INTRODUCTION TO SIMULATION

SOLIDWORKS Simulation

Contents

Overview

The Introduction to Simulation guide is intended for novice users of the finite element analysis method. It covers basic engineering concepts and techniques with the goal of providing to design analysts the knowledge needed to incorporate this powerful simulation tool successfully into the design process.

Fundamentals of FEA

The Finite Element Method (FEM) is a numerical technique used for providing approximate solutions to a set of partial differential equations that describe physical phenomena.

Definition of Finite Element Analysis

In mathematical terms, Finite Element Analysis (FEA), also known as the Finite Element Method (FEM), is a numerical technique for describing physical phenomena in terms of partial differential equations. Finite element analysis is widely used in many engineering disciplines for solving structural mechanics, vibration, heat transfer, and other problems.

You use the FEM to predict the behavior of mechanical and thermal systems under their operating conditions, to reduce the design cycle time, and to improve overall system performance.

The basic steps in any FEA process are as follows:

Geometric representation	Creates the geometric features of the system to be analyzed stored in a CAD database.
Discretization of geometry	Splits the geometry into relatively small and simple geometric entities, called finite elements. This discretization process is better known as mesh formulation. The elements are called "finite" to emphasize the fact that they are not infinitesimally small, but only reasonably small in comparison to the overall model size.
Element formulation	Develops the equations that describe the behavior of each element. Material properties for each element are considered in the formulation of the governing equations. This involves choosing a displacement function within each element. Linear and quadratic polynomials are frequently used functions.
Assembly	Obtains the set of global equations for the entire model from the equations of individual elements. The loads and support (boundary) conditions are applied to the appropriate nodes of the finite element mesh.
Solution of equations	Provides the solution for the unknown nodal degrees of freedom (or generalized displacements).
Postprocessing	Obtains visualization plots for quantities of interest, such as stresses and strains.

🖉 **Note:**

FEA is not the only tool available for numerical analysis. Other numerical methods used in engineering include the Finite Difference Method, Boundary Element Method, or Finite Volumes Method. However, due to its versatility and high numerical efficiency, FEA has come to dominate the software market for engineering analysis, while other methods have been relegated to niche applications. Using FEA, you can analyze models of complex shapes under their operating load environment and predict their behavior within the desired accuracy.

Components of a FEA Model

A FEA model is composed of several different components that together describe the physical problem to be analyzed and the results to be obtained. At a minimum the analysis model consists of the following information: discretized geometry, element properties, material data, loads and boundary conditions, analysis type, and output requests.

The following topics are discussed:

Discretized geometry

Finite elements and nodes define the basic geometry of the physical structure being modeled.

Each element in the model represents a discrete portion of the physical structure, which is, in turn, represented by many interconnected elements. Elements are connected to one another by shared nodes. The coordinates of the nodes and the connectivity of the elements—that is, which nodes belong to which elements—comprise the model geometry. The collection of all the elements and nodes in a model is called the mesh. Generally, the mesh is only an approximation of the actual geometry of the structure.

The element type and the overall number of elements used in the mesh affect the results obtained from a simulation. The greater the mesh density (that is, the greater the number of elements in the mesh), the more accurate the results. As the mesh density increases, the analysis results converge to a unique solution, and the computer time required for the analysis increases. The solution obtained from the numerical model is generally an approximation to the solution of the physical problem being simulated. The extent of the approximations made in the model's geometry, material behavior, boundary conditions, and loading determines how well the numerical simulation matches the physical problem.
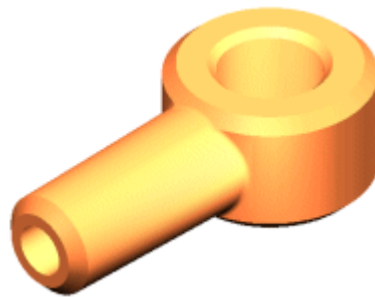

Figure 1: CAD model of a part
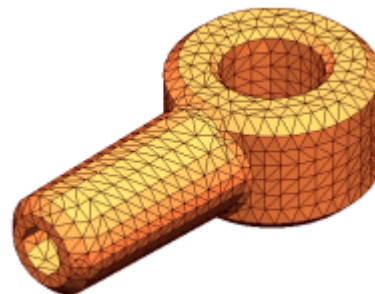
Figure 2: Model subdivided into small pieces (elements)

Element properties

The choice of the appropriate element is very important for the simulation of the physical problem. Element libraries include elements that are simple geometric shapes with one, two, or three dimensions.

Continuum or solid elements are appropriate for bulky or complex 3D models. Shell elements are suitable for thin parts with thickness significantly smaller than the other dimensions. Beam elements are suitable for structural members where the length is significantly greater than the other two dimensions.

Continuum   Shell   Beam
(solid) elements   elements   elements
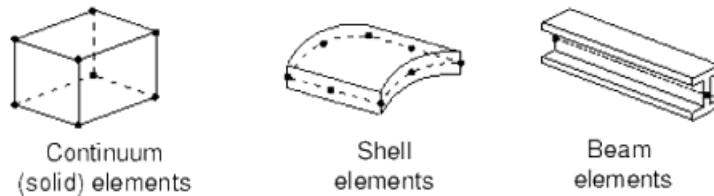
Figure 3: Common element families

Material data

Material properties for all elements must be specified. While high-quality material data are often difficult to obtain, particularly for the more complex material models, the validity of the simulation results is limited by the availability of accurate material data.

Loads and boundary conditions

The application of loads distorts the physical structure and, thus, creates stress in it. Boundary conditions are used to constrain portions of the model to remain fixed (zero displacements) or to move by a prescribed amount (nonzero displacements).

The most common forms of loading include:

- Point loads
- Pressure loads on surfaces
- Distributed tractions on surfaces
- Distributed edge loads and moments on shell edges
- Body forces, such as the force of gravity
- Thermal loads

In a static stress analysis adequate boundary conditions must be used to prevent the model from moving as a rigid body in any direction; otherwise, unrestrained rigid body motion causes the simulation to stop prematurely. The potential rigid body motions depend on the dimensionality of the model.

Analysis type

Depending on the type of applied load environment, inclusion of inertia effects, and material properties, the appropriate analysis type must be selected for the simulation. Common analysis types include linear static, nonlinear static, dynamic, buckling, heat transfer, fatigue, and optimization.

In a static analysis the long-term response of the structure to the applied loads (which are applied gradually and slowly until they reach their full magnitude) is obtained. In cases where the loads are changing with time or frequency, a dynamic analysis is required. For example, you perform a dynamic analysis to simulate the effect of an impact load on a component or the response of a building during an earthquake.

A nonlinear structural problem is one in which the structure's stiffness changes as it deforms. All physical structures exhibit nonlinear behavior. Linear analysis is a convenient approximation that is often adequate for

design purposes. It is obviously inadequate for many structural simulations including manufacturing processes, such as forging or stamping; crash analyses; and analyses of rubber components, such as tires or engine mounts.

Simple Rod Example: Obtaining Nodal Displacements Using Implicit Methods

A simple example of a rod, constrained at one end and loaded at the other end is used to introduce the basic concepts of FEA.

The objective of the analysis is to find the displacement of the free end of the rod, the stress in the rod, and the reaction force at the constrained end of the rod.

The first step of any finite element simulation is to discretize the actual geometry of the structure using a collection of finite elements. Each finite element represents a discrete portion of the physical structure. The finite elements are joined by shared nodes. The collection of nodes and finite elements is called the mesh. The number of elements per unit of length, area, or volume is referred to as the mesh density. In a stress analysis, the displacements of the nodes are the fundamental variables for which a solution is required. Once the nodal displacements are known, the stresses and strains in each finite element can be determined easily.

Consider a simple example of a rod, constrained at one end and loaded at the other end. The rod is modeled with two truss elements, which can carry axial loads only.
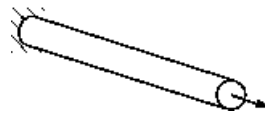
Figure 4: Axially-loaded rod

The discretized model with the node and element labels and the free-body diagrams for each node in the model are shown below.
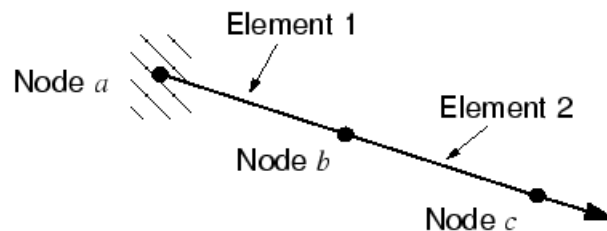
Figure 5: Discretized model of rod
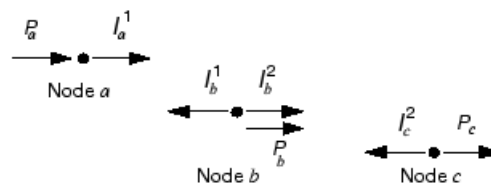
Figure 6: Free-body diagram for each node

In general each node will carry an external load applied to the model, P, and internal loads, l, caused by stresses in the elements attached to that node. For a model to be in static equilibrium, the net force acting on each node must be zero; that is, the internal and external loads at each node must balance each other. For node a this equilibrium equation can be obtained as follows.

Assuming that the change in length of the rod is small, the strain in element 1 is given by

$$\varepsilon_{11} = \frac{u^b - u^a}{L},$$

where u^a and u^b are the displacements at nodes a and b, respectively, and L is the original length of the element.

Assuming that the material is elastic, the stress in the rod is given by the strain multiplied by the Young's modulus E.

$$\sigma_{11} = E_{11}\varepsilon_{11}$$

The axial force acting on the end node is equivalent to the stress in the rod multiplied by its cross-sectional area, A. Thus, a relationship between internal force, material properties, and displacements is obtained.

$$I_a^1 = \sigma_{11}A = E\varepsilon_{11}A = \frac{EA}{L}\left(u^b - u^a\right).$$

Equilibrium at node b must take into account the internal forces acting from both elements joined at that node. The internal force from element 1 is now acting in the opposite direction and so becomes negative. The resulting equation is

$$P_b - \frac{EA}{L}\left(u^b - u^a\right) + \frac{EA}{L}\left(u^c - u^b\right) = 0.$$

For node c the equilibrium equation is

$$P_c - \frac{EA}{L}\left(u^c - u^b\right) = 0.$$

For implicit methods, the equilibrium equations need to be solved simultaneously to obtain the displacements of all the nodes. This requirement is best achieved by matrix techniques; therefore, we write the internal and external force contributions as matrices. If the properties and dimensions of the two elements are the same, the equilibrium equations can be simplified as

$$\begin{Bmatrix} P_a \\ P_b \\ P_c \end{Bmatrix} - \left(\frac{EA}{L}\right)\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{Bmatrix} u^a \\ u^b \\ u^c \end{Bmatrix} = 0.$$

In general, it may be that the element stiffnesses, the EA/L terms, are different from element to element; therefore, write the element stiffnesses as K_1 and K_2 for the two elements in the model. We are interested in obtaining the solution to the equilibrium equation in which the externally applied forces, P, are in equilibrium with the internally generated forces, I. When discussing this equation with reference to convergence and nonlinearity, we write it as

$$\{P\} - \{I\} = 0.$$

For the complete two-element, three-node structure we, therefore, modify the signs and rewrite the equilibrium equation as

$$\begin{Bmatrix} P_a \\ P_b \\ P_c \end{Bmatrix} - \begin{bmatrix} K_1 & -K_1 & 0 \\ -K_1 & (K_1 + K_2) & -K_2 \\ 0 & -K_2 & K_2 \end{bmatrix}\begin{Bmatrix} u^a \\ u^b \\ u^c \end{Bmatrix} = 0.$$

In an implicit method, this system of equations can then be solved to obtain values for the three unknown variables: u^b, u^c, and P_a (u^a is specified in the problem as 0.0).

Once the displacements are known, we can go back and use them to calculate the stresses in the truss elements. Implicit finite element methods require that a system of equations is solved at the end of each solution increment.

Note: In contrast to implicit methods, an explicit method, does not require the solving of a simultaneous system of equations or the calculation of a global stiffness matrix. Instead, the solution is advanced kinematically from one increment to the next.

Finite Elements and Rigid Bodies

Finite elements and rigid bodies are the fundamental components of an FEA model. Finite elements are deformable, whereas rigid bodies move through space without changing shape.

Definition of Finite Elements

Finite elements are the fundamental components of an FEA model. They are basic geometric shapes that discretize the model geometry and are represented mathematically by element equations. The element equations are simple equations that locally approximate the original complex equations of the problem to be analyzed.

Each finite element is characterized by the following:

Family (related to spatial dimensionality)

Depending on the geometry features of the model, you assign elements from a suitable element family to create the mesh.

Commonly used element libraries for a stress/displacement simulation include: continuum (or solid), shell, beam, truss, rigid elements, etc.

Continuum (solid) elements   Shell elements   Beam elements   Rigid elements   Truss elements
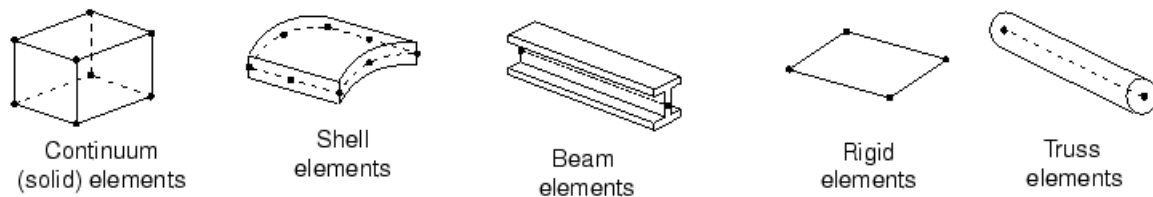
Figure 7: Common element families

Degrees of freedom (directly related to the element family)

The degrees of freedom (dof) are the fundamental variables calculated during the analysis.

For a stress/displacement simulation the degrees of freedom are the translations at each node. Some element families, such as the beam and shell families, have rotational degrees of freedom as well. For a heat transfer simulation the degrees of freedom are the temperatures at each node.

🖉 **Note:** A heat transfer analysis, therefore, requires the use of different elements than a stress analysis, since the degrees of freedom are not the same.

Depending on the type of analysis, the calculated degrees of freedom are:

- Three translations in the X-, Y-, and Z-directions (based on a global or local coordinate system)
- Three rotations about the X-, Y-, and Z-axes (based on a global or local coordinate system)
- Temperature

Number of Nodes - order of interpolation

Displacements, rotations, temperatures, and any other degrees of freedom mentioned in the previous section are calculated only at the nodes of the element.

At any other point in the element, the displacements are obtained by interpolating from the nodal displacements. Usually the interpolation order is determined by the number of nodes used in the element.

Linear or First-Order elements Elements that have nodes only at their corners use linear interpolation.

Quadratic or Second-Order elements Elements with mid-side nodes use quadratic interpolation

4-node tetrahedron
linear element

10-node tetrahedron
quadratic element
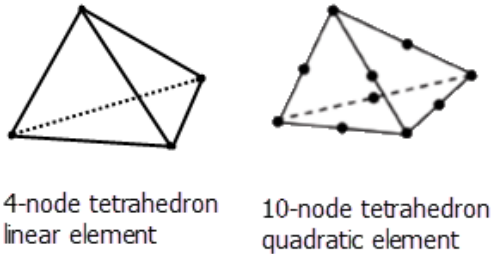
**Figure 8: Linear and quadratic tetrahedral elements**

Integration

The term integration refers to numerical techniques, like Gaussian quadrature, to integrate the polynomial terms in an element's stiffness matrix, over the volume of each element.

Using Gaussian quadrature for most elements, an FEA solver evaluates the material response at each integration point in each element. Some elements can use full or reduced integration, a choice that can have a significant effect on the accuracy of the element for a given problem.

Linear elements

Quadratic elements
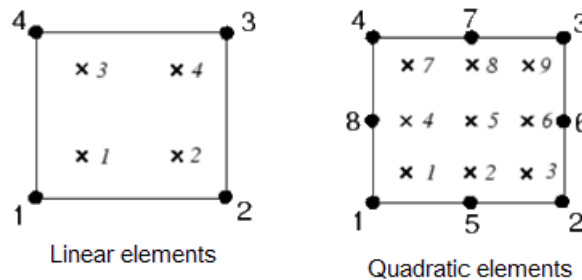
Figure 9: Integration points in fully integrated, two-dimensional, quadrilateral elements

Linear elements

Quadratic elements
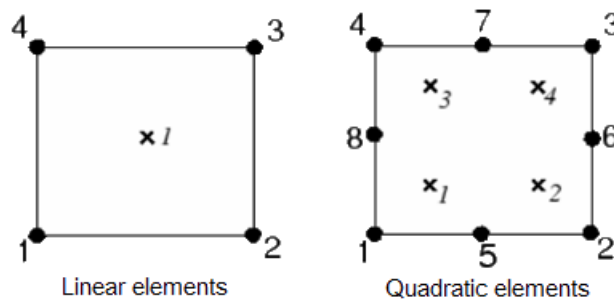
Figure 10: Integration points in reduced integrated, two-dimensional, quadrilateral elements

The expression "full integration" refers to the number of Gauss points required to integrate the polynomial terms in an element's stiffness matrix when the element has a regular shape. For hexahedral and quadrilateral elements

a "regular shape" means that the edges are straight and meet at right angles and that any edge nodes are at the midpoint of the edge.

Reduced-integration elements use one less integration point in each direction than fully integrated elements. Reduced-integration, linear elements have just a single integration point located at the element's centroid.

Quadratic reduced-integration elements are generally the best choice for most general stress/displacement simulations, except in large-displacement simulations involving very large strains and in some types of contact analyses.
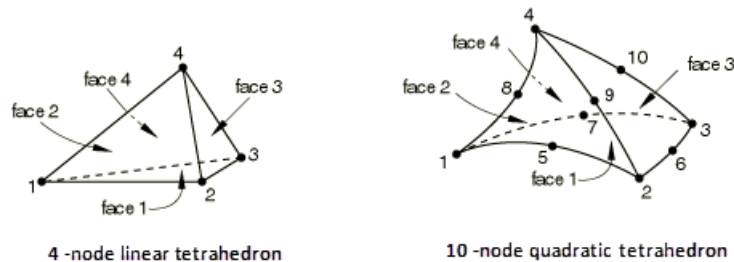
Continuum Elements

Continuum or solid elements can be used to model the widest variety of components. Conceptually, continuum elements simply model small blocks of material in a component.

Since they may be connected to other elements on any of their faces, continuum elements, like bricks in a building or tiles in a mosaic, can be used to build models of nearly any shape, subjected to nearly any loading. There are stress/displacement, nonstructural, and coupled field continuum elements; this guide will discuss only stress/displacement elements. The following topics are discussed:

Three-dimensional continuum element library

Three-dimensional continuum elements are tetrahedra.

4 -node linear tetrahedron   10 -node quadratic tetrahedron

Figure 11: Three-dimensional continuum elements

A library of solid elements also includes two-dimensional elements. The most commonly used two-dimensional continuum elements are as follows:

- Plane strain elements suitable for modeling thick structures (out-of-plane strain is zero); for example, a cross-section of a dam under water pressure.
- Plane stress elements suitable for modeling thin structures (out-of-plane stress is zero); for example, thin slabs with one dimension smaller than the other two.
- Axisymmetric elements suitable for analyzing structures with axisymmetric geometry subjected to axisymmetric loading; for example, an O-ring pressed into a groove by a top plate.
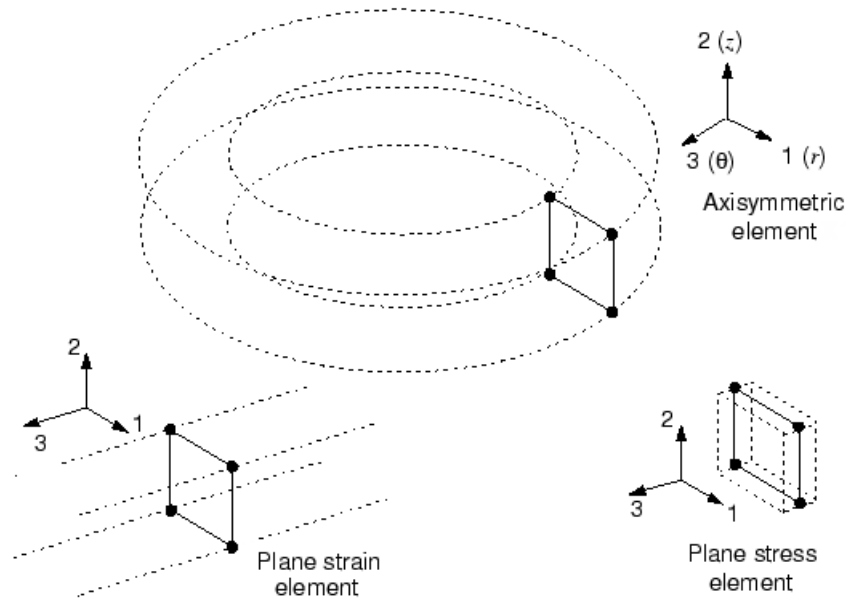
Figure 12: Plane strain, plane stress, and axisymmetric elements

Selecting Continuum Elements

The correct choice of element for a particular simulation is vital if accurate results are to be obtained at a reasonable cost.

If you are a new user with little experience of FEA applications, the task of selecting the most suitable element is usually executed by the software. The most commonly used element for meshing arbitrary geometries is the quadratic tetrahedron. As you become more experienced with FEA applications, you can follow these recommendations on element selection:

- Minimize the mesh distortion as much as possible. Coarse meshes with distorted linear elements can give very poor results.
- Use a fine mesh of linear, reduced-integration elements for simulations involving very large mesh distortions (large-strain analysis).
- Use quadratic elements for general analysis work, unless you need to model very large strains or have a simulation with complex, changing contact conditions.
- For contact problems use a fine mesh of linear elements.

Shell Elements

Use shell elements to model structures in which one dimension (the thickness) is significantly smaller than the other dimensions, and in which the stresses in the thickness direction are negligible.

Shell elements are 2D elements capable of resisting membrane and bending loads. A structure, such as a pressure vessel, whose thickness is less than 1/10 of a typical global structural dimension generally can be modeled with shell elements. The following are examples of typical global dimensions:

- Distance between supports
- Distance between stiffeners or large changes in section thickness
- Radius of curvature, and
- Wavelength of the highest vibration mode of interest.

The following topic is discussed:

Shell formulation - Thick or Thin

Shell problems generally fall into one of two categories: thick shell problems and thin shell problems.

Thick shell problems assume that the effects of transverse shear deformation are important to the solution. Thin shell problems, on the other hand, assume that the transverse shear deformation is small enough to be neglected. Material lines that are initially normal to the shell surface remain straight and normal throughout the deformation.

Hence, transverse shear strains are assumed to vanish ($\gamma = 0$).
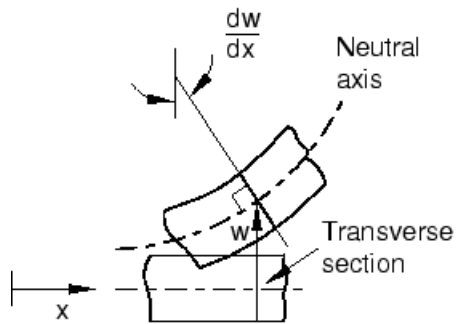
Figure 13: Transverse shear behavior of thin shells

Material lines that are initially normal to the shell surface do not necessarily remain normal to the surface throughout the deformation, thus adding transverse shear flexibility ($\gamma \neq 0$).
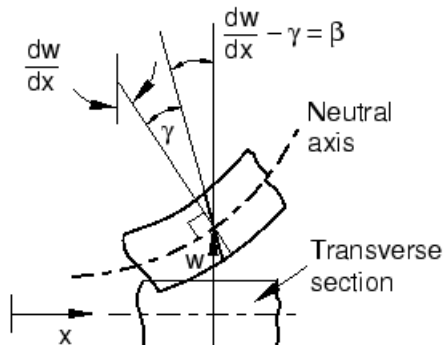
Figure 14: Transverse shear behavior of thick shells

To decide if a given application is a thin or thick shell problem, we can offer a few guidelines. For thick shells, transverse shear flexibility is important; while for thin shells, it is negligible. The significance of transverse shear in a shell can be estimated by its thickness-to-span ratio. A shell made of a single isotropic material with a thickness-to-span ratio greater than 1/15 is considered "thick"; if the ratio is less than 1/15, the shell is considered "thin."

🖉 **Note:** For structural analysis problems, each node in a shell element has the following degrees of freedom: three translations and three rotations. The translational degrees of freedom are motions in the global *X*-, *Y*-, and *Z*-axes. The rotational degrees of freedom are the rotations about the global *X*-, *Y*-, and *Z*-axes.
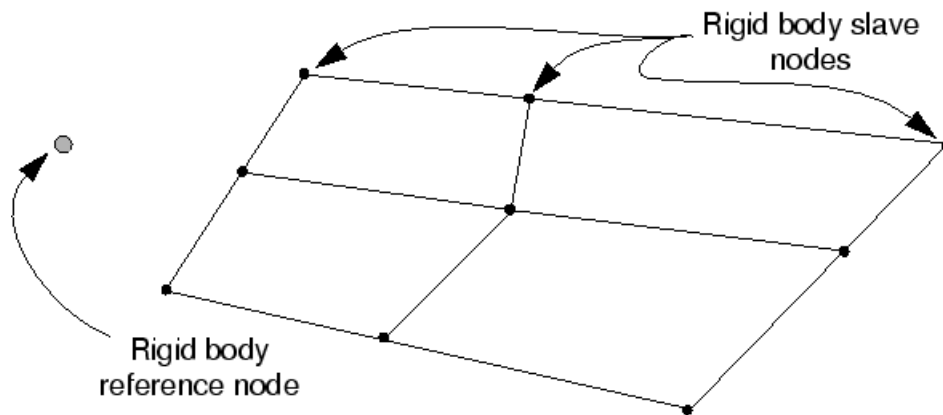
Rigid Bodies

A rigid body is a collection of nodes and elements whose motion is governed by the motion of a single node, known as the rigid body reference node. Rigid bodies can be used to model very stiff components that are either fixed or undergoing large rigid body motions.

Rigid bodies are ideally suited for modeling tooling materials (such as a punch, die, drawbead, blank holder, roller, etc.) and stiff indenters pressing against soft samples.

The main advantage of representing portions of a model with rigid bodies is computational efficiency. Element-level calculations are not performed for elements that are part of a rigid body. Although some computational effort is required to update the motion of the nodes of the rigid body and to assemble concentrated and distributed loads, the motion of the rigid body is determined completely by a maximum of six degrees of freedom (three translations and three rotations about the $X-$, $Y-$, and $Z-$directions) at the rigid body reference node.

It may be useful to make parts of a model rigid for verification purposes. For example, in complex models elements far away from the particular region of interest could be included as part of a rigid body, resulting in faster run times at the model development stage. When you are satisfied with the model, you can remove the rigid body definitions and incorporate an accurate deformable finite element representation throughout.

Only the rigid body reference node has independent degrees of freedom and must be defined for every rigid body.

The nodes attached to rigid elements have only slave degrees of freedom. The motion of these nodes is determined entirely by the motion of the rigid body reference node.

🖉 **Note:** The position of the rigid body reference node is not important unless rotations are applied to the body or reaction moments about a certain axis through the body are desired. In either of these situations the node should be placed such that it lies on the desired axis through the body.

Loads on a rigid body are generated from concentrated loads applied to nodes and distributed loads applied to elements that are part of the rigid body or from loads applied to the rigid body reference node. Rigid bodies interact with the rest of the model through nodal connections to deformable elements and through contact with deformable elements.

Mesh Formulation

A mesh is an assembly of small geometric shapes called elements. The process of dividing the geometry of a model into elements is called meshing. A mesh can contain anywhere from thousands to millions of elements.

Mesh Convergence

It is important that you use a sufficiently refined mesh to ensure that the results from your simulation are accurate. The numerical solution for your FEA model will tend toward a unique value as you increase the mesh density. The computer resources required to run your simulation also increase as the mesh is refined. The mesh is said to be converged when further mesh refinement produces a negligible change in the solution.
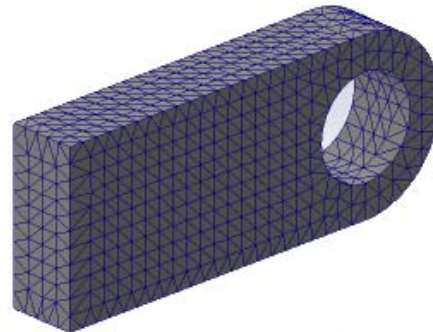
As you gain experience, you will learn to judge what level of refinement produces a suitable mesh to give acceptable results for most simulations. However, it is always good practice to perform a mesh convergence study, where you simulate the same problem with a finer mesh and compare the results. You can have confidence that your model is producing a mathematically accurate solution if the two meshes give essentially the same result.

Mesh Convergence for a Connecting Lug
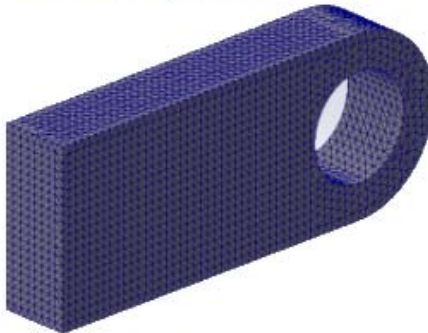
We use the geometry of a connecting lug to demonstrate a mesh refinement study using four different mesh densities. The number of mesh elements used in each case is indicated in the figure.
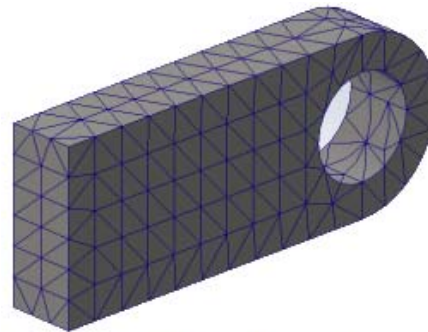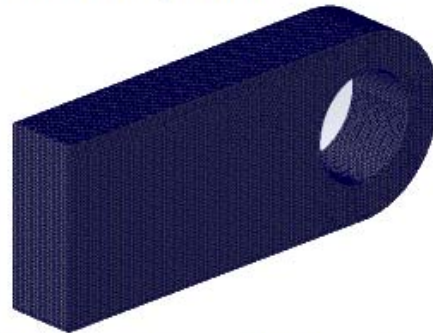

Coarse mesh (1,055 elements)   Normal mesh (7,164 elements)

Fine mesh (50,604 elements)   Very Fine mesh (550,436 elements)

Figure 15: Different mesh refinement for the connecting lug model

We consider the influence of the mesh density on three particular results from this model:

- The displacement of the bottom of the hole.
- The peak von Mises stress at the stress concentration on the bottom surface of the hole.
- The peak von Mises stress where the lug is attached to the parent structure.

The locations where the results are compared are shown below:
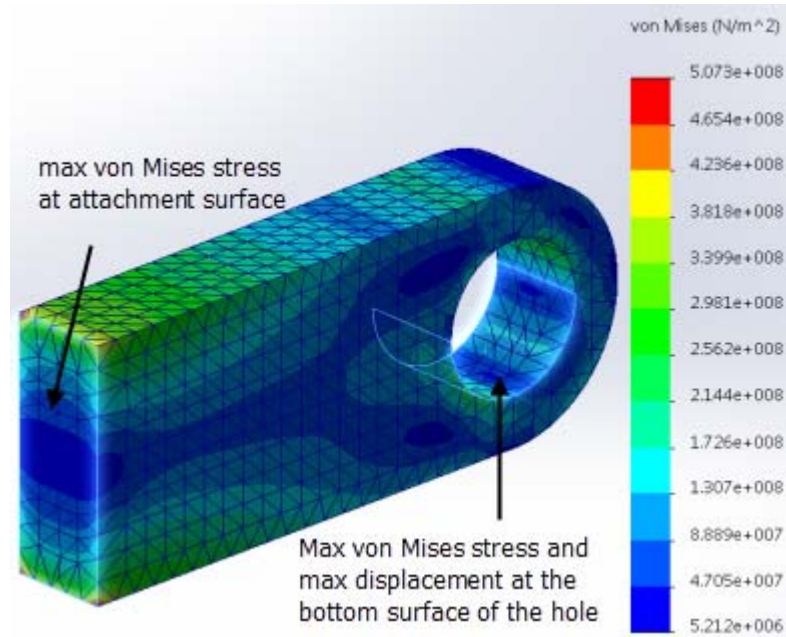
Figure 16: Locations where results are compared in the mesh refinement study

The results for each of the four mesh densities are compared in the table below.

Table 1: Results of mesh refinement study

Mesh	Max displacement at bottom inner surface of hole (mm)	Max Von Mises stress at bottom inner surface of hole (N/m^2)	Max stress at attachment (N/m^2)
Coarse	0.392	3.261 E8	3.926 E8
Normal	0.396	3.402 E8	5.073 E8
Fine	0.397	3.468 E8	6.019 E8
Very Fine	0.397	3.47 E8	8.12 E8

The coarse mesh predicts less accurate displacements at the bottom inner surface of hole, but the normal, fine, and very fine meshes all predict similar results. The normal mesh is, therefore, converged as far as the displacements are concerned.

The peak stress at the bottom inner surface of the hole converges much more slowly than the displacements because stress and strain are calculated from the displacement gradients; thus, a much finer mesh is required to predict accurate displacement gradients than is needed to calculate accurate displacements.

Mesh refinement significantly changes the stress calculated at the attachment of the connecting lug; it continues to increase with continued mesh refinement. A stress singularity exists at the corner of the lug where it attaches to the parent structure. Theoretically the stress is infinite at this location; therefore, increasing the mesh density will not produce a converged stress value at this location.

> **Note:** This singularity occurs because of the idealizations used in the finite element model. The connection between the lug and the parent structure has been modeled as a sharp corner, and the parent structure has been modeled as rigid. These idealizations lead to the stress singularity. In reality there probably will be a small fillet between the lug and the parent structure, and the parent structure will be deformable, not rigid. If the exact stress in this location is required, the fillet between the components must be modeled accurately and the stiffness of the parent structure must also be considered.
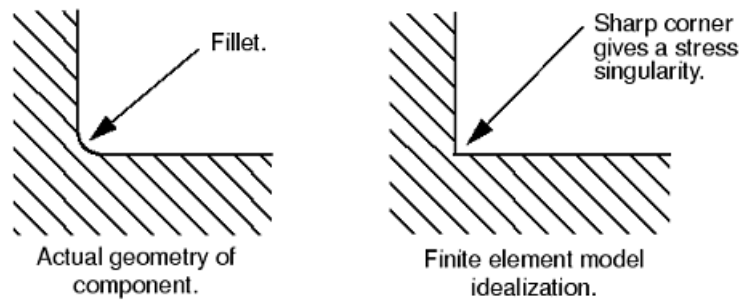
Figure 17: Idealizing a fillet as a sharp corner

It is common to omit small details like fillet radii from a finite element model to simplify the analysis and to keep the model size reasonable. However, the introduction of any sharp corner into a model will lead to a stress singularity at that location. This normally has a negligible effect on the overall response of the model, but the predicted stresses close to the singularity will be inaccurate.

Coarse meshes are often adequate to predict trends and to compare how different modeling concepts behave relative to each other. However, you should use the actual magnitudes of displacement and stress calculated with a coarse mesh with caution.

Mesh Control

It is rarely necessary to use a uniformly fine mesh throughout the structure being analyzed. You can use a finer mesh mainly in the areas of high stress gradients and use a coarser mesh in areas of low stress gradients or where the magnitude of the stresses is not of interest.

Mesh control refers to specifying different element sizes at different regions in the model. A smaller element size in a region improves the accuracy of results in the region of interest.

For example, the figure below shows a mesh that is designed to give an accurate prediction of the stress concentration at the bottom of the hole.
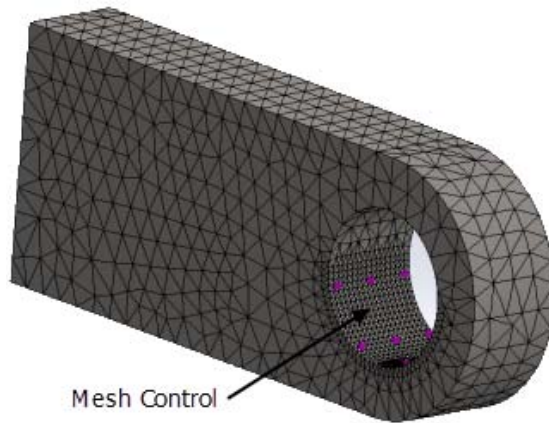
Mesh Control

Figure 18: Mesh refined around the hole

The results from the simulation with this locally refined mesh show that they are comparable to those from the very fine mesh.

Table 2: Mesh refinement (control) around the hole

Mesh	Max displacement at bottom inner surface of hole (mm)	Max Von Mises stress at bottom inner surface of hole (N/m^2)
Very fine	0.397	3.47 E8
Locally refined	0.397	3.468 E8

You can often predict the locations of the highly stressed regions of a model (and, hence, the regions where a fine mesh is required) using your knowledge of similar components or with hand calculations. This information can also be gained by setting a coarse mesh initially to identify the regions of high stress and then refining the mesh in these regions. It is simple to mesh the geometry coarsely for the initial simulation and then to refine the mesh in the appropriate regions, as indicated by the stress results from the coarse simulation.

Mesh Quality Checks

A good quality mesh is important for obtaining accurate results from your simulations. There are different quality check criteria that measure the quality of the mesh. This guide covers the following criteria:
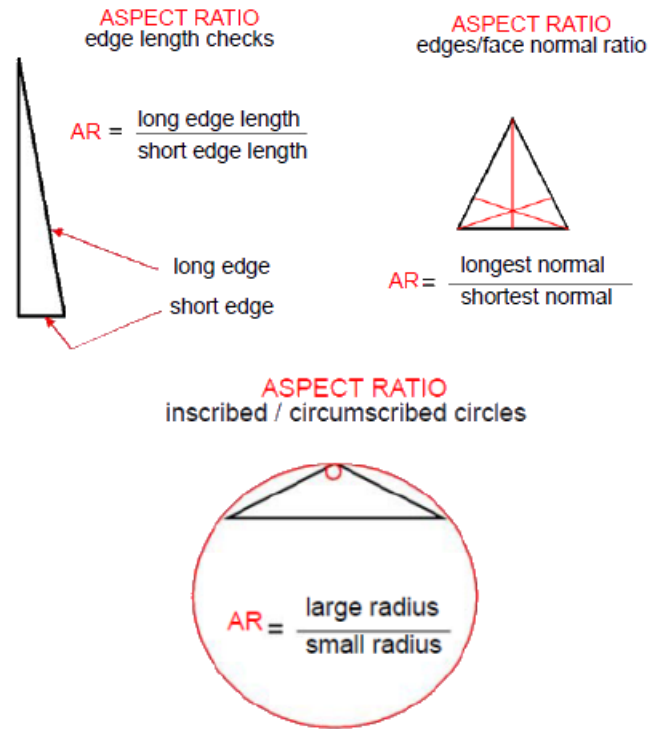
Aspect Ratio Checks

Numerical accuracy is best achieved by a mesh with uniform size elements whose edges are equal in length. For most CAD geometries, due to the existence of small edges, curvature changes, thin features, and sharp corners, it is not possible to create a mesh of perfect elements with equal lengths.

The aspect ratio of a perfect, tetrahedral element is used as the basis for calculating aspect ratios of other elements. By definition, the aspect ratio of a perfect tetrahedral element is 1.0. Finite elements provide more accurate results when their aspect ratio is close to 1.0. When the lengths of an element edges differ significantly, the accuracy of the results deteriorates.

There are different ways to measure the aspect ratio of an element:

- The ratio of the longest to the shortest element edge.
- The ratio of the longest normal to the shortest normal dropped from a vertex to the opposite face.
- The ratio of the largest circumscribed circle to the smallest inscribed circle.

Figure 19: Aspect Ratio Checks

Jacobian Check

Second-order elements map to curved geometry more accurately than linear elements of the same size. The mid-side nodes of the boundary edges of an element are placed on the actual geometry of the model. At the location of sharp or curved boundaries, placing the mid-side nodes on the actual geometry can result in generating distorted elements with overlapping edges.

The Jacobian ratio at a point inside the element provides a measure of the degree of distortion of the element at that location. Parabolic, tetrahedral elements with all mid-side nodes located exactly at the middle of the edges have a Jacobian ratio of 1.0. The Jacobian ratio increases as the curvatures of the element edges increase. A highly distorted element has a negative Jacobian ratio. It is a good practice to avoid these elements with "concave" faces by applying mesh controls or by adjusting the global element size.

Materials

A material definition in a FEA simulation specifies all the relevant material properties and the proper material behavior (such as elasticity, metal plasticity, hyperelasticity, thermal conductivity, etc.) that best describe the physical behavior of the model.

Material Classification

Materials can be broadly categorized into two types: isotropic and anisotropic. Most simulations are performed under the assumption of isotropic and homogeneous material properties.

Isotropic materials have identical properties in any direction and are characterized by two independent material constants (Young's modulus and Poisson's ratio). Most metals (steel, aluminum) are isotropic materials.

Homogeneous materials have consistent properties throughout the volume. That means that any impurities, localized changes due to heat or other processing effects, and internal voids that reduce the inherent stiffness of the material are not accounted for. You must keep these approximations in mind when you assess your simulation results.

Anisotropic materials have different properties depending on the direction and require you to specify the material orientation. Materials such as wood and fiber-reinforced composites are very anisotropic; they are much stronger along the direction of grain (or fiber) than across it.

Orthotropic materials belong to a special subtype of anisotropic materials. Othrotropic materials exhibit different behavior in three orthogonal planes (three mutually perpendicular material directions). Nine independent elastic stiffness parameters (Young's modulus, Poisson's ratio, and shear modulus in three principal material directions) are needed to define an orthotropic material.

Wood is an example of an orthotropic material. The material properties of wood are described in the longitudinal, radial, and tangential directions. The longitudinal axis (1) is parallel to the grain (fiber) direction; the radial axis (2) is normal to the growth rings; and the tangential axis (3) is tangent to the growth rings.
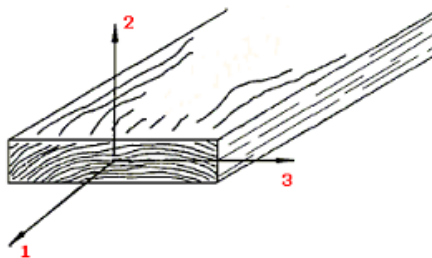
Figure 20: Wood as an orthotropic material.

The following common material properties apply to FEA applications:

Elastic Modulus or Young's Modulus

Measures the stiffness of a material and its ability to resist loads. For a linear elastic material the elastic modulus in a certain direction is defined as the stress value in that direction that causes a unit strain in the same direction (the slope of the stress-strain curve in the elastic region).

For example, a one inch bar of a material with Young's modulus of 35 million psi elongates by one thousandth of an inch under a tensile load of 35 thousand psi.

Shear Modulus or Modulus of Rigidity	Measures the ability of a material to resist shear loads. It is the ratio between the shearing stress in a plane divided by the associated shearing strain.
Poisson's Ratio	When a material extends in the longitudinal direction, it usually tends to contract in the lateral directions (similarly, when is compressed in one direction, it usually expands in the other two directions). If a body is subjected to a tensile stress in the *X*-direction, then the Poisson's ratio, v_{xy}, is defined as the ratio of lateral contraction in the *Y*-direction divided by the longitudinal strain in the *X*-direction.
	Most materials have Poisson's ratio values ranging between 0.0 and 0.5. Some materials (like polymer foams) have a negative Poisson's ratio; when these materials are stretched in one direction, they expand in the lateral directions.
Coefficient of Thermal Expansion	Measures the fractional change in size (length, area, or volume) per unit change in temperature at a constant pressure. Materials generally expand in all directions when subjected to a temperature increase.
	Depending on which dimensions are considered when measuring changes in size per unit change in temperature, there are definitions of volumetric, area, and linear coefficient of thermal expansion.
Thermal Conductivity	Measures the effectiveness of a material in transferring heat energy by conduction. It is defined as the rate of heat transfer through a unit thickness of the material per unit change in temperature.
Density	Mass per unit volume.
Specific Heat	Measures the quantity of heat required to raise the temperature of a unit mass of a material by one unit of temperature.
	The relationship between heat and temperature change is expressed in the form: Heat = Specific Heat * mass * (temperature change)

FEA applications typically offer material libraries that include most standard materials. The reported nominal material properties are accepted average values taken from many tests. When you accept a tabulated property value as a single number to be used in a simulation, keep in mind it actually has a probability distribution associated with it. For example the Young's modulus for steel has a standard deviation of approximately 7.5%. Most material tests yield results that follow a normal curve (or "bell shaped") distribution. If your part is made of a less predictable material, you need to consider the variability of the material properties in the total factor of safety of your design.

Material Models

Applying the proper material behavior that best describes the physical behavior of the model is crucial in the process of setting up a FEA simulation.

A wide collection of materials is encountered in stress analysis problems, and for any one of these materials a range of constitutive models is available to describe the material's behavior. For example, a component made from a standard structural steel can be modeled as an isotropic, linear elastic, material with no temperature dependence. This simple material model would probably suffice for routine design. However, if the component might be subjected to a severe overload, it is important to determine how it might deform under that load and if it has sufficient ductility to withstand the overload without catastrophic failure.

We can broadly classify the material behaviors of interest as follows:

- Materials that exhibit almost purely elastic response, possibly with some energy dissipation during rapid loading by viscoelastic response (such as rubber or solid propellant elastomers)
- Materials that yield and exhibit considerable ductility beyond yield (such as mild steel and other commonly used metals)
- Materials that flow by rearrangement of particles that interact generally through some dominantly frictional mechanism (such as sand)
- Materials that break without any significant deformation and absorb little energy prior to fracture (such as rocks, concrete, ceramics, glass)

Linear Elasticity

Linear elastic materials under this category follow Hooke's law; that is, stress is directly proportional to strain as the load increases or decreases.

The linear elastic model:

- can define isotropic, orthotropic, or anisotropic material behavior;
- is valid for small elastic strains (normally less than 5%);
- has properties that depend on temperature or other variables.

For example, in a linear elastic model, if stress reaches 100 MPa under a load of 1,000 N, then stress will reach 1,000 MPa under a load of 10,000 N. The simplest form of linear elasticity is the isotropic case, and the stress-strain relationship is given by:

$$
\begin{Bmatrix} \varepsilon_{11} \\ \varepsilon_{22} \\ \varepsilon_{33} \\ \gamma_{12} \\ \gamma_{13} \\ \gamma_{23} \end{Bmatrix} = \begin{bmatrix} 1/E & -v/E & -v/E & 0 & 0 & 0 \\ -v/E & 1/E & -v/E & 0 & 0 & 0 \\ -v/E & -v/E & 1/E & 0 & 0 & 0 \\ 0 & 0 & 0 & 1/G & 0 & 0 \\ 0 & 0 & 0 & 0 & 1/G & 0 \\ 0 & 0 & 0 & 0 & 0 & 1/G \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix}.
$$

The elastic properties are completely defined by the Young's modulus, E, and the Poisson's ratio, v. The shear modulus, G, can be expressed in terms of E and v as $G = E/2(1+v)$. These parameters can be given as functions of temperature and of other predefined fields, if necessary.

The material yielding is not modeled in a linear elastic model.

Plasticity

Elastic material models cease to be valid at the material points where the stresses exceed the yield stress (or else elastic limit). In such cases, the deformations are no longer fully recoverable, and permanent (inelastic) deformation begins to occur.

Plasticity theories model the material's mechanical response as it undergoes such non-recoverable deformation in a ductile fashion. The theories have been developed most intensively for metals, but they are also applied to soils, concrete, rock, ice, crushable foam, and so on. These materials behave in very different ways. For example, large values of pure hydrostatic pressure cause very little inelastic deformation in metals, but quite small hydrostatic pressure values may cause a significant, non-recoverable volume change in a soil sample.

The fundamental concepts of plasticity theories are sufficiently general that models based on these concepts have been developed successfully for a wide range of materials.

Yield Criterion

A yield criterion (or yield surface) specifies the shift from elastic to plastic behavior (onset of the plastic flow) in terms of a test function that determines if the material responds purely elastically at a particular state of stress. Several yield criteria have been developed to describe different material behaviors.

For example, the Tresca's yield criterion assumes that yielding occurs when the maximum shear stress (at a material point under a general state of stress) reaches the value of the maximum shear stress when yielding occurs in a uniaxial tension test.

The von Mises yield criterion assumes that yielding occurs when the distortion energy (at a material point under a general state of stress) is equal to the distortion energy at the onset of yielding in a uniaxial tension test. The von Mises yield criterion applies best to ductile materials, such as metals.

The Drucker-Prager yield criterion describes best the behavior of geological materials that exhibit pressure-dependent yield.

Hardening Rules

A hardening rule specifies how the loading and unloading of materials affect their yield strength during the plastic flow phase. Often the plastic deformation of the material increases its yield stress for subsequent loadings: this behavior is called work hardening.
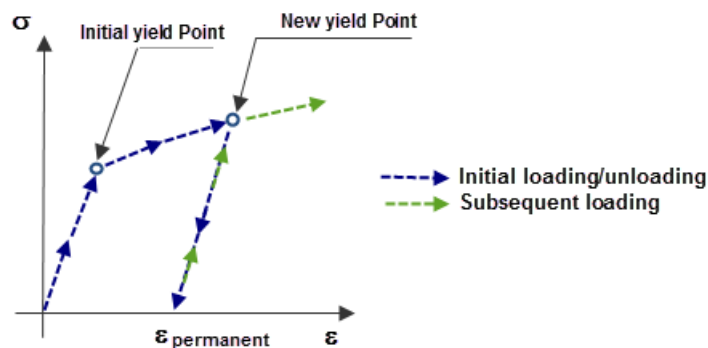
**Figure 21: Strain hardening for subsequent loading**

✏️ **Note:**

> In isotropic hardening, the yield surface changes size uniformly in all directions such that the yield stress increases (or decreases) in all stress directions as plastic straining occurs. In kinematic hardening, the yield surface remains constant in size and the surface translates in stress space with progressive yielding.

Characteristics of Metal Plasticity

The deformation of the metal prior to reaching the yield point creates only elastic strains, which are fully recovered if the applied load is removed. However, once the stress in the metal exceeds the yield stress, permanent (inelastic) deformation begins to occur. The strains associated with this permanent deformation are called plastic strains. Both elastic and plastic strains accumulate as the metal deforms in the post-yield region.

The stiffness of a metal typically decreases dramatically once the material yields. A ductile metal that has yielded will recover its initial elastic stiffness when the applied load is removed.
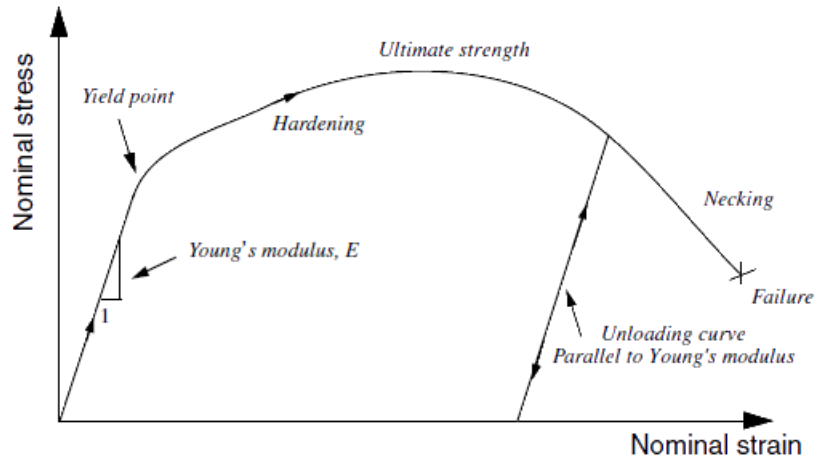
Figure 22: Nominal stress-strain behavior of an elastic-plastic material in a tensile test

A metal deforming plastically under a tensile load may experience highly localized extension and thinning, called necking, as the material fails. The engineering stress (force per unit undeformed area) in the metal is known as the nominal stress, with the conjugate nominal strain (length change per unit undeformed length).

The nominal stress in the metal as it is necking is much lower than the material's ultimate strength. This material behavior is caused by the geometry of the test specimen, the nature of the test itself, and the stress and strain measures used. For example, testing the same material in compression produces a stress-strain plot that does not have a necking region because the specimen is not going to thin as it deforms under compressive loads.

🖉 **Note:** A mathematical model describing the plastic behavior of metals should account for differences in the compressive and tensile behavior independent of the structure's geometry or the nature of the applied loads.

The post-yield material behavior is approximated in a FEA model with data points on a stress-strain curve that are gathered from material test data. The strains provided in material test data used to define the plastic behavior are not likely to be the plastic strains in the material but rather the total strains in the material. You must decompose these total strain values into the elastic ϵ^{el} and plastic strain components ϵ^{pl}. The plastic strain is obtained by subtracting the elastic strain, defined as the value of true stress divided by the Young's modulus, from the value of total strain.
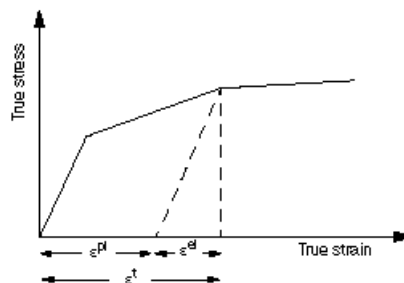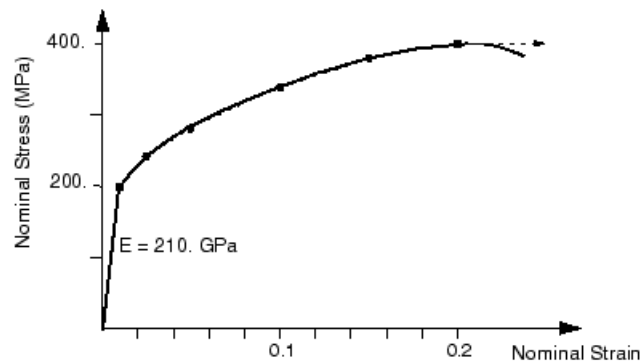

Figure 23: Decomposition of the total strain into elastic and plastic components

Depending on the required input format for the stress/strain curve data, you may need to convert the nominal stress and nominal strain to true stress and true strain. Although, the differences between the nominal and true values at small strains are small, there are significant differences at larger strain values; therefore, it is extremely important to provide the proper stress-strain data if the strains in the simulation will be large.

The figure below shows a nominal stress-strain curve defined by six data points.

Figure 24: Elasto-plastic material behavior

Depending on the required input format for the stress/strain curve data, you may need to convert the nominal stress and nominal strain to true stress and true strain. Although, the differences between the nominal and true values at small strains are small, there are significant differences at larger strain values; therefore, it is extremely important to provide the proper stress-strain data if the strains in the simulation will be large.

✏️ **Note:**

The relationship between true strain ϵ and nominal strain ϵ_{nom} is:
$$\epsilon = \ln(1 + \epsilon_{nom})$$

The relationship between true stress σ and nominal stress σ_{nom} and strain ϵ_{nom} is:
$$\sigma = \sigma_{nom}(1 + \epsilon_{nom})$$

True Stress/Strain vs Engineering Stress/Strain

Engineering strain is a small strain measure that is invalid once the strain in your model is no longer small (approximately 5%). True strain, which is a nonlinear strain measure that is dependent upon the final length of the model, is used for large strain simulations.

True Stress and Strain

If the deformation of a bar in tension becomes significant, its cross-sectional area will change. The traditional engineering definitions for stress and strain are no longer accurate, and new measures, namely true stress and true strain, are introduced. Alternative names for these quantities are Cauchy stress, logarithmic strain, and natural strain.
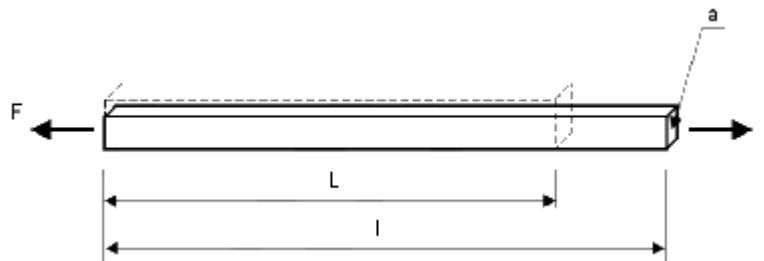
The true stress is $\sigma_T = \dfrac{F}{a}$, where a is the final deformed cross-sectional area.

The true strain is $\epsilon_T = \ln\left(\dfrac{l}{L}\right)$, where l is the final length and L is the initial undeformed length of the bar.

Engineering Stress and Strain
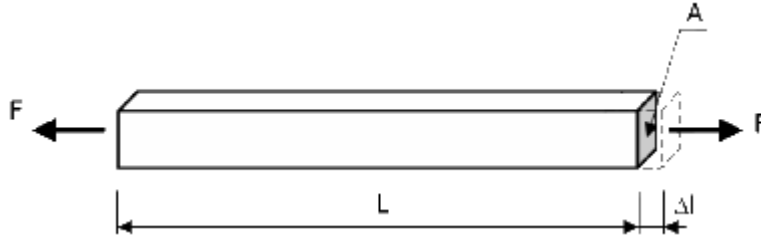
The engineering stress (or nominal stress) is $\sigma_E = \dfrac{F}{A}$, where A is the initial undeformed cross-sectional area.

The engineering strain (or nominal strain) is $\epsilon_E = \ln\left(\dfrac{\Delta l}{L}\right)$, where Δl is the final bar deformation.

Stresses are related to the strains through constitutive laws, which are governed by material properties. The simplest of the stress/strain laws is Hooke's law, which is given by $\sigma = E\epsilon$.

Hyperelasticity

Hyperelastic materials are characterized by their capacity to take large amounts of strain with relatively small stress (rubbers, for example). The name hyperelastic describes the ability to deform significantly, even under a small load.

The deformation of hyperelastic materials, such as rubber, remains elastic up to large strain values (often well over 100%). The stress-strain behavior of typical rubber materials is elastic but highly nonlinear and it depends on the loading mode. Different stress-strain behaviors are observed for elastic materials in uniaxial tension, biaxial tension, or pure shear types of loading.
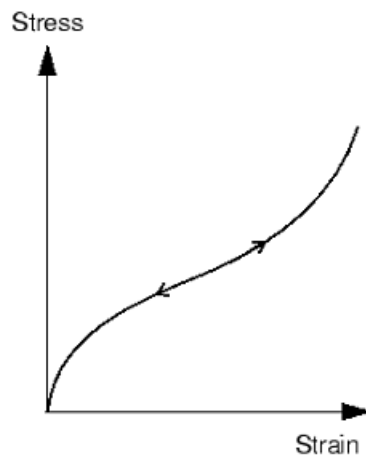

**Figure 25: Typical stress-strain curve for rubber**

Because of the complex behavior that hyperelastic materials exhibit, their elastic constants are derived from a strain energy potential function, which defines the strain energy stored in the material per unit of reference volume (volume in the initial configuration).

There are several forms of strain energy potentials available to model approximately incompressible isotropic elastomers:

- Polynomial models (Mooney-Rivlin and Blatz-Ko models)
- Ogden model

To define the material parameters for a hyperelastic material model, you usually provide experimental test data from:

- Uniaxial tension and compression
- Equibiaxial tension and compression
- Planar tension and compression (pure shear)
- Volumetric tension and compression (needed if the material's compressibility is important)
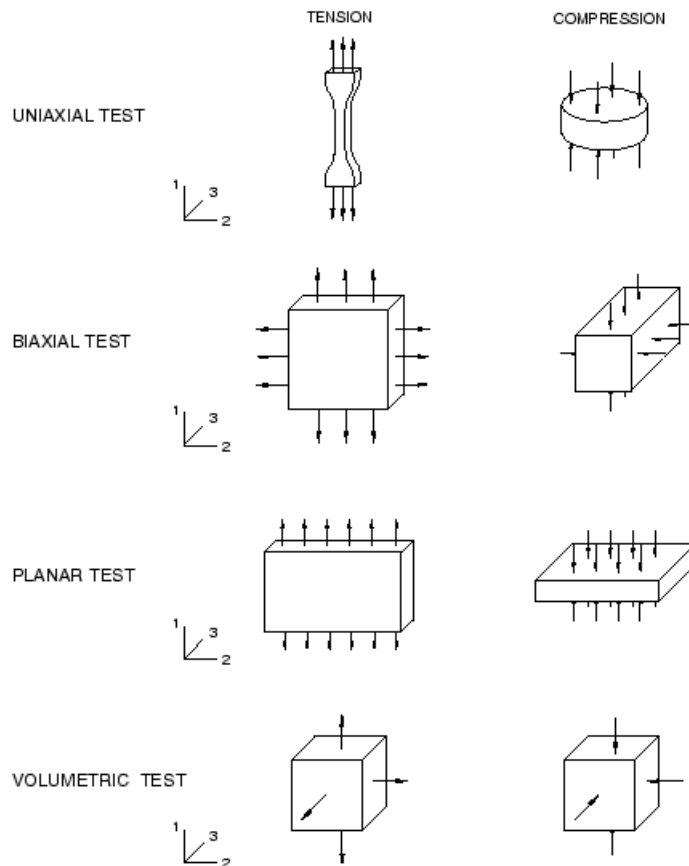
Figure 26: Experimental tests for defining hyperelastic material behavior

Recommendations for Using Hyperelastic Materials

The quality of the results from a simulation using hyperelastic materials strongly depends on the quality of the material test data that you provide.

To improve your hyperelastic material model, follow these recommendations:

- Obtain test data for the deformation modes that are likely to occur in your simulation. For example, if your component is loaded in compression, make sure that your test data include compressive, rather than tensile loading.

- Both tension and compression data are allowed, with compressive stresses and strains entered as negative values. If possible, use compression or tension data depending on the application, since the fit of a single material model to both tensile and compressive data will normally be less accurate than for each individual test.

- Try to include test data from the planar test. This test measures shear behavior, which can be very important.

- Provide more data at the strain magnitudes that you expect the material will be subjected to during the simulation. For example, if the material will only have small tensile strains, say under 50%, do not provide much, if any, test data at high strain values (over 100%).

> **Note:** Most solid rubber materials have very little compressibility compared to their shear flexibility and are modeled as incompressible in a FEA simulation. This behavior is not a problem with plane stress, shell, or membrane elements. However, it can be a problem when using other elements, such as plane strain, axisymmetric, and three-dimensional solid elements. For example, in applications where the material is not highly confined, it would be quite satisfactory to assume that the material is fully incompressible: the volume of the material cannot change except for thermal expansion.
>
> In cases where the material is highly confined (such as an O-ring used as a seal), the compressibility must be modeled correctly to obtain accurate results.
>
> Compressibility is defined as the ratio of the initial bulk modulus K_0 to the initial shear modulus μ_0.
>
> Poisson's ratio, v, also provides a measure of compressibility since it is defined as $v = \dfrac{3(K_0/\mu_0) - 2}{6(K_0/\mu_0) + 2}$.

Linear vs Nonlinear Analysis

In a linear analysis there is a linear relationship between the applied loads and the induced response of the system. In a nonlinear analysis the response of the system is not a linear function of the magnitude of the applied loads.

Conditions for Linear Analysis

When loads are applied to a deformable body, internal forces and reactions are developed to render the body into a state of equilibrium. A linear analysis calculates displacements, strains, stresses, and reaction forces under the effect of applied loads.

For a linear analysis to be valid, the following assumptions must be true:

*   All materials in the model exhibit linear elastic behavior; that is, stresses are linearly proportional to strains (Hooke's law). If the load is removed, the geometry returns to its original shape with no permanent deformation.
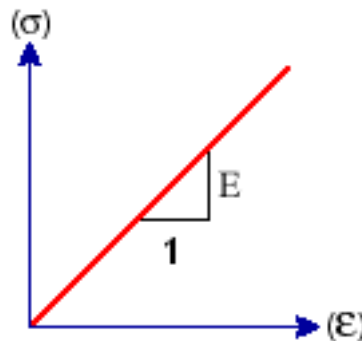
*   Deformations are small in relation to the dimensions of the model.
*   All loads are applied slowly and gradually until they reach their full magnitudes. After reaching their full magnitudes, loads remain constant in magnitude and direction.

    Note: This assumption allows us to neglect inertial and damping forces due to negligibly small accelerations and velocities. Time-variant loads that induce considerable inertial and damping forces may warrant dynamic analysis. Dynamic loads (such as earthquake, oscillatory, impact, or shock loads) change with time and in many cases induce considerable inertial and damping forces that cannot be neglected.

*   Boundary conditions do not vary during the application of loads. The loads do not cause changes in contact conditions between parts.

In a linear analysis there is a linear relationship between the applied loads and the induced response of the system. For example, if a linear spring extends statically by 1 m under a load of 10 N, it will extend by 2 m when a load of 20 N is applied.

This means that in a linear analysis the flexibility of the structure need only be calculated once (by assembling the stiffness matrix and inverting it). The linear response of the structure to other load cases can be found by multiplying the new vector of loads by the inverted stiffness matrix (or flexibility matrix).

In addition, the structure's response to various load cases can be scaled by constants to determine its response to a completely new load case, provided that the new load case is the sum (or multiple) of previous ones. This principle of superposition of load cases assumes that the same boundary conditions are used for all the load cases.

A linear static solution is not valid if any of these points are violated. The relationship between the applied loads (generalized force) and the response (generalized displacement) becomes nonlinear, and a nonlinear analysis must be performed to get accurate results that reflect the true behavior of the model.

A graphical representation of the analysis type based on the generalized force/displacement relationship is shown below.
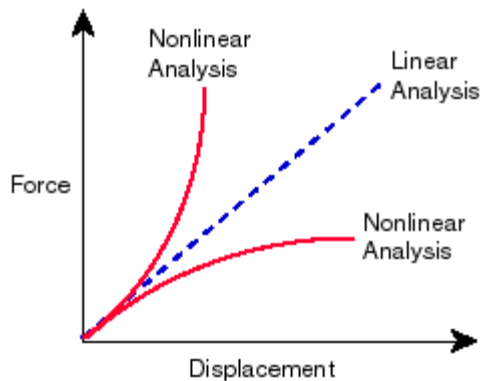
Sources of Nonlinearity

If the relationship between the applied loads (generalized force) and the response (generalized displacement) becomes nonlinear, a nonlinear analysis must be performed to get accurate results that reflect the true behavior of the model.

The following topics are discussed:
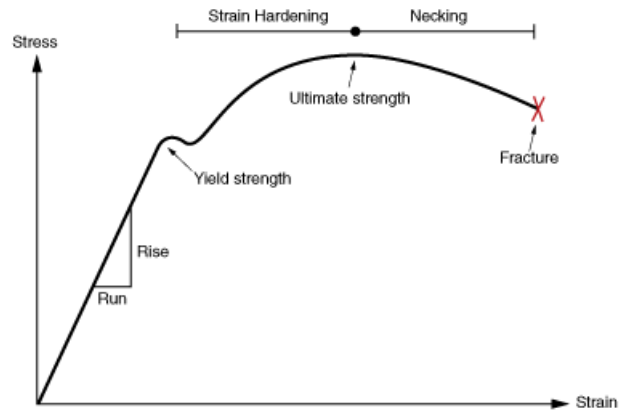
Material Nonlinearity

This class of nonlinear behavior stems from the nonlinear relationship between the stress and strain.

Most metals have a fairly linear stress/strain relationship at low strain values; but at higher strains the material yields, at which point the response becomes nonlinear and irreversible. Several factors can affect the stress-strain relationship, such as:

- Load history: plasticity problems
- Load duration: creep analysis, viscoelasticity
- Temperature: thermo-plasticity

A typical stress-strain diagram of a ductile material model is shown in the figure below. The elastic range of the material ends when the stress has reached the yielding point. Once the material has reached the yielding point, permanent deformation, which is non-reversible, begins to develop. As the tensile stress increases, the plastic deformation is characterized by a strain hardening region, necking region, and finally fracture.

Rubber materials can be approximated by a nonlinear, reversible (elastic) response. A typical stress-strain curve of a rubber material is shown below.
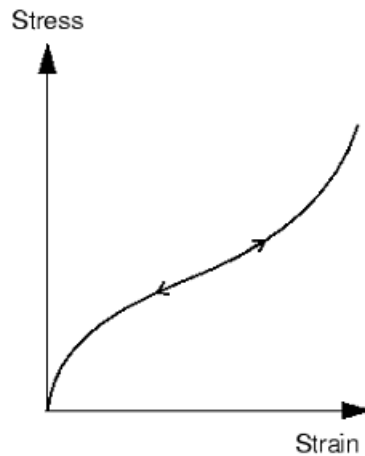

Boundary Nonlinearity

This class of nonlinear behavior occurs if the boundary conditions change during the analysis. Boundary nonlinearities occur in manufacturing processes, such as forging and stamping.

An example of boundary nonlinearity is blowing a sheet of material into a mold. The sheet expands relatively easily under the applied pressure until it begins to contact the mold. From then on, because of the change in boundary conditions, the pressure must be increased to continue forming the sheet.

Note: Boundary nonlinearities are extremely discontinuous: when contact occurs during a simulation, there is an instantaneous change in the response of the structure.

Consider the cantilever beam that deflects under an applied load until it hits a "stop."
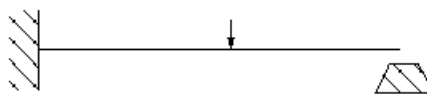
The vertical deflection of the tip is linearly related to the load (if the deflection is small) until it contacts the stop. There is then a sudden change in the boundary condition at the tip of the beam, preventing any further vertical deflection; hence, the response of the beam is no longer linear.

Geometric Nonlinearity

The third source of nonlinearity is related to changes in the geometry of the structure during the analysis. Geometric nonlinearities occur whenever the magnitude of the displacements affects the response of the structure. This may be caused by:

- Large deflections or rotations
- Snap through behavior
- Initial stresses or load stiffening

In general, large displacements can cause the structure to respond in a stiffening or softening manner as shown in the figure below.
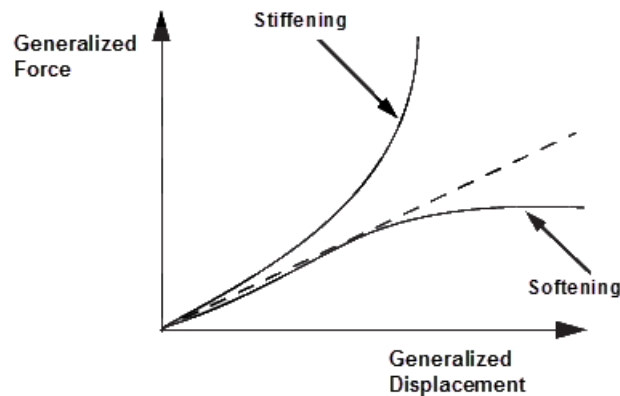
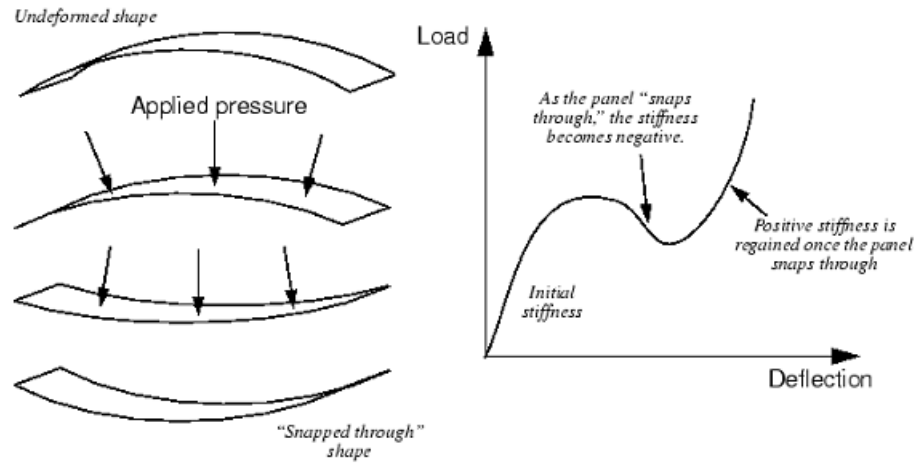For example, consider a cantilever beam loaded vertically at the tip.
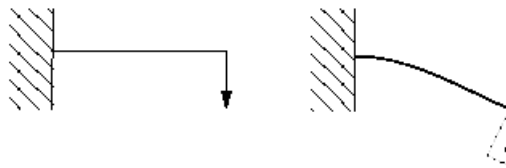
If the tip deflection is small, the analysis can be considered as being approximately linear. However, if the tip deflections are large, the shape of the structure and, hence, its stiffness changes. In addition, if the load does not remain perpendicular to the beam, the action of the load on the structure changes significantly.

As the cantilever beam deflects, the load can be resolved into a component perpendicular to the beam and a component acting along the length of the beam. Both of these effects contribute to the nonlinear response of the cantilever beam (i.e., the changing of the beam's stiffness as the load it carries increases).

One would expect large deflections and rotations to have a significant effect on the way that structures carry loads. However, displacements do not necessarily have to be large relative to the dimensions of the structure for geometric nonlinearity to be important.

Consider the "snap through" under applied pressure of a large panel with a shallow curve.

Undeformed shape

Applied pressure

"Snapped through" shape

Load

As the panel "snaps through," the stiffness becomes negative.

Positive stiffness is regained once the panel snaps through

Initial stiffness

Deflection

In this example there is a dramatic change in the stiffness of the panel as it deforms. As the panel "snaps through," the stiffness becomes negative. Thus, although the magnitude of the displacements, relative to the panel's dimensions, is quite small, there is significant geometric nonlinearity in the simulation, which must be taken into consideration.

Note: To obtain solutions for nonlinear problems, the Newton-Raphson numerical method is most often used. In a nonlinear analysis the solution cannot be calculated by solving a single system of equations, as would be done in a linear problem. Instead, the solution is found by applying the specified loads gradually and incrementally working toward the final solution. At the end of each load increment, an approximate equilibrium configuration is reached after several iterations. The sum of all of the incremental responses is the approximate solution for the nonlinear analysis.

Results

The final step of the FEA process involves the visualization and interpretation of results. The FEA results should be qualified in terms of accuracy and correctness. It is important to verify that the results are accurate and fulfill the original design goals of the analysis.

Guidelines for Interpreting Results

In structural simulations you are usually interested in viewing plots of deformed shapes and stress distributions; for thermal simulations, temperature contour plots and heat flux vector plots; for frequency studies, fundamental frequencies and mode shapes.

Results accuracy relates to the convergence level and quality of the solution method. It is good practice to assess the correctness of results through the validation of all your modeling assumptions.

When viewing FEA results:

- Check the displacement results. Is the order of magnitude of the deformations what you are expecting? Unexpected discrepancies could be caused by an inconsistent set of units and improper loading definitions. If the displacement magnitudes are so large that the linear assumption is violated, consider running a nonlinear analysis. Is the overall deformed shape in agreement with the applied boundary conditions and loading definitions? Animations of the displacement results from the model's initial undeformed state to its fully deformed shape help you visualize the response of your model and detect any modeling errors.

- Check the stress results. Are the magnitudes of the stress results in line with what you are expecting? Examine the regions of high stress concentration. Are these caused by "bad" quality mesh elements? Consider refining the mesh locally at areas of high stress concentration to resolve any convergence issues. Eliminating insignificant geometry features (such as narrow faces, sharp corners, or short edges) can also help to eliminate fictitious high stress values. Verify that the transition of stress values is smooth throughout the geometry.

  Stress results are generally used to predict yielding or guard against failure. Stress quantities are related to the failure criteria that best describe the particular material. The most popular stress quantity to display when evaluating the onset of yield for ductile materials is the von Mises stress. The material starts yielding when its von Mises stress reaches a critical value known as the yield strength.

- Verify that the reactions at the supports balance the applied forces.

- Review and qualify your modeling assumptions. Keep in mind the inherent assumptions of each input variable in your FEA model (loads, restraints, material properties, element formulation, solution method). When you interpret results, review these assumptions and try to quantify their effects on the solution. You must have a good understanding of the mechanics of materials, the potential failure modes of the products, and the product's actual operating environment.

  > ✏️ **Note:**
  >
  > The consideration of the proper restraints (fixtures, rollers, or hinges) that define how the model interacts with the surrounding environment is a challenging task, which carries significant modeling uncertainty. Over-simplification of the true support structure often leads to erroneous results. Small changes in the support conditions can cause large changes in the results.

- Verify that the solution satisfies the original design intent. The setup of an FEA model serves a design purpose; for example, minimize deflection at a certain location, control oscillation amplitudes, reduce weight of a

component, or predict possible failure modes. It is important to succeed in getting meaningful result quantities that best serve your original design goals.