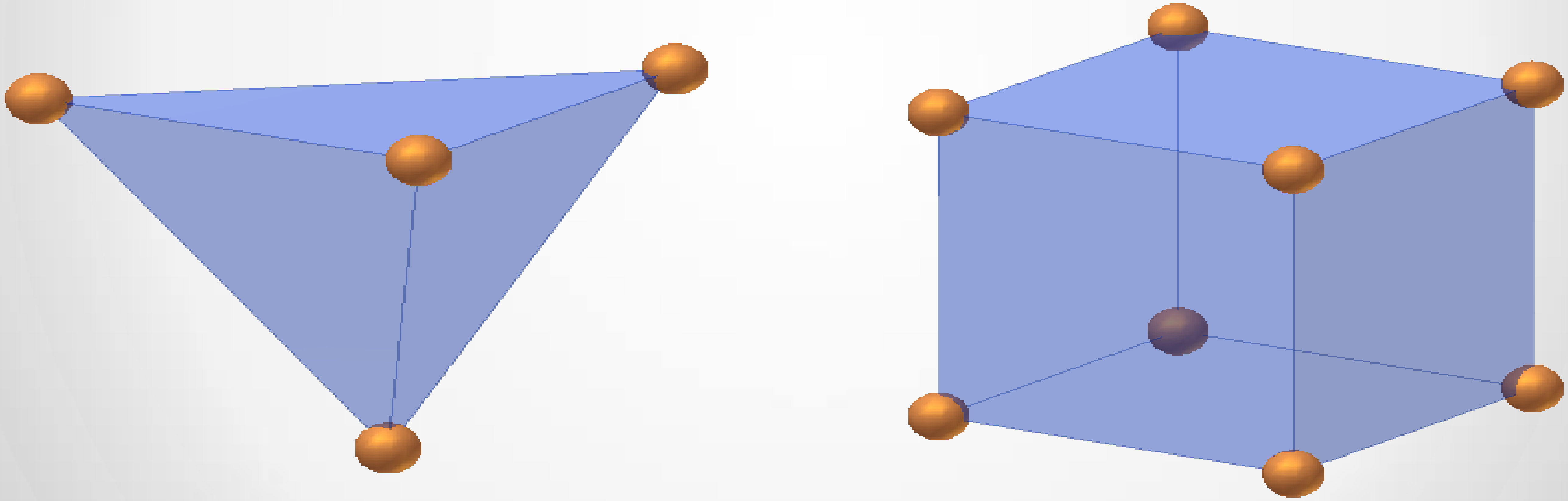
1. CAD Model Creation
2. Simulation Setup
3. Solve Simulation
4. Review Results
5. Optimize

Meshed 3D Model Example



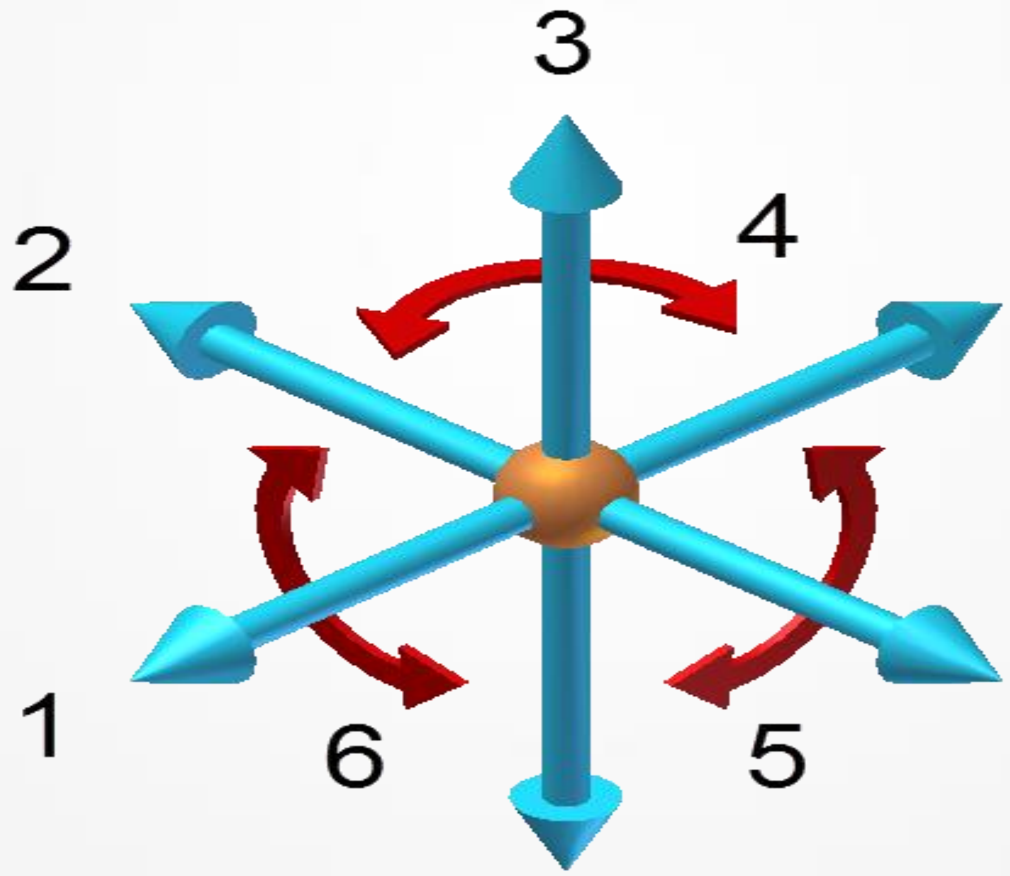
Element Overview

- An element is a mathematical relation that defines how the DOFs of one node relate to the next.



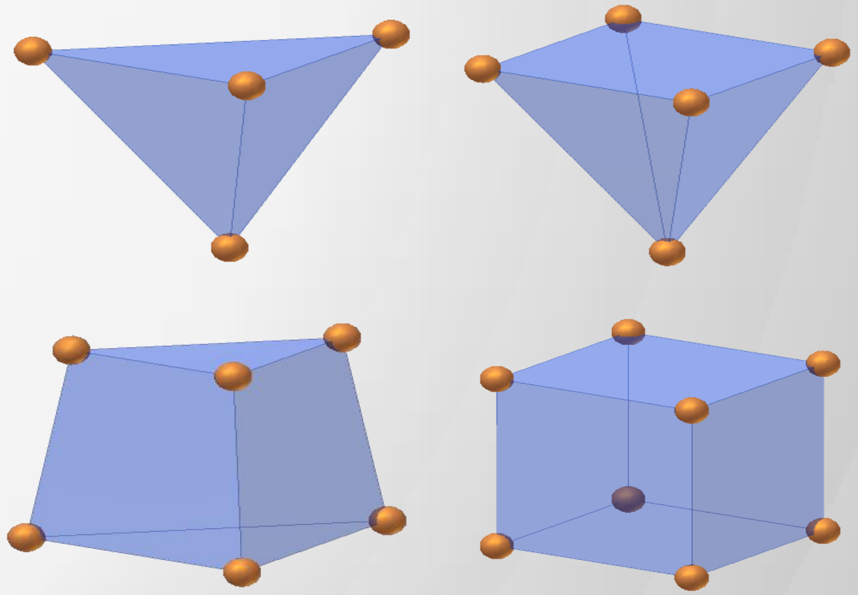
Node Overview

- A node is a coordinate location in space where the Degrees of Freedom (DOFs) are defined.



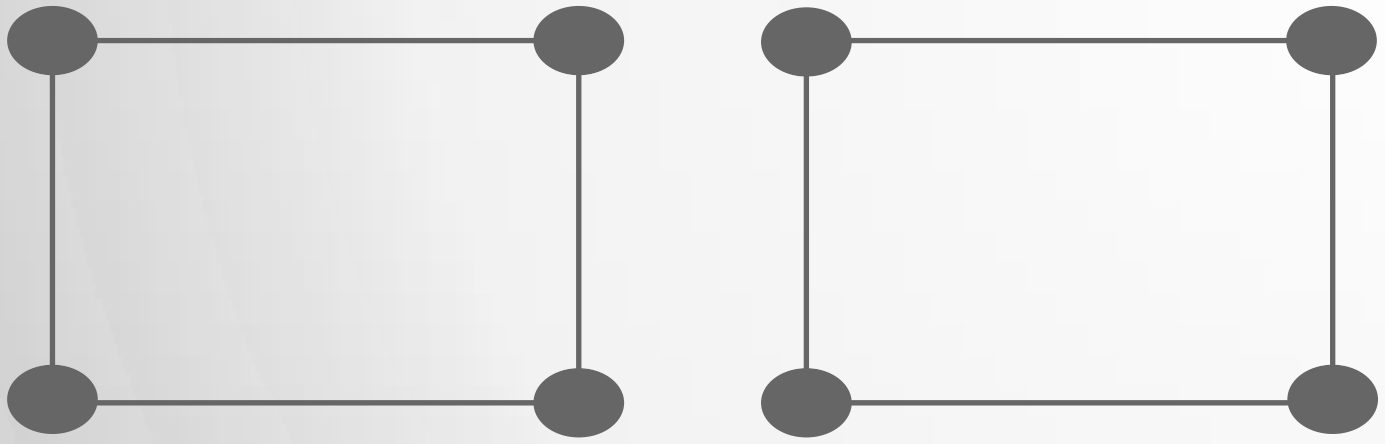
Types of Elements

- 1D elements
  - A line connecting 2 nodes only for items like beams and springs
- 2D elements
  - Planar or axisymmetric elements with either three or four edges enclosing an area
  - Plates or Shell Elements: Planar elements that are triangular or quadrilateral with a specified thickness
- 3D (solid) elements
  - Enclosed 3D volumes with 4, 5, 6 or 8 corner nodes

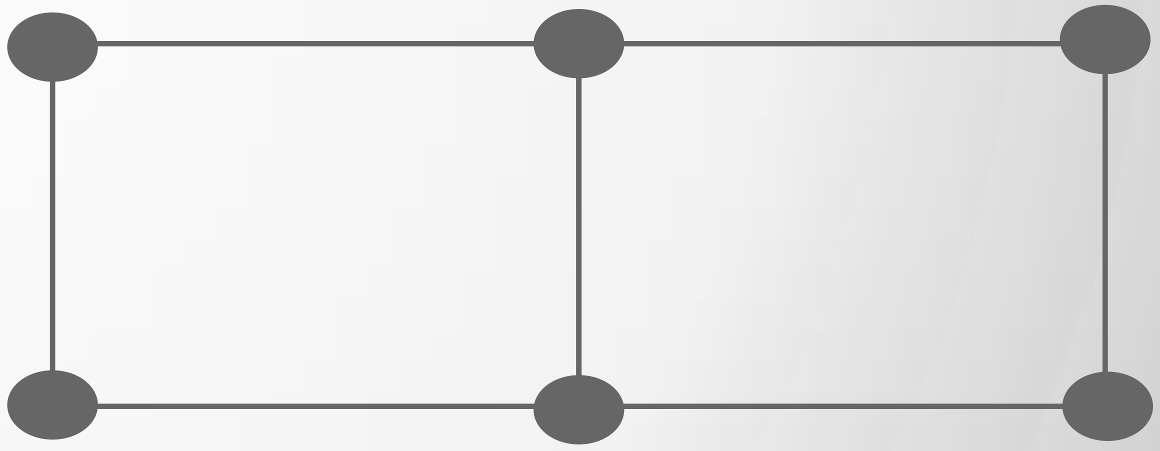


Standard Element Bonding

- Elements within a single part body can only communicate to one another via common nodes for transferring data information



No Connection



Connected Nodes

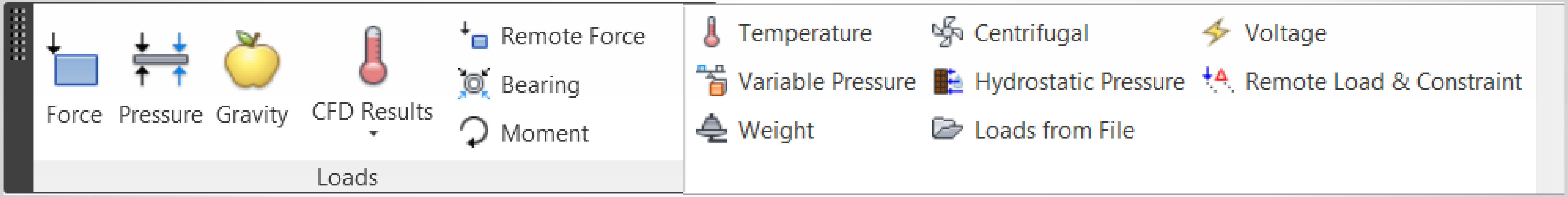
Material Assignment

- Material properties define the structure characteristics of the part
- Material property information can be located at matweb.com

0.046 lbmass/in^3	Density
478.955 ksi	Young's Modulus
0.360	Poisson's Ratio
0.000E+000 psi	Yield Strength
1.655E+004 psi	Ultimate Tensile Strength
2.922 btu in/(ft^2 hr f)	Thermal Conductivity
100.800 microin/(in f)	Linear Expansion
0.000E+000 btu/(lbmass f)	Specific Heat

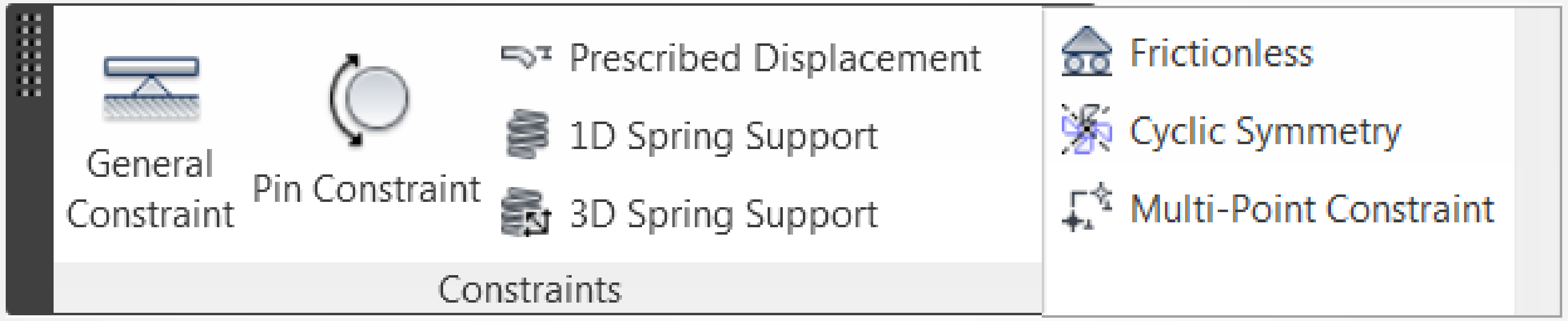
Loads

Structural loads are forces applied to a part or assembly during operation and cause the model to displace, deflect, and induce stresses and strains



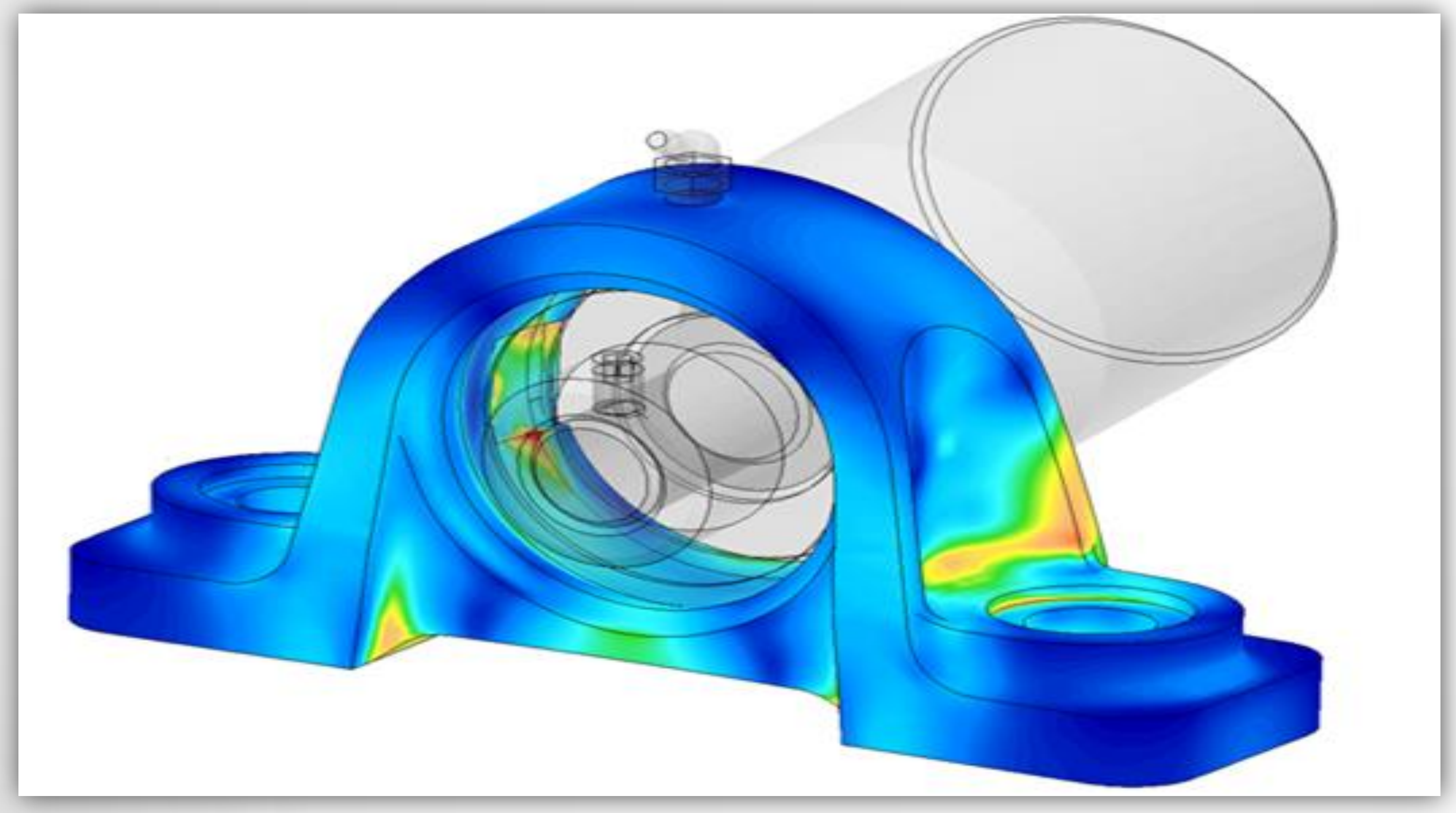
Constraints

Structural constraints restrict or limit the displacement of the model mesh nodes



Contact Conditions

Contact conditions are used to establish relationships between the nodes of contacting parts within an assembly



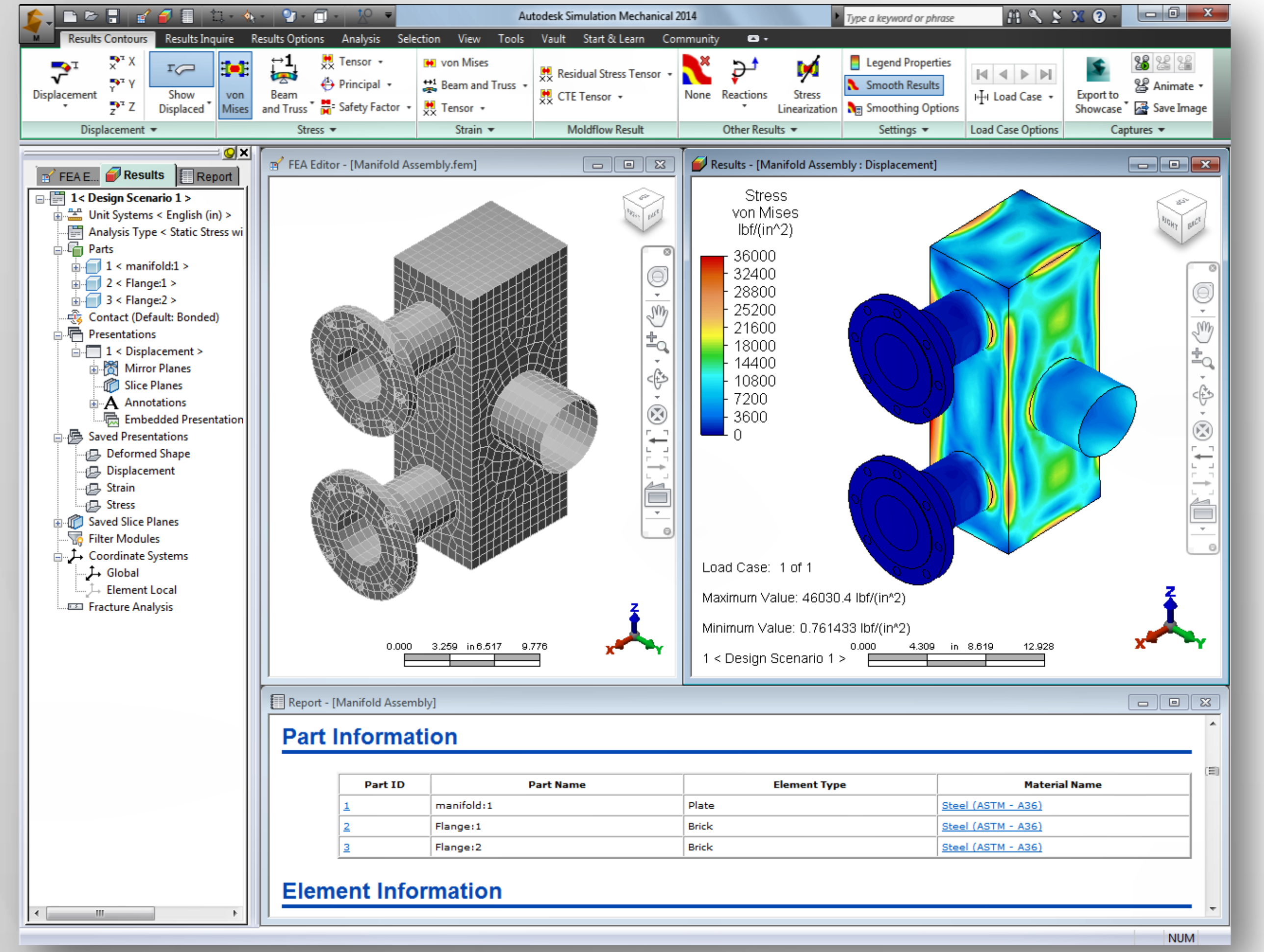
- Bonded
- Welded
- Free/No Contact
- Surface Contact
- Sliding/No Separation
- Separation/No Sliding
- Shrink Fit/Sliding
- Shrink Fit/No Sliding
- Edge Contact

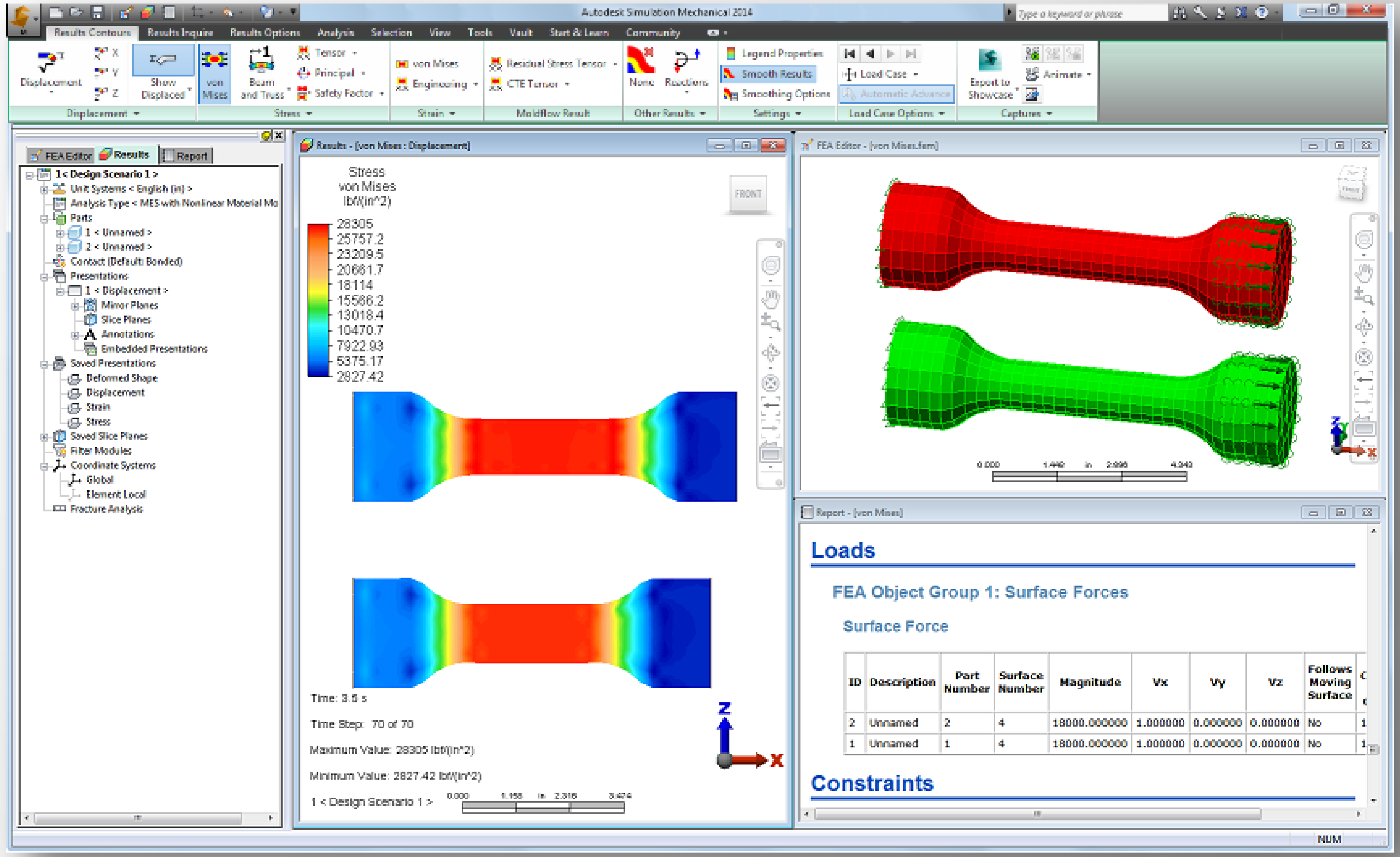
Simulation Solving

Running or solving the simulation processes and calculates the results based on the parameters established

Results

The simulation results can be reviewed and exported as a report to make intelligent decisions





Reviewing Results

- Simulation does not always replace the need for physical testing
- The engineer / analyst still needs to interpret the results to make final decisions

Analysis Types

- Linear ⟵ Focus for this presentation
- Nonlinear
- Thermal / Electrostatic
- Natural Frequency / Modal Analysis
- Vibration
- Fatigue Analysis

Linear vs. Nonlinear

- Linear ← Focus for this presentation
  - Structure returns to original form
  - Small changes in shape stiffness
  - No changes in loading direction or magnitude
  - Material properties do not change
  - Small deformation and strain
- Nonlinear
  - Geometry changes resulting in stiffness change
  - Material deformation that may not return to original form
  - Supports changes in load direction and constraint locations
  - Support of nonlinear load curves

Mild Steel Material Properties

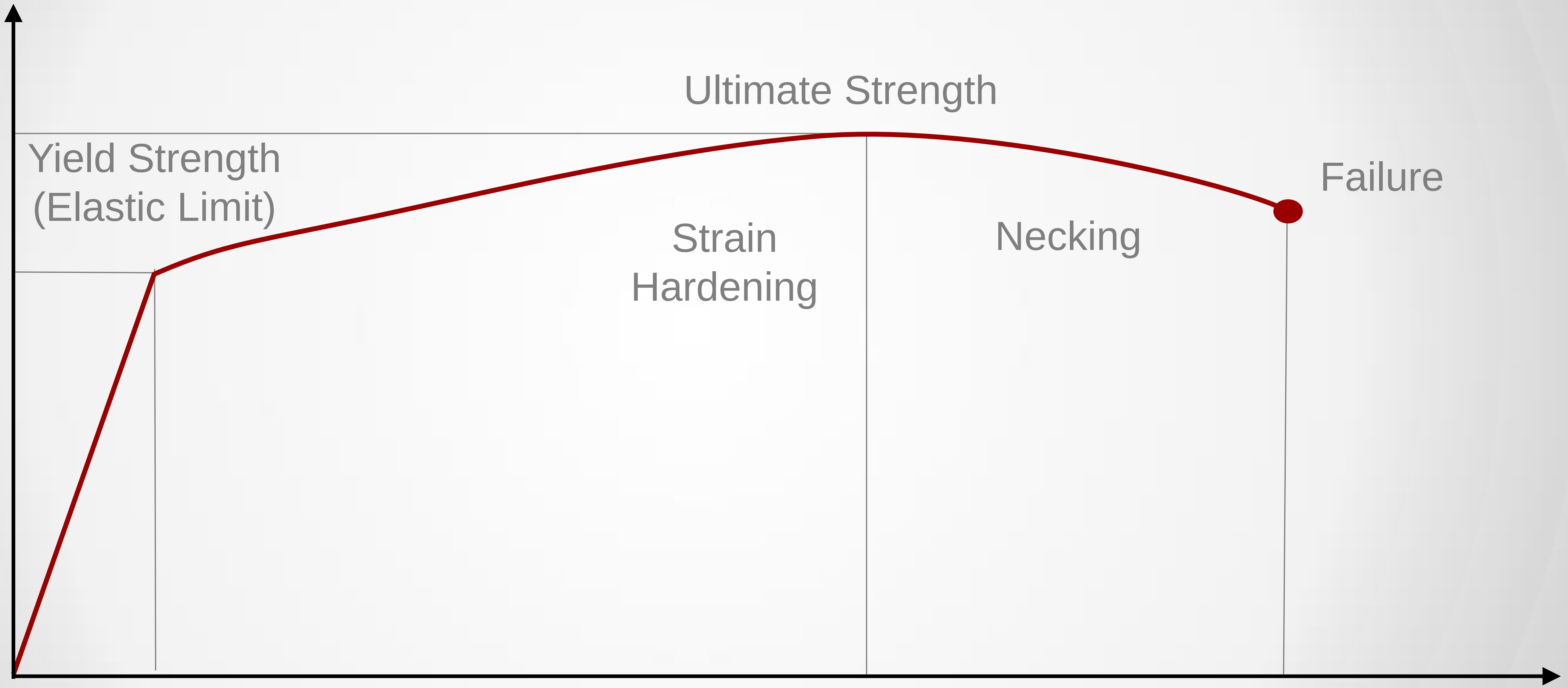
- Density = 0.284 lbmass/in^3
- Young's Modulus = 3.193E+4 ksi
- Poisson's Ratio = 0.275

- Yield Strength = 3.004E+4 psi
- Ultimate Tensile Strength = 5.007E+4 psi

- Thermal Conductivity = 1.259E+3 btu in/(ft^2 hr*F)
- Linear Expansion = 21.600E-6 in/(in*F)
- Specific Heat = 0.356 btu/(lbmass*F)

Mild Steel Stress Strain Curve

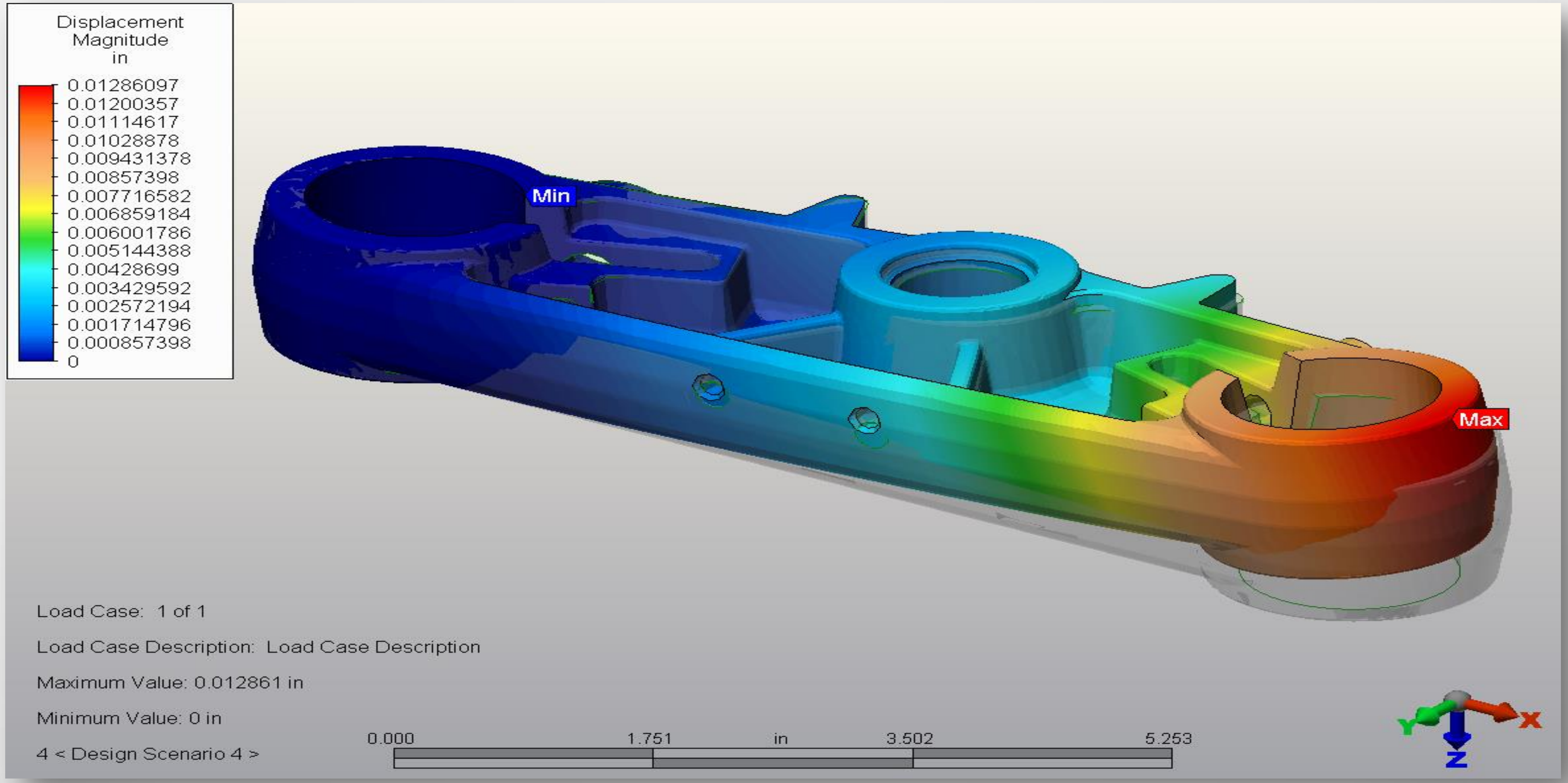
$$\textbf{Stress} = \frac{\text{Force}}{\text{Area}}$$



$$\textbf{Strain} = \frac{\text{Change in Length}}{\text{Original Length}}$$

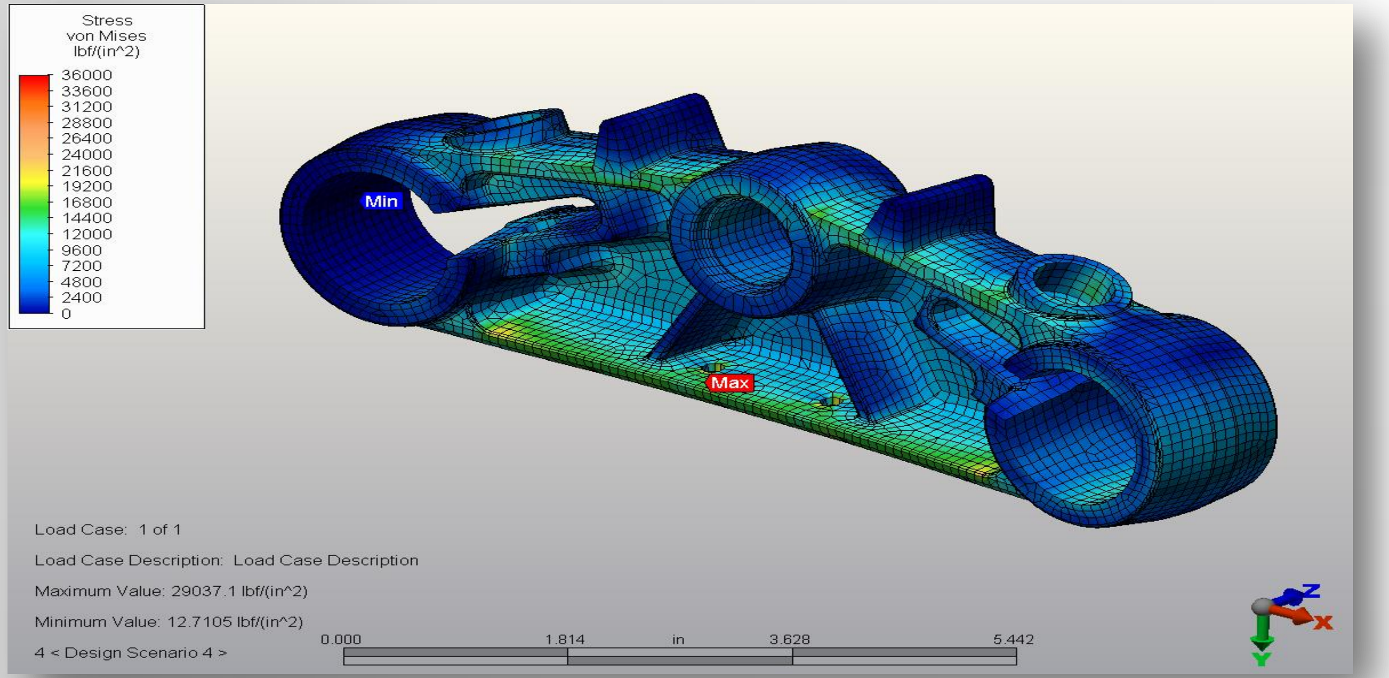
Displacement

The displacement results show the magnitude of the model deformation from the original shape



Von Mises Stress

Formula for combining three principal stresses into an equivalent stress to compare to the material stress properties



The equation used is:

$$\sqrt{0.5\left[(S_x - S_y)^2 + (S_y - S_z)^2 + (S_z - S_x)^2\right] + 3\left(S_{xy}^2 + S_{yz}^2 + S_{zx}^2\right)}$$

where Sx, Sy, and Sz are the axial stresses in the global directions, and Sxy, Syz, and Sxz are the shear stresses.

In terms of the principal stresses S1, S2 and S3:

$$\sqrt{0.5\left[(S_1 - S_2)^2 + (S_2 - S_3)^2 + (S_3 - S_1)^2\right]}$$

Note: von Mises value is always positive.

Safety Factor

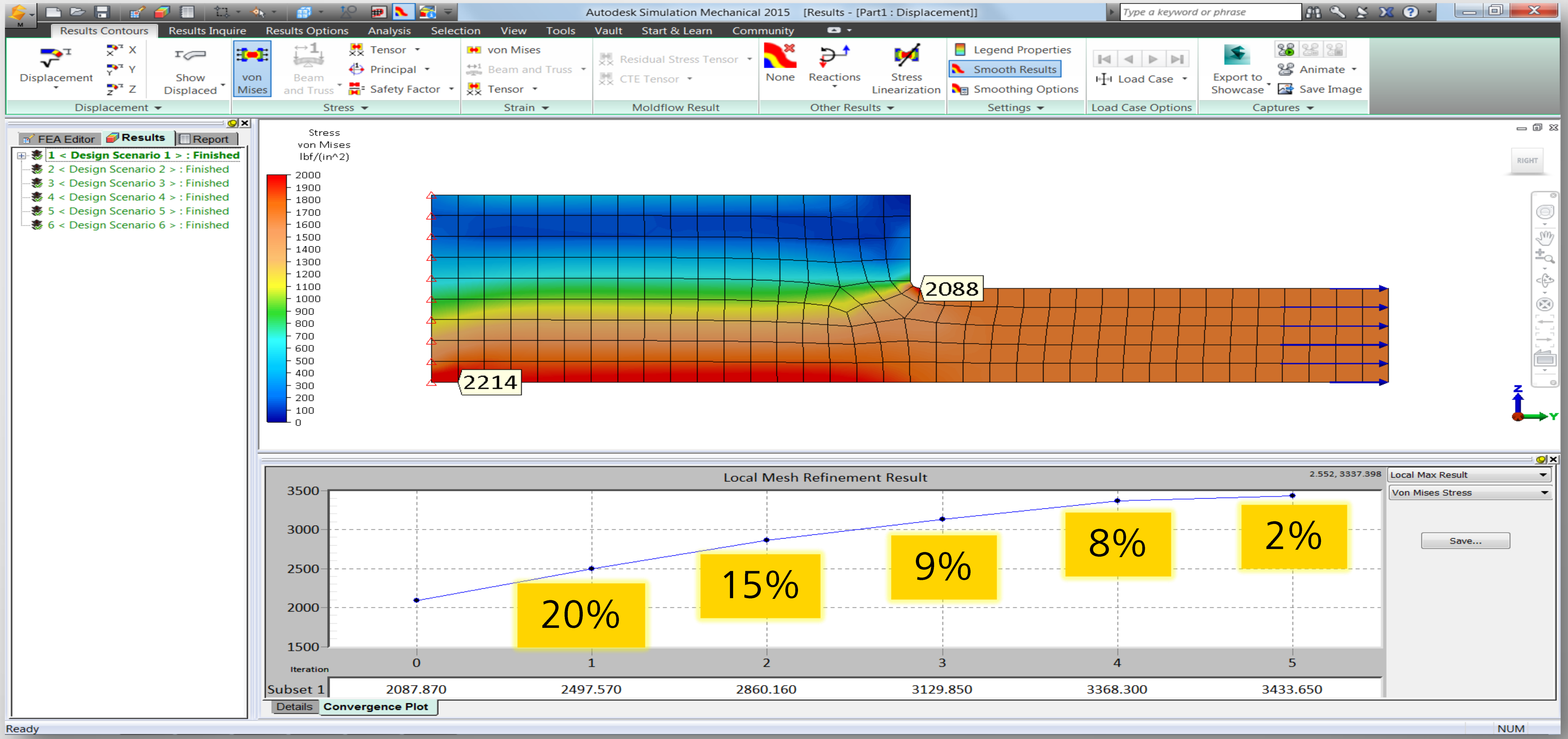
Provides a ratio of how much stronger the object is than it usually needs to be for an intended load

$$\text{Safety Factor} = \frac{\text{Material Yield Strength}}{\text{Maximum Von Mises Stress}}$$

$$2 = \frac{40{,}000 \text{ psi}}{20{,}000 \text{ psi}}$$

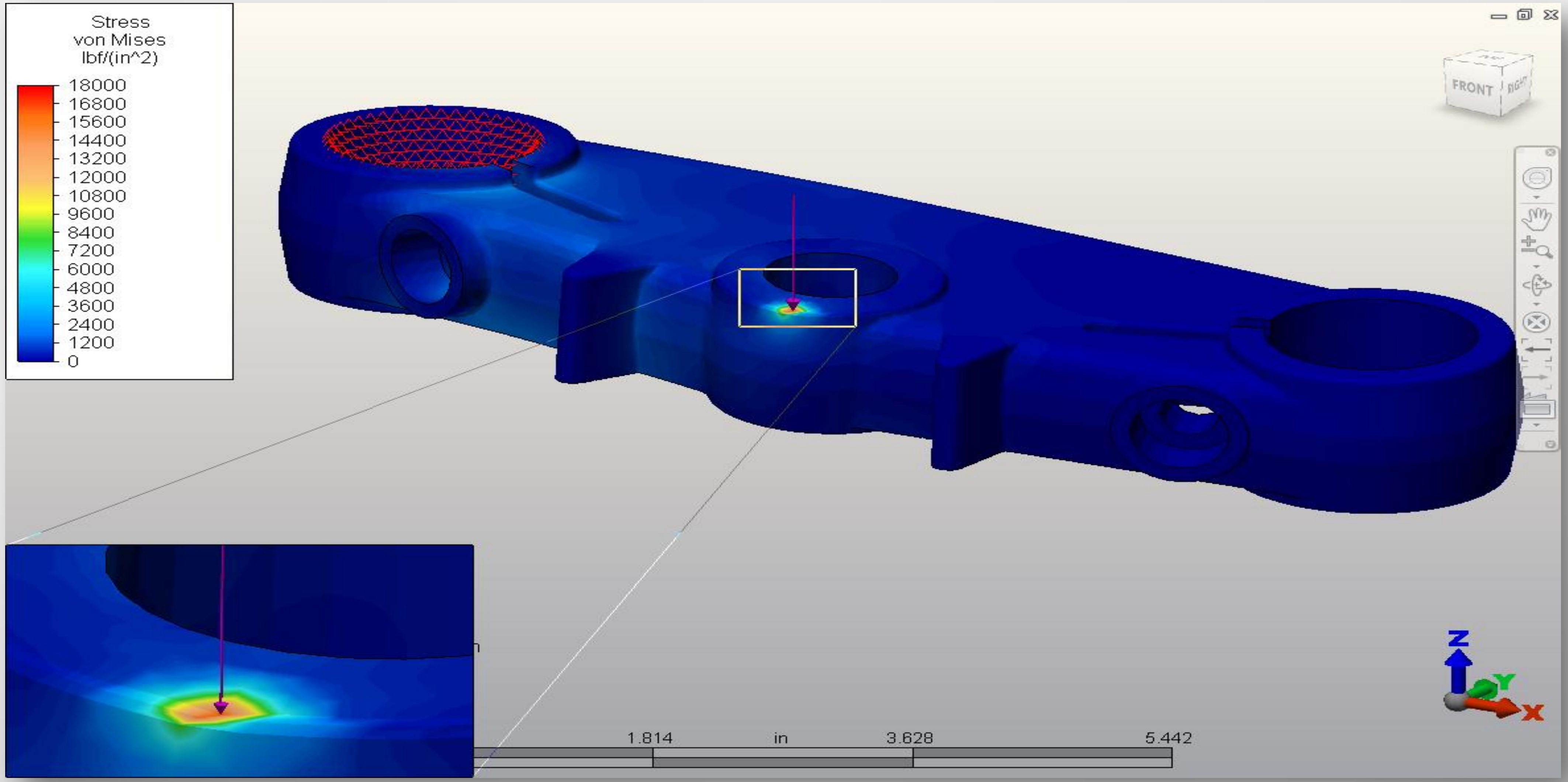
Convergence

Convergence is the process of altering element sizes in high stress areas to ensure the specified result criteria has converged



Stress Singularities

A localized high stress area where the stress becomes infinite resulting in distorted results



Best Practices

- Setup simulation to match real world
- Verify material properties
- Use engineering knowledge judgment
- Avoid putting loads on nodes or small edges
- Ensure solution type (Linear / Nonlinear)
- Identify stress singularities
- Ensure your results converge

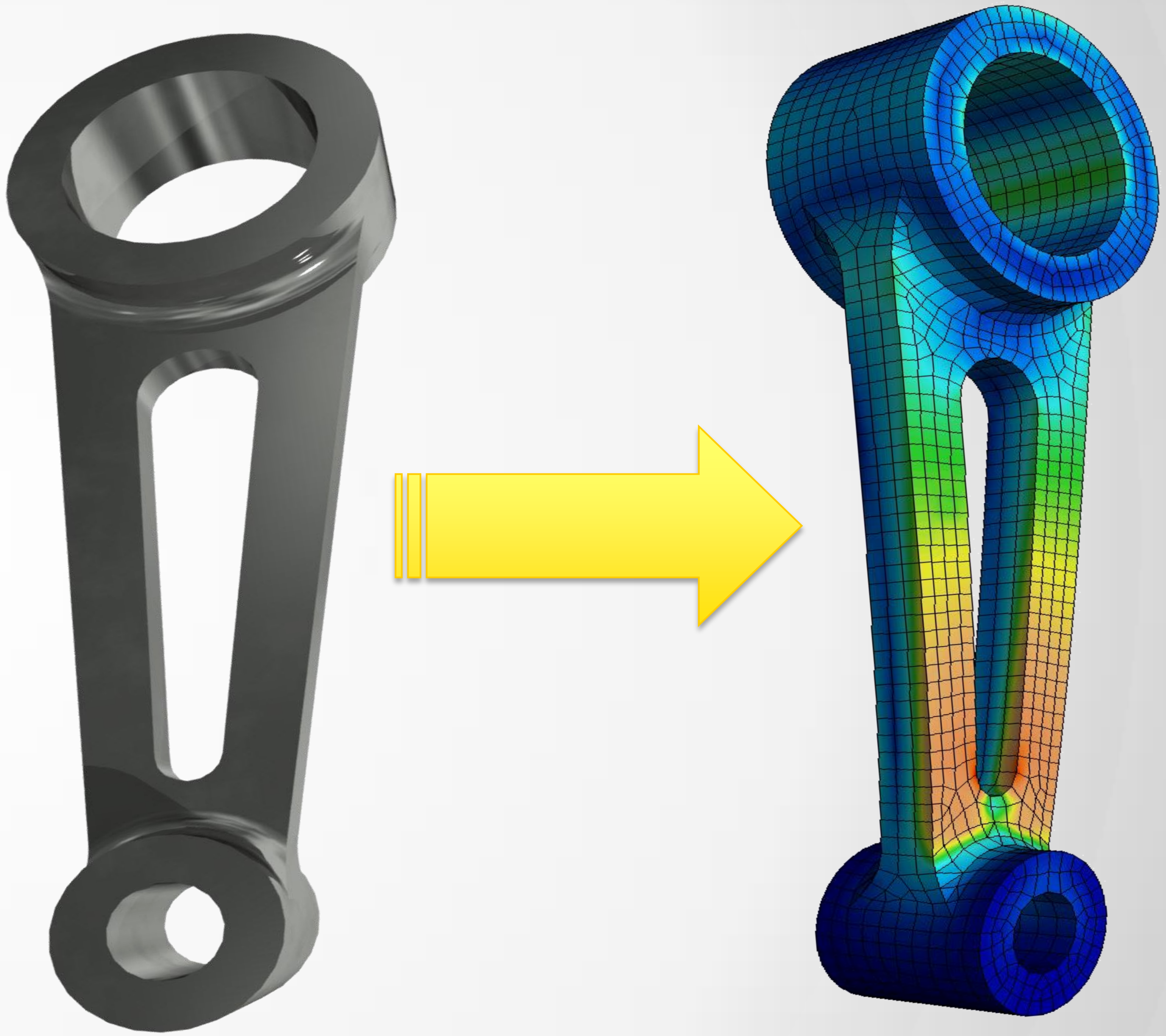
FEA Software

- Conceptual Simulation (Apps)
- FEA Features Built into Design Tools
- Purpose Built Simulation Software ⟵ Focus for this presentation

Autodesk Simulation Mechanical

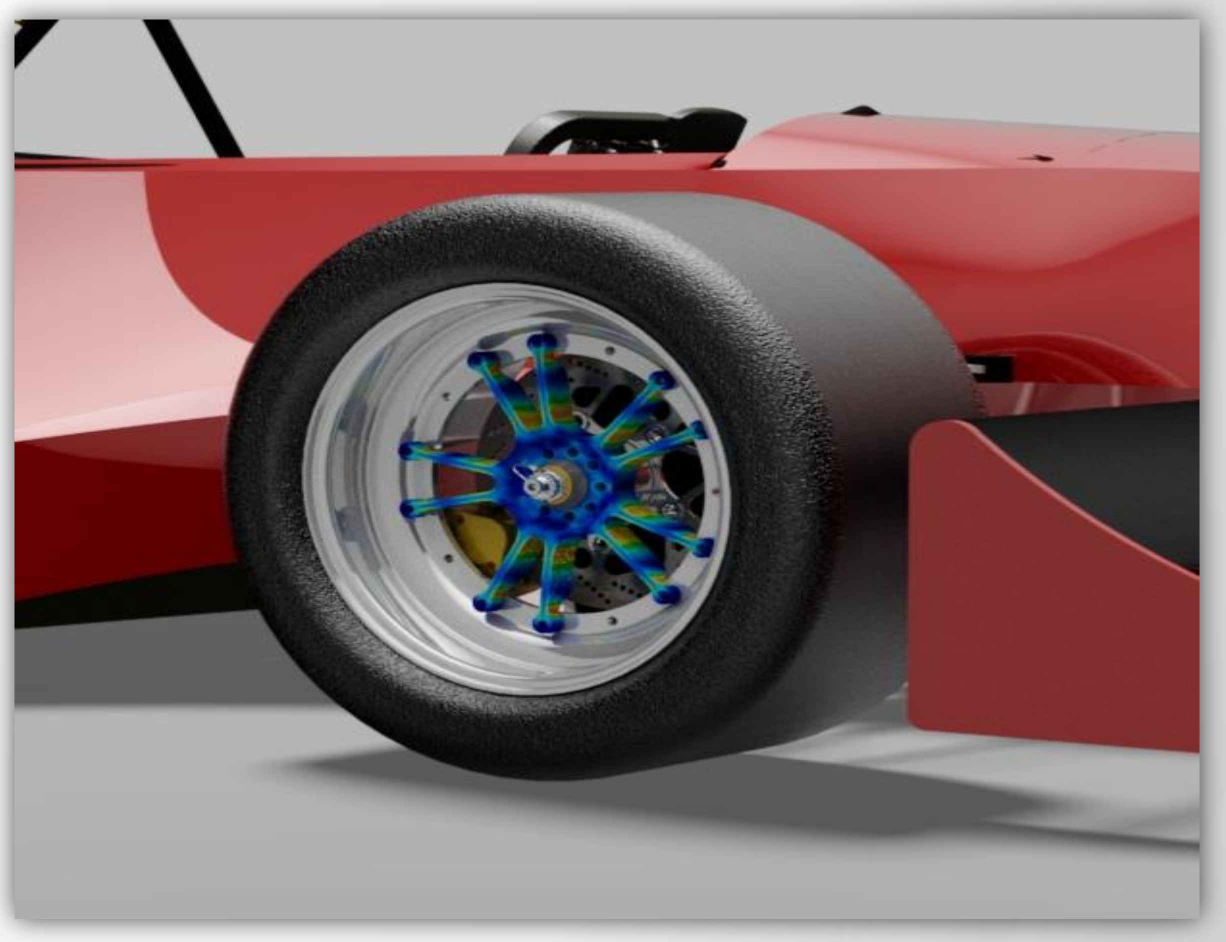
Walkthrough 1

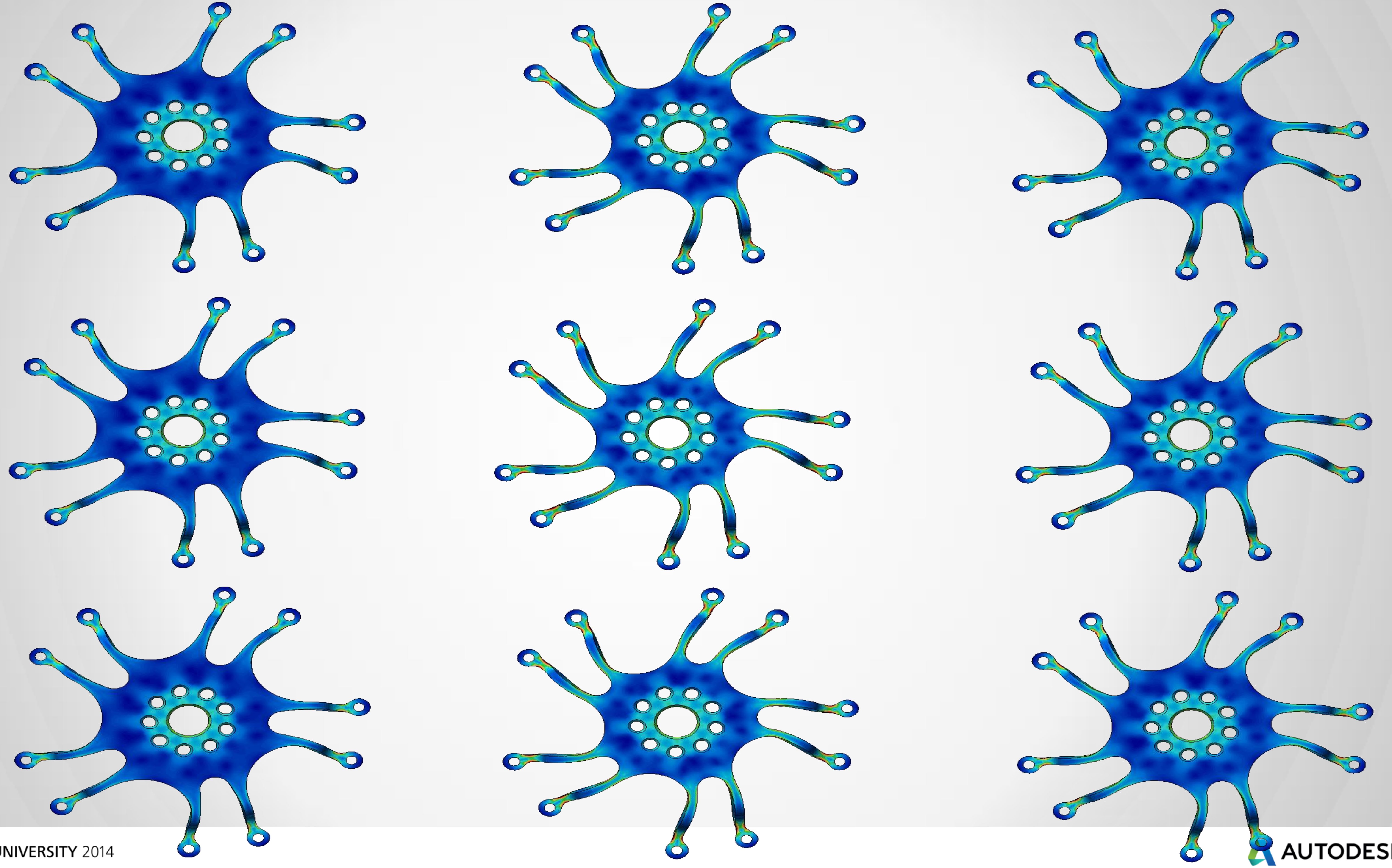
- Importing geometry
- Assigning loads and constraints



Walkthrough 2

- Evaluating results
- Results based optimization





Recap

- Finite Element Analysis (FEA) Overview
- FEA Parameters
- FEA Best Practices
- FEA Software Introduction
- Analysis Walkthrough

Thank You!

Questions?

- Contact: sualp.ozel@autodesk.com
- Products: http://autodesk.com/simulation
- Community: http://simhub.autodesk.com

Session Feedback

- Via the Survey Stations, email or mobile device
- AU 2015 passes given out each day!
- Best to do it right after the session
- Instructors see results in real-time

Students, educators, and *schools* now have

FREE access to Autodesk design software & apps.

Download at www.autodesk.com/education

AUTODESK®