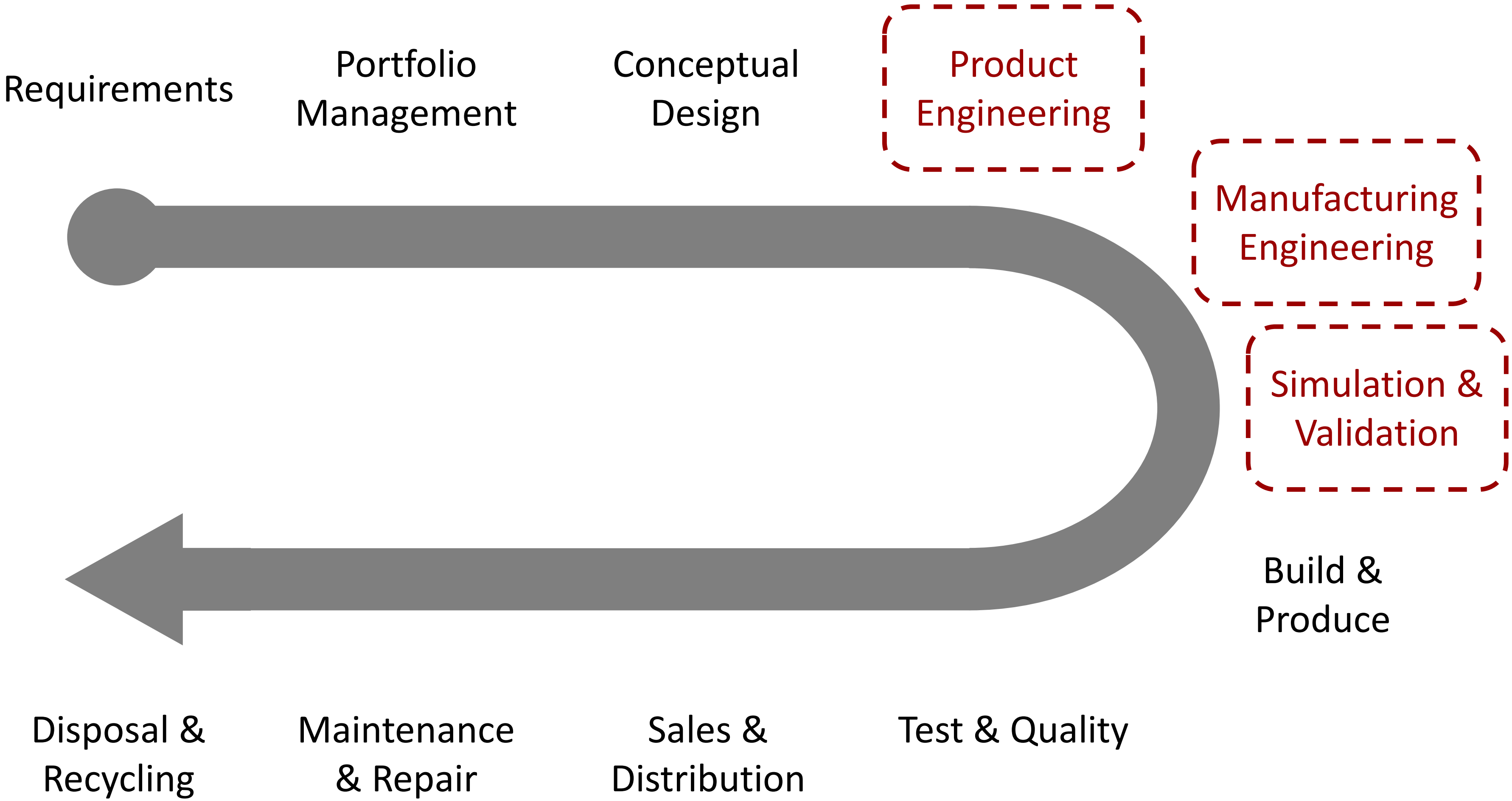
Week 9 - Lecture
Linear Structural Analysis

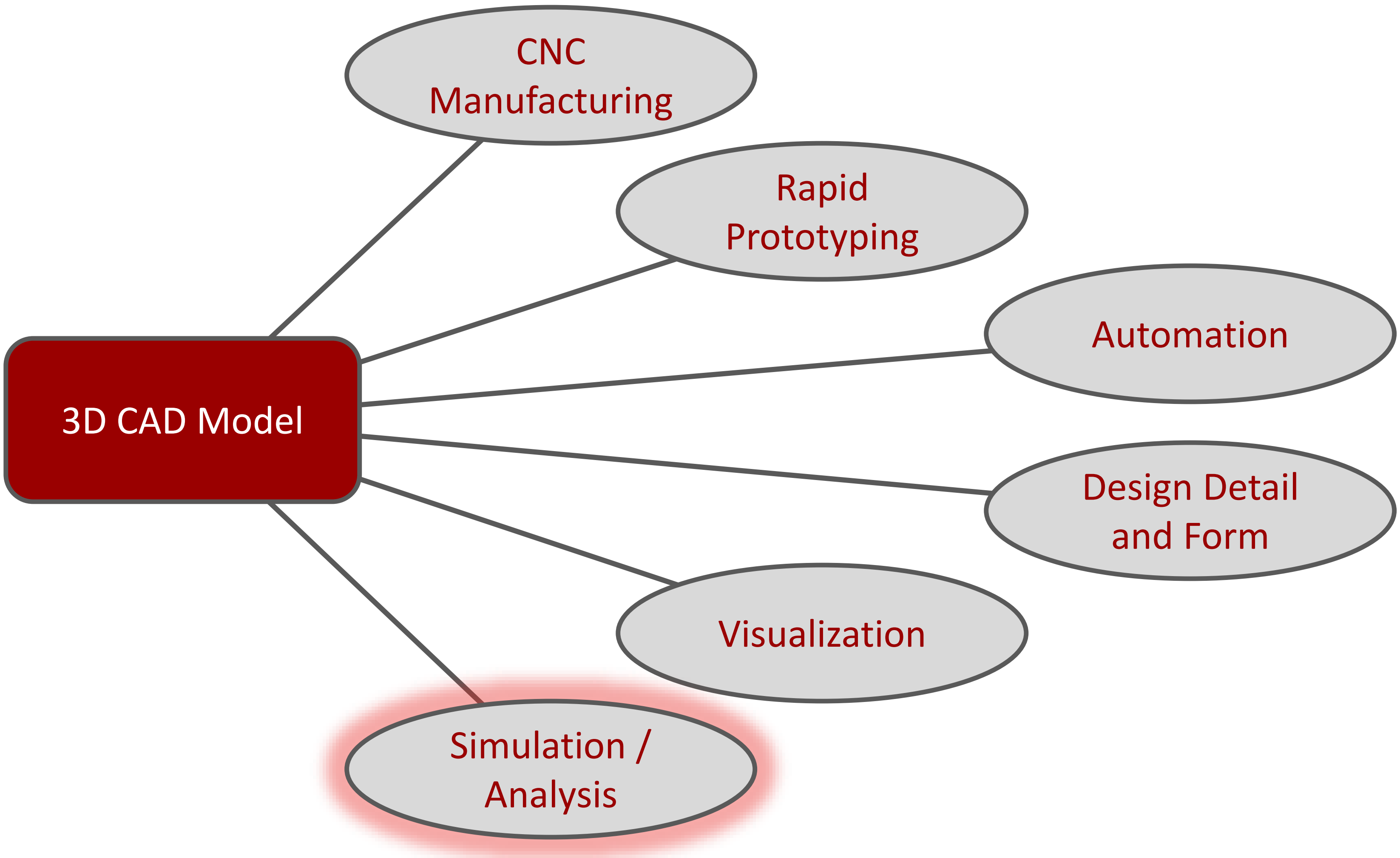
Lecture Topics

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

Product Lifecycle – Week 9

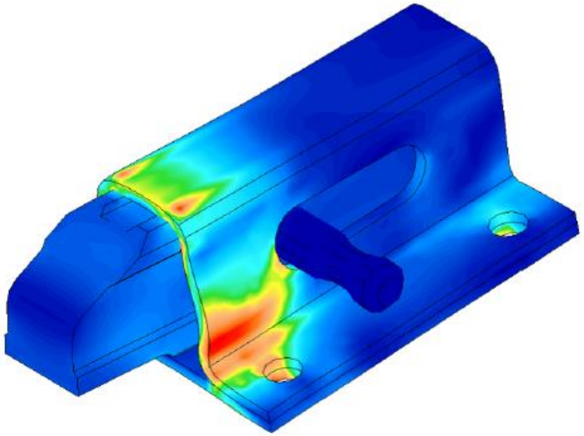


3D Design Use



What is FEA?

- Finite Element Analysis (FEA) is a computerized method for predicting how a real-world object will react to forces, vibration, heat, and etc. in terms of whether 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

Reasons for Adoption by the Masses

- Better Computing (Faster and Cloud-based)
- Affordable Software
- Easier-to-Use Software
- 3D Design Data has become common.
- The Need to Improve Products Further

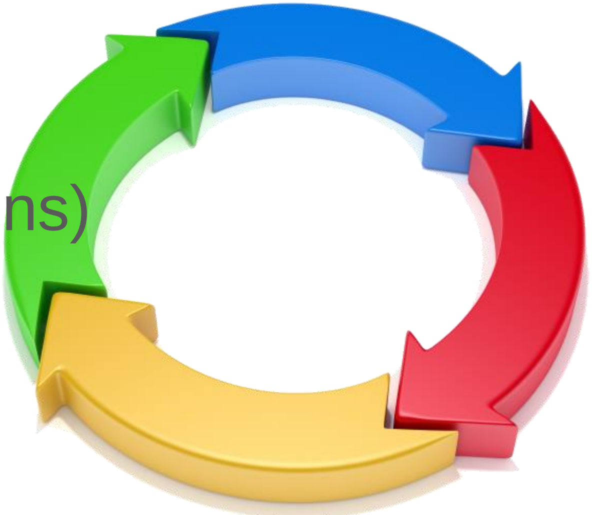
Who Uses Simulation?

Design Engineer

Analysis Engineer

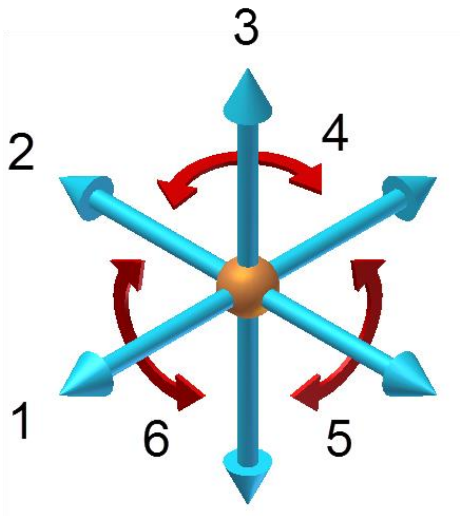
FEA Process Overview

1. CAD Model Input
2. Simulation Setup (Pre-process)
   – Analysis Type
   – Material Property Assignment
   – Add Constrains (Boundary Conditions)
   – Add Loads (Loading Conditions)
   – Mesh Generation
3. Solve Simulation
4. Review Results (Post-process)



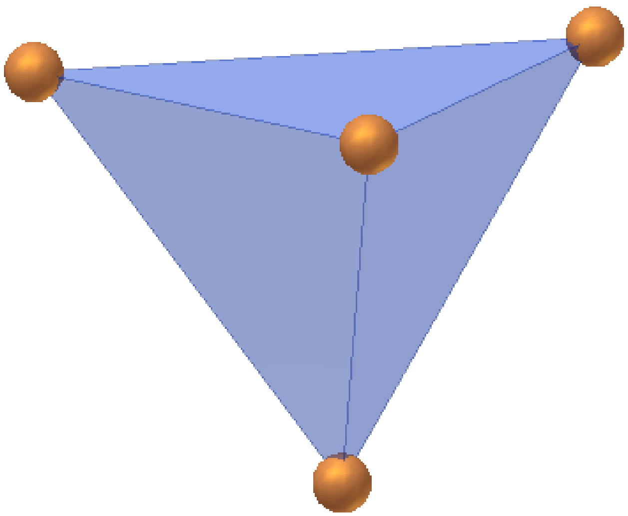
Node Overview

A node is a coordinate location in space where the Degrees of Freedom (DOFs) and physical property (stress, strain, temperature, velocity, etc.) are defined.



Element Overview

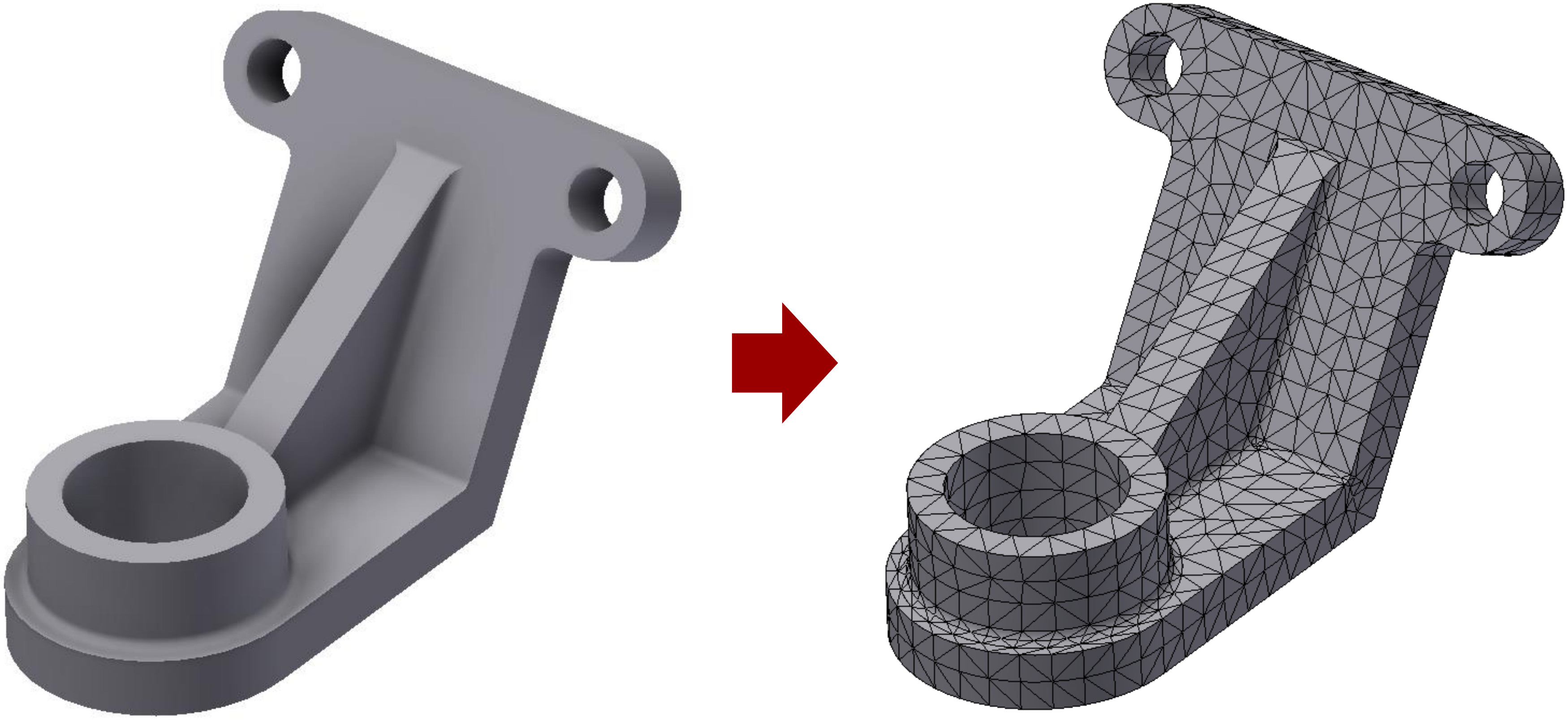
An element is a mathematical entity that defines how the shape and physical property of an internal point is interpolated from the node positions and physical properties.



How FEA Works

- Models are defined by nodes and elements forming a mesh.
- Governing engineering equations (PDE, ODE) are solved at the nodes and elements.
- A matrix equation, including terms from each element, is solved.
- Predicts changes within the element.
- The results are plotted on the model using colors and line plots.

Meshed 3D Model Example

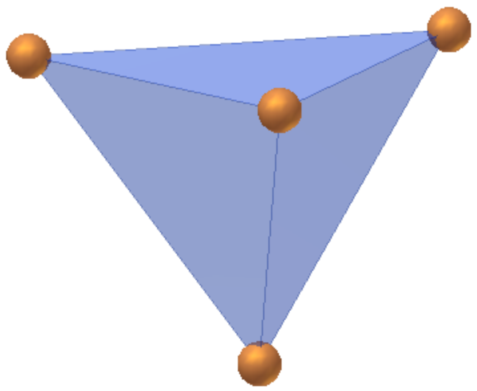


Types of Elements

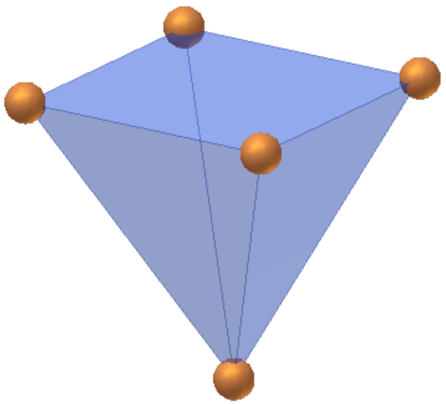
- Line Elements
  - A line connecting 2 nodes only for items like beams and springs.
- 2D Elements
  - Planar elements with either three or four edges enclosing an area.
- 3D Plates or Shell Elements
  - Planar elements that are triangular or quadrilateral with a specified thickness.
- Brick (Solid)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

Brick (Solid) Element Types

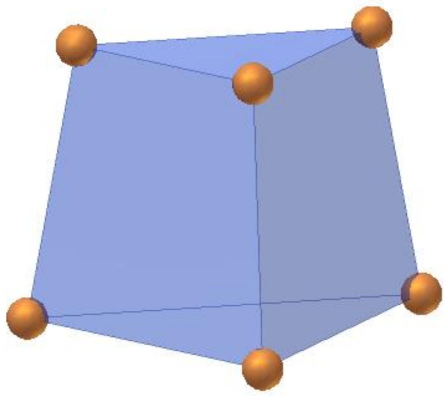
4-Noded Tetrahedral



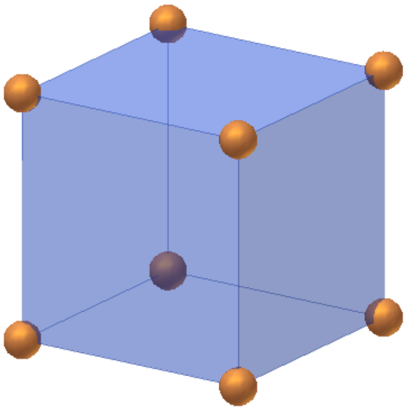
5-Noded Pyramid



6-Noded Wedge



8-Noded Brick



Material Assignment

- Material properties define the structure characteristics of the part.
- Material property information can be found on the web at www.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

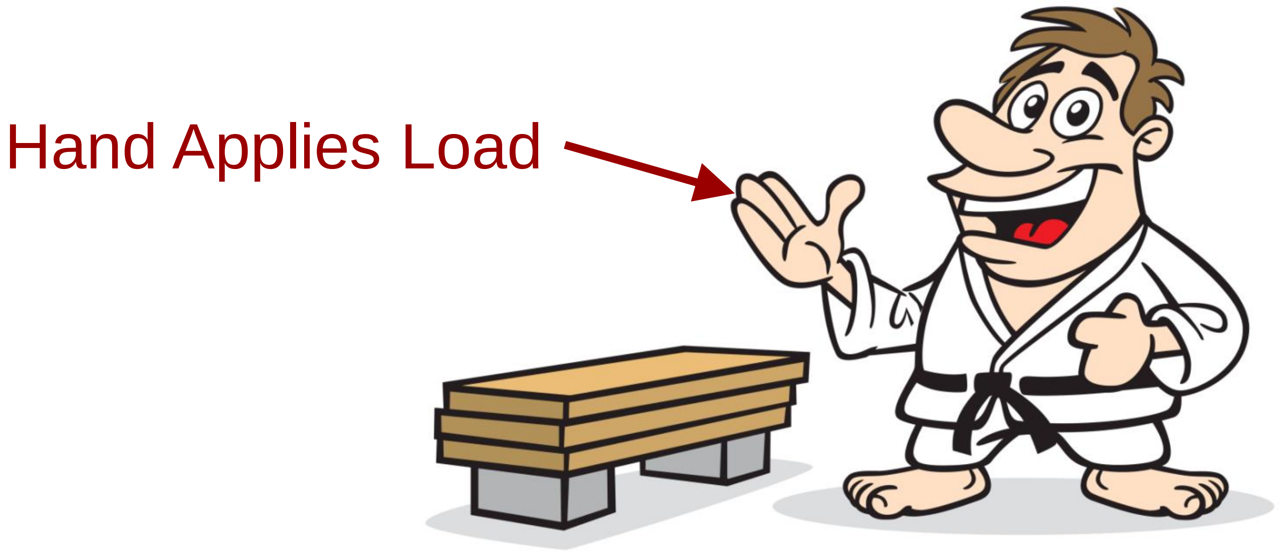
Constraints

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

Floor is Fixed Constraint

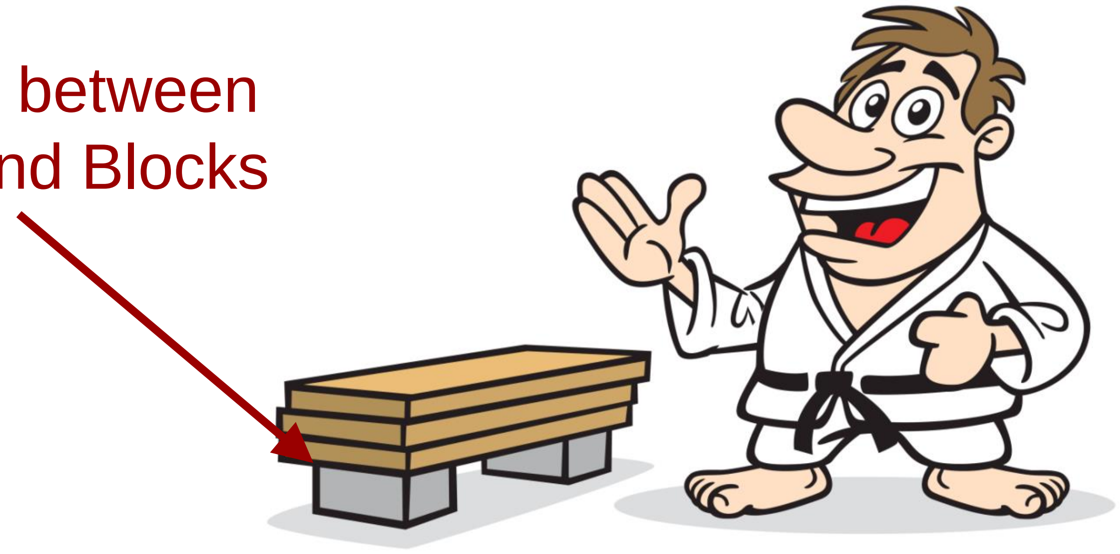
Loads

Structural loads are forces applied to a part or assembly during operation. Such loads cause stresses, deformations, and displacements in components.



Contact Conditions

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

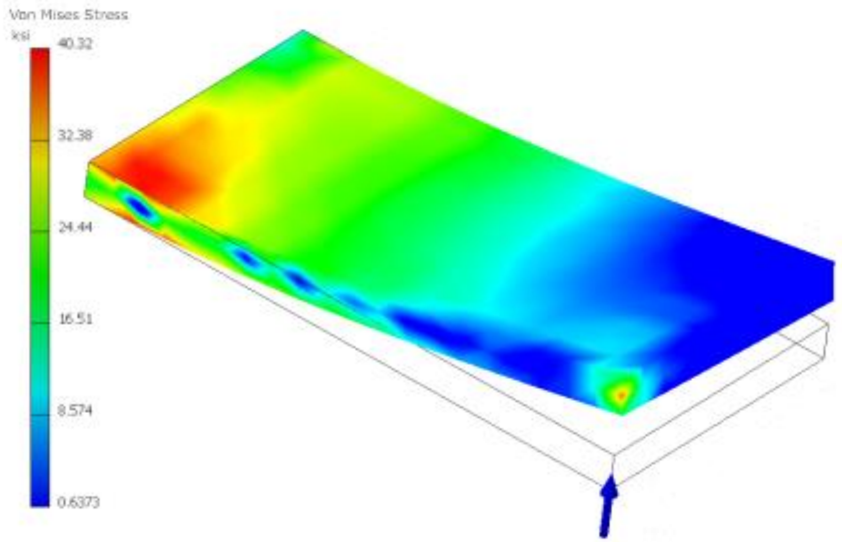


Simulation Solving

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 design decisions.



Reviewing Results

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

Analysis Types

- Linear ← Focus for this week
- Nonlinear
- Thermal
- Natural Frequency – Modal
- Fatigue Analysis
- Fluid Flow

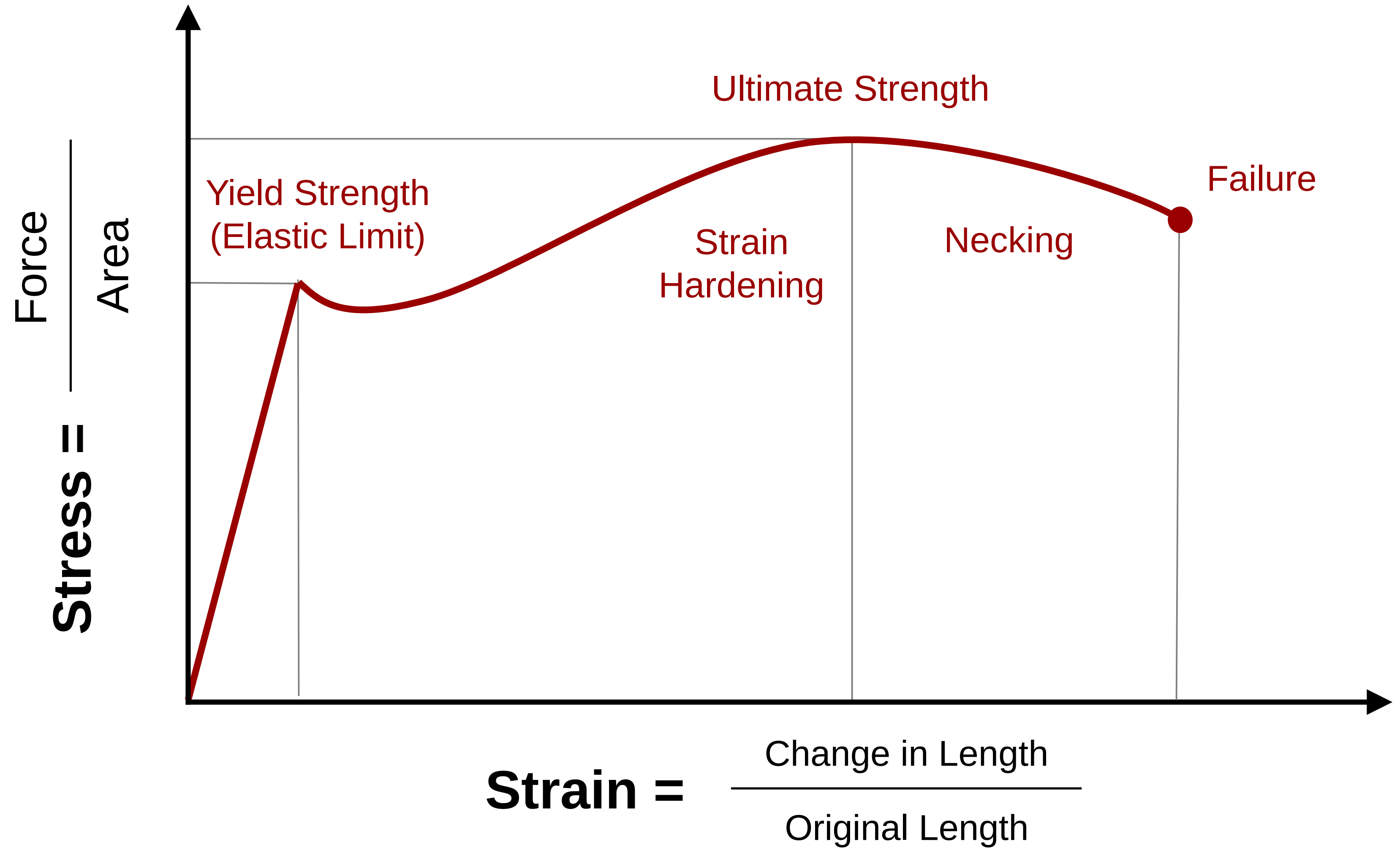
Linear vs. Nonlinear

- Linear ⟵ Focus for this week
  - 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

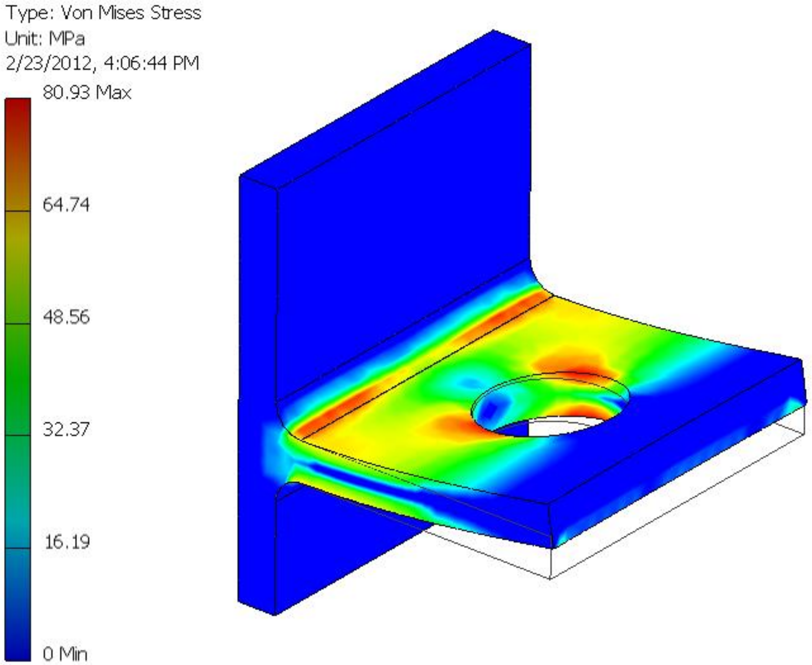
- Density = 0.284 lbmass/in^3
- Young’s Modulus = 3.193E+004 ksi
- Poisson’s Ratio = 0.275
- Yield Strength = 3.004E+004 psi
- Ultimate Tensile Strength = 5.007E+004 psi
- Thermal Conductivity = 1.259E+003 btu in/(ft^2 hr f)
- Linear Expansion = 21.600 Micoin/(in f)
- Specific Heat = 0.356 btu/(lbmass f)

Mild Steel Stress Strain Curve



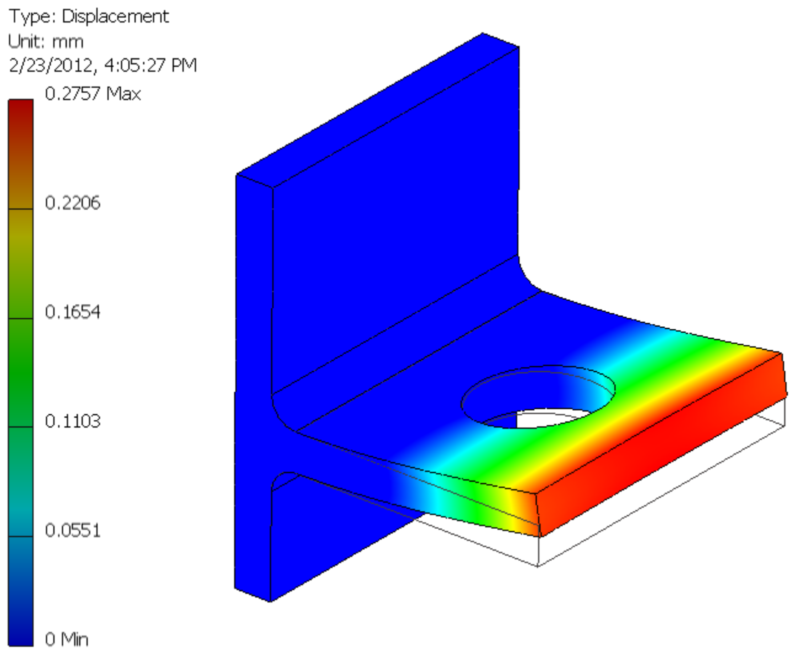
Von Mises Stress

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



Displacement

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



Safety Factor

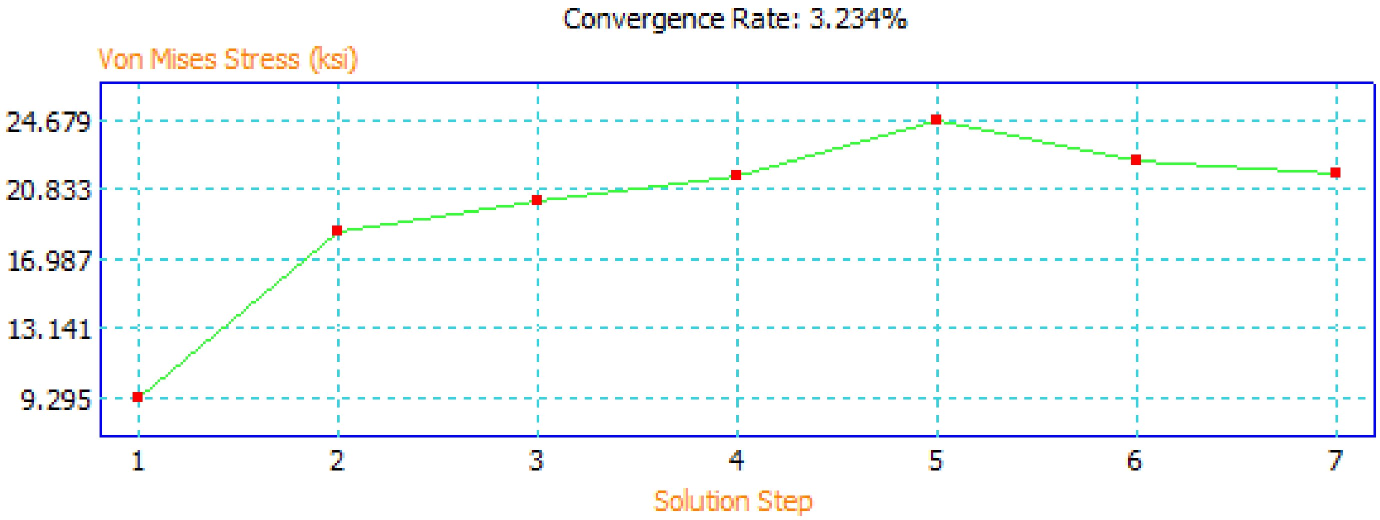
Provides a ratio of how much stronger the object is than it usually needs to be for an intended load.

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

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

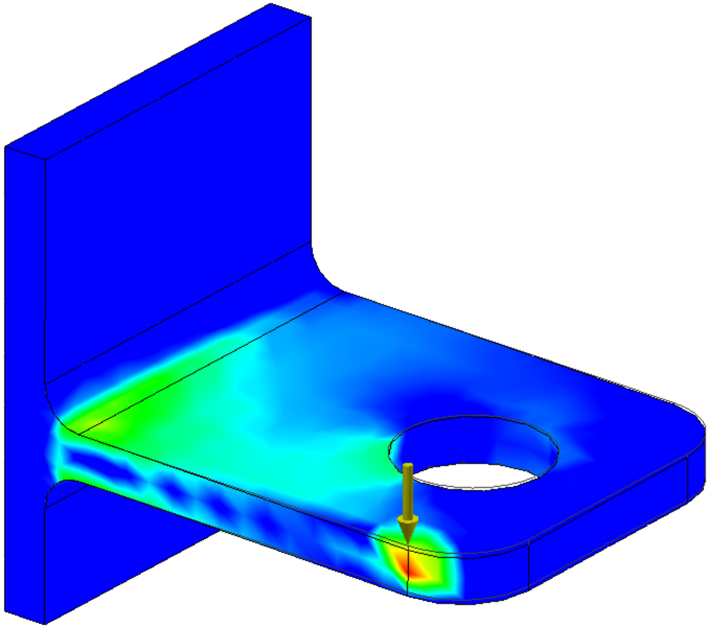
Convergence (Mesh Independence Study)

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 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
- Choose formulation type (Linear / Nonlinear)
- Identify stress singularities
- Ensure your results converge

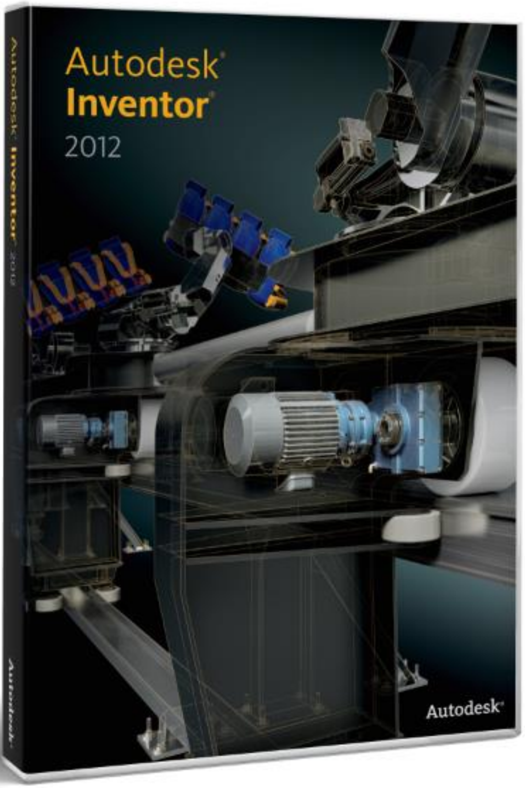
FEA Software

- FEA Features Built into Design Applications
  - General functionality for engineers to use upfront
  - Often limited to linear analysis with limited element types
  - General load and constraint options
  - Very affordable and easy to use
- Specialized Simulation Applications
  - Robust capabilities (Nonlinear, Fatigue, Metaphysics)
  - Focused more on dedicated analysis engineers' needs instead of design engineers' needs.
  - Advanced mesh creation, loads, constraints, etc.
  - More expensive and often harder to use *(This is changing)*

Autodesk Inventor Professional

FEA Capability Summary

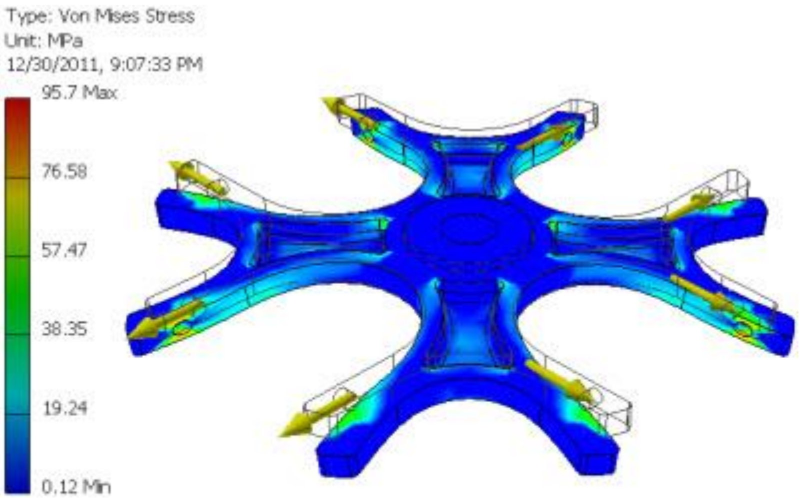
- Linear Analysis
- Tetrahedron Elements Only
- Static and Modal Analysis
- Automatic Mesh Creation
- Frame Analysis (Line Elements)
- General Loads, Constraints, Contacts



Computer-Cluster Projects (CP9)

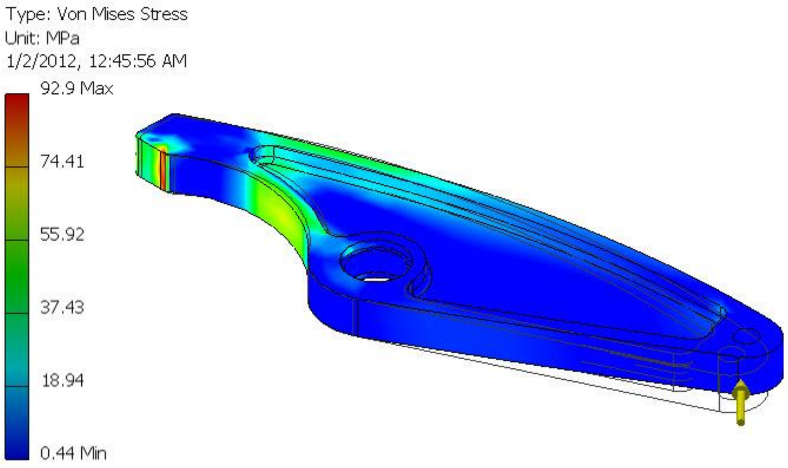
Guided Lab Project 1

Guided instructions for assigning loads and constraints.



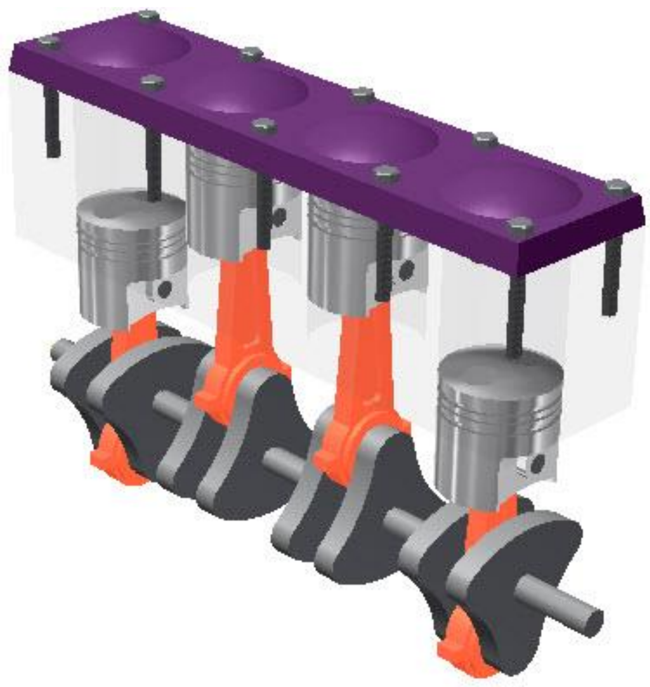
Guided Lab Project 2

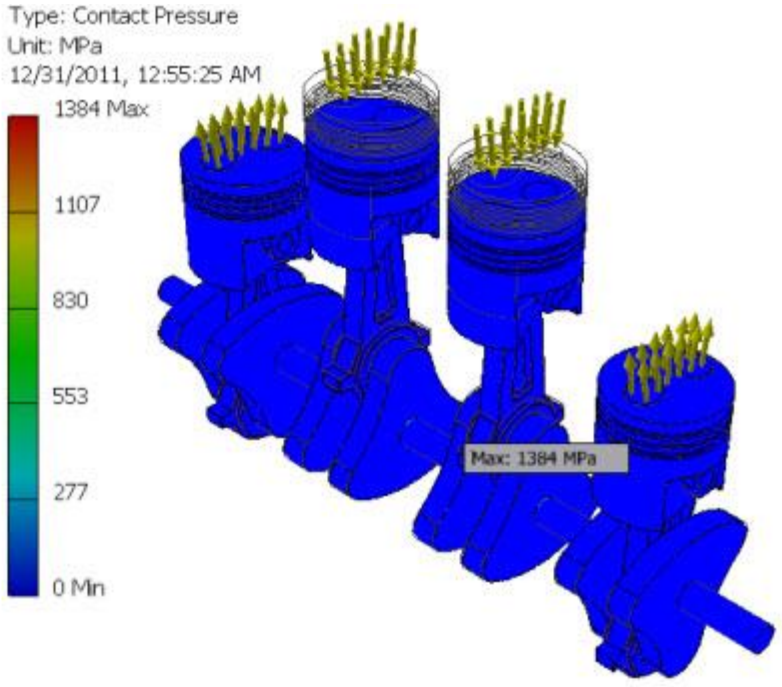
Guided instructions for performing an analysis on the clamp arm to optimize the design.



Guided Lab Project 3

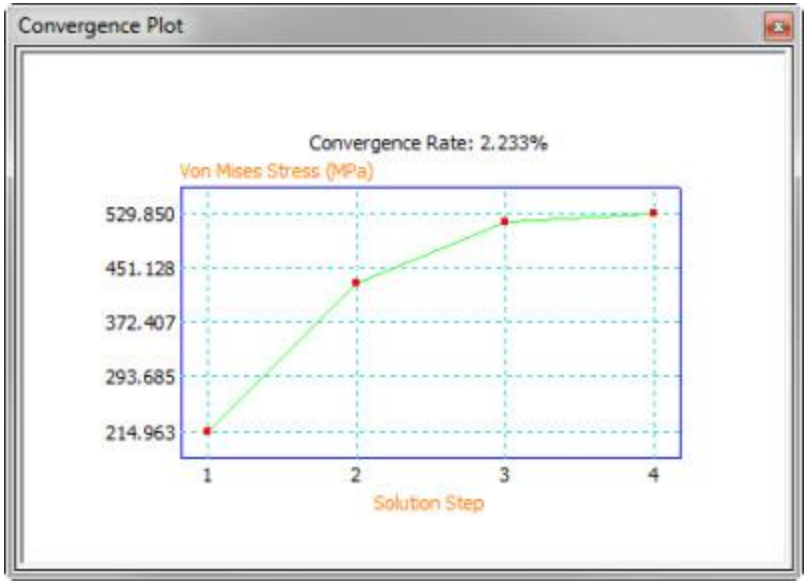
Guided instructions for performing an assembly analysis.





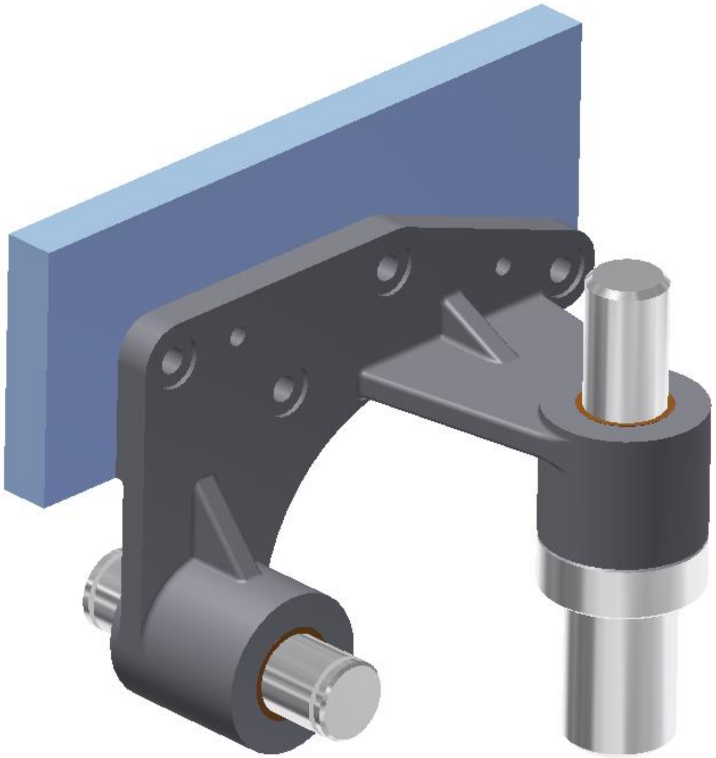
Guided Lab Project 4

Guided instructions for performing a design study and convergence.



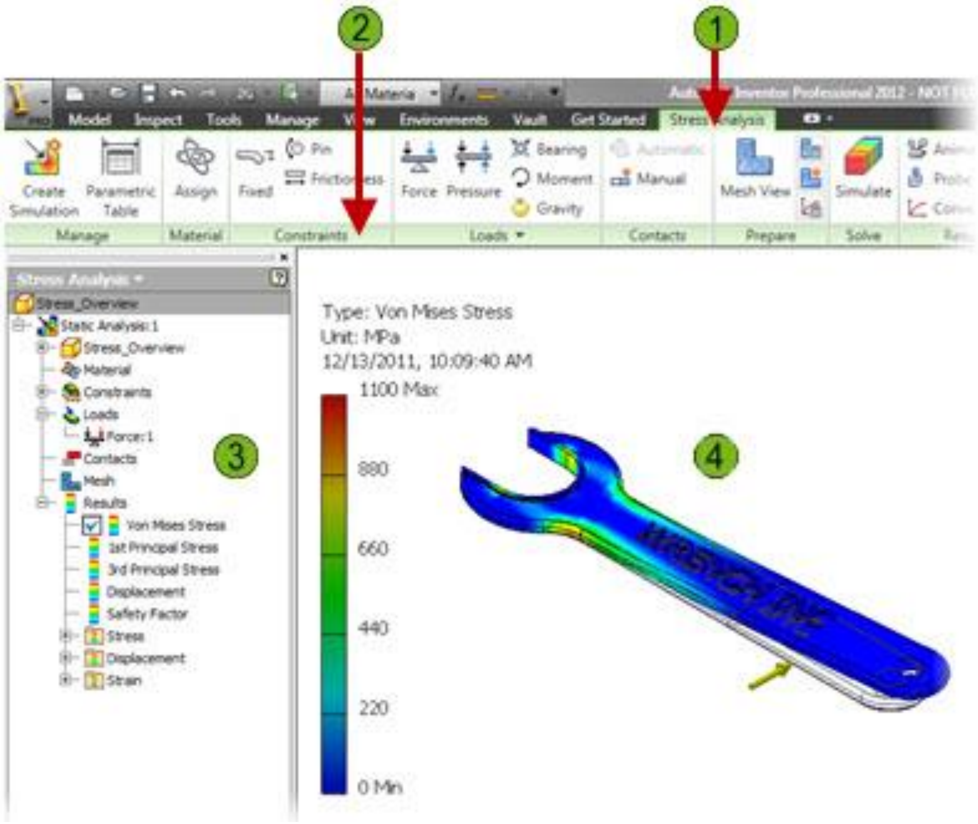
Problem Set Assignment

Analyze the bracket to ensure the optimal design is produced.



Demo Topics

User Interface



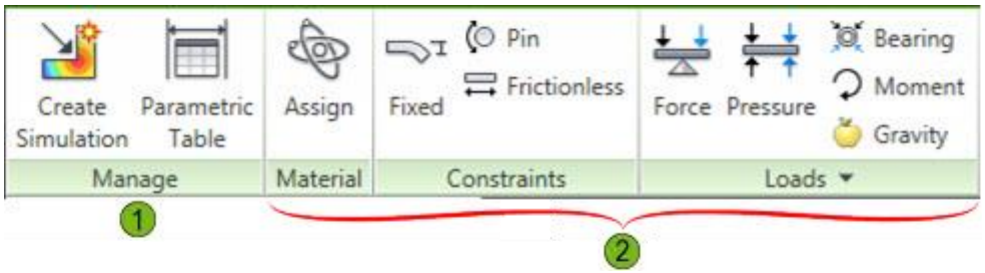
1 Stress Analysis tab

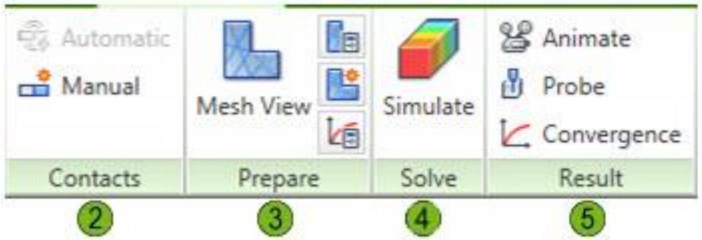
2 Stress Analysis panels

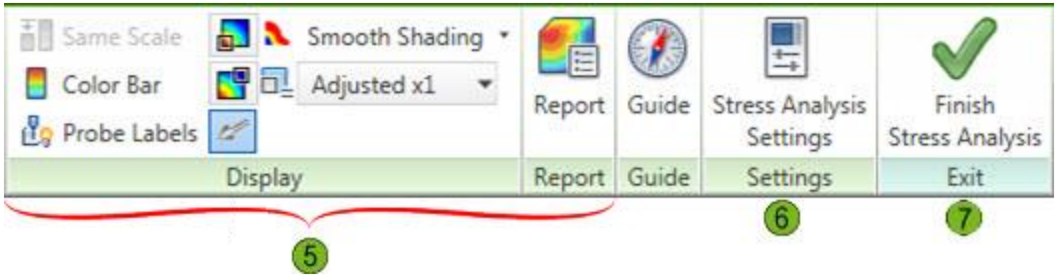
3 Stress Analysis browser

4 Graphical display

Stress Analysis Panels

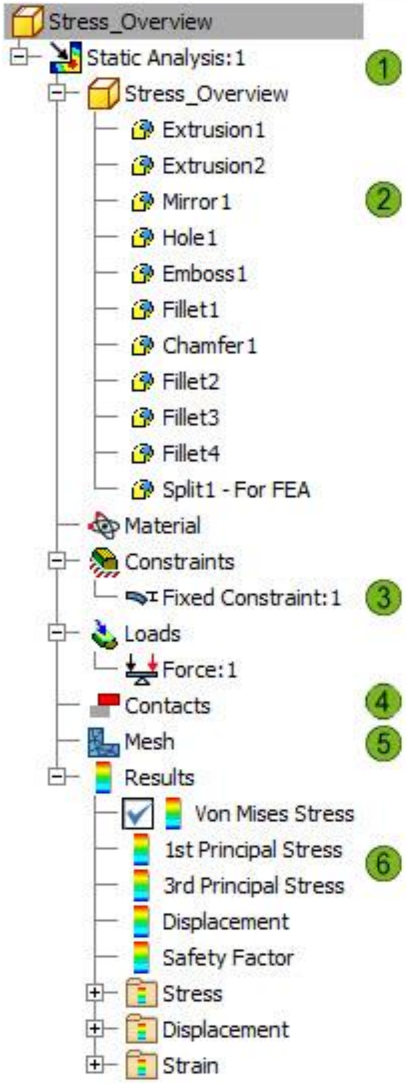






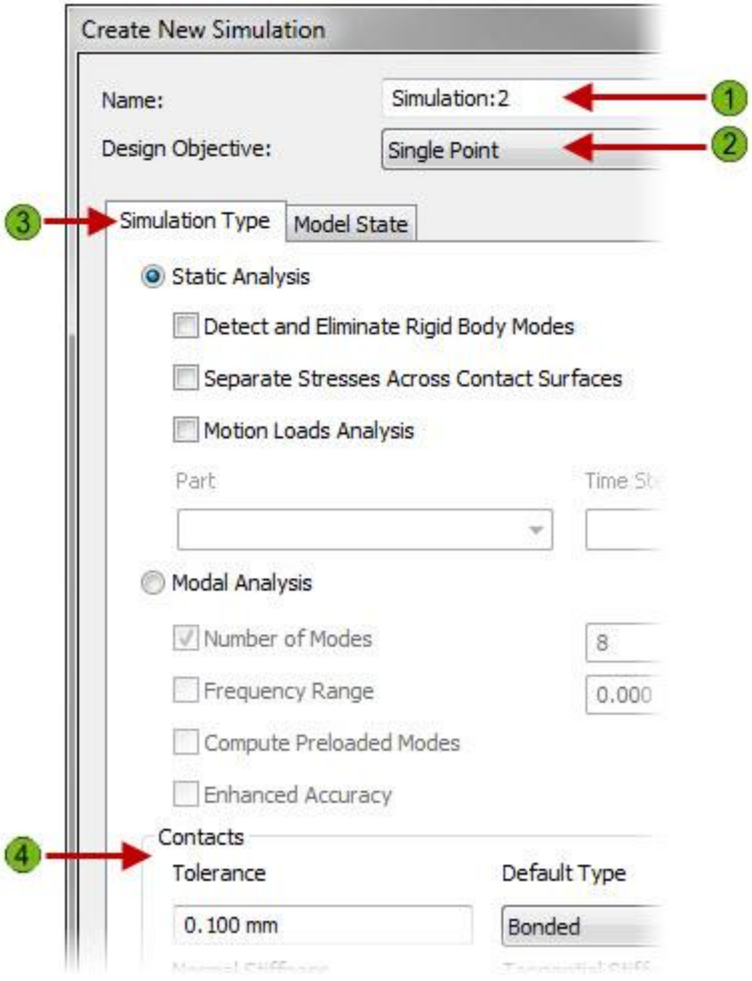
1. Manage panel
2. Materials, Constraints, Loads, and Contacts panels
3. Prepare panel
4. Solve panel
5. Result, Display, and Report panels
6. Settings panel
7. Exit panel

Stress Analysis Browser



1. Multiple simulations
2. For a part, features.
   For an assembly, parts
3. Constraints and Loads
4. Contacts
5. Mesh settings
6. Results folder

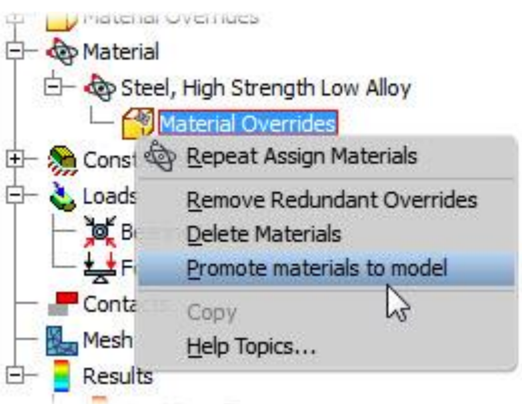
Simulation Properties



1. Name of the simulation
2. Single Point design objective
   Parametric Dimension design objective
3. Static Analysis or Modal Analysis
4. Defaults for the Automatic Contacts tool

Assign Materials

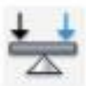
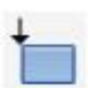
1. Component column
2. Original Material column
3. Override Material column
4. Safety Factor column



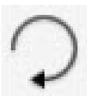
Assign Constraints

Icon	Constraint type	Can be applied to	Description
	**Fixed**	Vertex Face	A fixed constraint restricts the translation of the constrained geometry in one, two, or three directions. Use a fixed constraint to model rigid connection points to other components. Fix all three directions when you know that the part is fully fixed to a rigid support, such as where an edge or face of the part is welded or bonded to another part. Use components of the fixed constraint to fix or release motion in specific directions.
	**Pin**	Cylindrical Face	You use a pin constraint to prevent a cylindrical surface on the part from moving radially, tangentially, or axially. You typically use pin constraints where parts pivot on bearings or pins. You can select which directions to fix with respect to the cylindrical surface. For a bearing or pin, you free the tangential direction to enable the surface to rotate freely.
	**Frictionless**	Face	A frictionless constraint enables a surface to freely slide along a plane or surface but prevents the surface from moving normal to itself. You use frictionless constraints to model face-to-face and surface-to-surface contact between parts where one part can slide on the other. Most surfaces in contact are not entirely frictionless so frictionless constraints give conservative results because the friction's contribution to the overall model stiffness is not included.

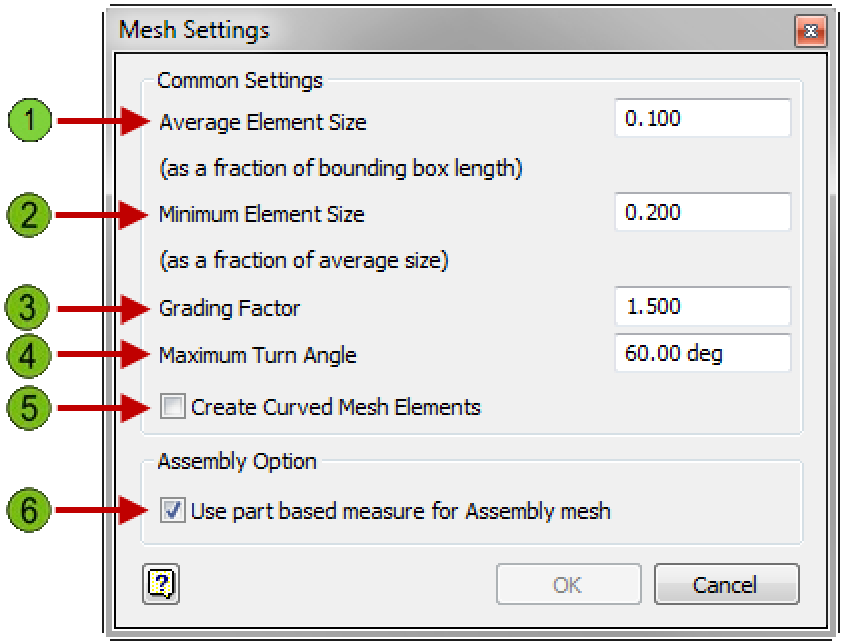
Assign Loads

Icon	Load type	Can be applied to	Description
	**Force**	Vertex Face	Applies a force of the specified magnitude to the selected faces, edges, or vertices. The force is distributed evenly over the selected geometry.
	**Pressure**	Face	Applies a pressure of the specified magnitude to the selected faces. Pressure loads are always normal to the face. Positive pressure points into the face; negative pressure points away from the face.
	**Bearing Load**	Face (cylindrical only)	Applies a load of the specified magnitude to an internal or external cylindrical face. Distributes the force over the projected area of the face. Typically used to define the load that a tight pin or shaft transfers to a hole in the part.
	**Remote Force**	Face	A remote force is similar to a regular force except that you specify a point through which the force acts. The point is typically not on the part. The resulting load on the part will be a force and a moment.

Assign Loads Cont'd

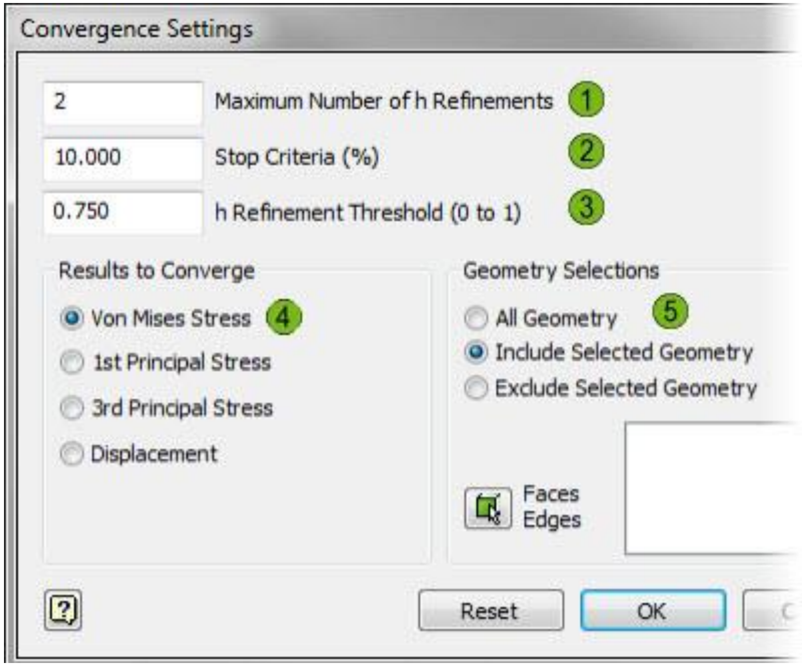
Icon	Load type	Can be applied to	Description
	**Moment**	Face	Applies a moment of the specified magnitude to the face.
	**Body Loads**	Acts on the whole model	Applies linear acceleration or angular velocity and acceleration of the specified magnitude to the entire model. Accounts for inertial forces and centripetal forces in bodies that accelerate and rotate.
	**Gravity**	Acts on the whole model	Applies gravity of the specified magnitude so that the weight of the model is included in the simulation.
	**Enforced Displacement**	Vertex Face	Specifies the displacement of a point, edge, or face using the Fixed Constraint tool. Use when you know the distance that you want the model to displace and you want to find the forces and stresses required. Enforced displacement loads are useful to determine how much force is required to close the gap between two parts or to deform a part a given distance.

Mesh Settings



1. Specifies the size of the elements in the initial mesh as a fraction of the largest overall dimension of the model. Recommended range is 0.05 to 0.1
2. Specifies the minimum size of elements as a fraction of the average element size. Recommended range 0.1 to 0.2
3. Specifies the maximum ratio of adjacent mesh edges for transitioning between coarse and fine regions. Recommended values from 1.5 to 3.
4. When an arc is meshed, the arc is broken into one or more elements according to the specified turn angle.
5. Creates meshes with curved edges and faces. If you clear this option, you produce meshes with straight elements, which can lead to a less accurate representation of the model.
6. If unchecked, the average element size is based on the overall size of the assembly, resulting in mesh elements that may be too large for small parts. (Only available in Assemblies)

Automatic Convergence Settings



1. Specifies the maximum number of refinement that takes place during convergence.
2. Specifies when the convergence stops.
3. Specifies the refinement threshold (between 0 to 1). A zero setting means include all the elements in the set as candidates for refinement. 1 means exclude all elements in the set from refinement. The default is .75, which means, of the elements with equivalent errors at the top, 25% are subject to refinements.
4. Specifies which analysis result to check for convergence.
5. A simulation will not converge if there is a stress singularity. If the singularity is not in an area of interest or importance, you typically ignore the stress in that area for the purpose of convergence.

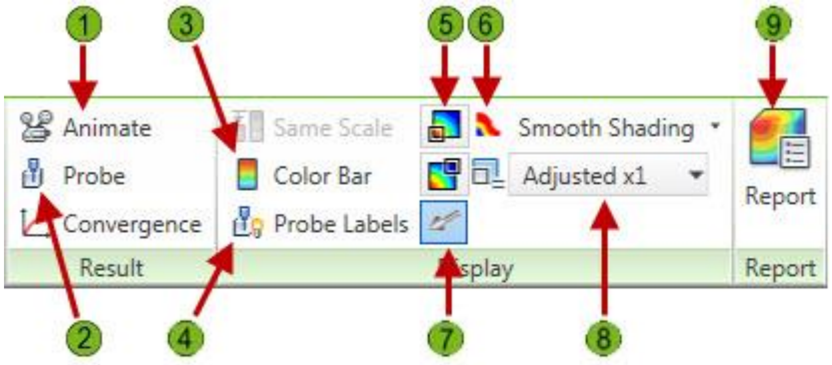
Contacts

Icon	Option	Description
	Bonded	Bonds contact faces rigidly. The two faces deform together and do not separate or penetrate. Typical uses include welded, bonded, and rigid bolted joints. Transfers all force directions between parts.
	Separation	Contact faces can separate and slide but cannot penetrate. Use where one face can push against another face but is not connected to the other face. Can be used to prevent penetration of components that are not initially in contact. Transfers just positive normal forces between parts.
	Sliding/No Separation	Contact faces can slide along each other but cannot separate or penetrate. Use where one face can push and pull against another face but can slide. Common examples include shafts in holes. The shaft cannot pull away from the hole but can rotate in the hole. Transfers positive and negative normal forces between faces.
	Separation/ No Sliding	Contact faces can separate but cannot slide. Use where one face rests against another face but can pull away from the other face. Can be used to simulate contact conditions between flexible parts that are bolted or welded. The bolts and welds prevent sliding but the faces away from the bolts or welds can separate but not penetrate.

Contacts Cont’d

Icon	Option	Description
	**Shrink Fit/ Sliding**	Like Separation, but where faces are initially overlapping. Contact faces can separate and slide but cannot penetrate further. Typically used to model the interface between parts that are pressed into or onto other parts where the faces may separate under loading.
	**Shrink Fit/No Sliding**	Like Separation/No Sliding, but where faces are initially overlapping. Contact faces can separate but cannot penetrate further. Typically used to model the interface between parts that are pressed into or onto other parts where the faces may separate under loading.
	**Spring**	Creates a flexible contact between faces using springs with user-specified normal and tangential stiffness. Typically used to model nonrigid connections between faces.

Results Tools



1. Animates the displacement
2. Probe the results at a Particular Node
3. Adjust the color bar position and scale
4. Controls the visibility of Probe Labels
5. Displays the maximum and minimum Labels
6. Selects the type of color shading
7. Controls the visibility of Boundary Conditions
8. Controls the model displacement scale
9. Generates a report