

Chapter I

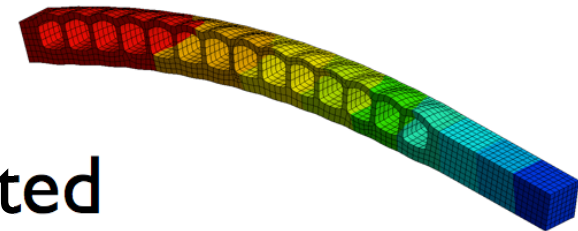
Introduction

- I.1 Case Study: Pneumatically Actuated PDMS Fingers
- I.2 Structural Mechanics: A Quick Review
- I.3 Finite Element Methods: A Concise Introduction
- I.4 Failure Criteria of Materials
- I.5 Review

Section I.I

Case Study: Pneumatically Actuated PDMS Fingers

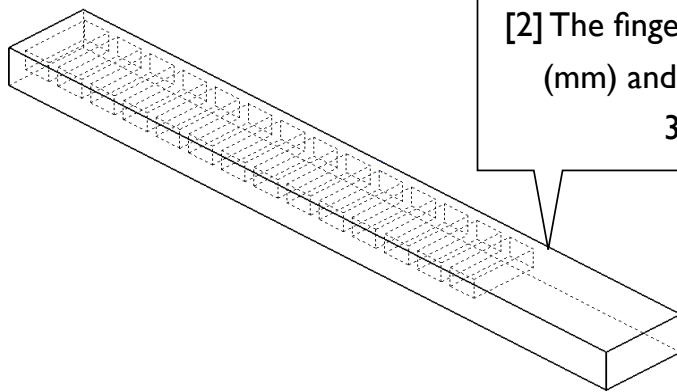
Problem Description



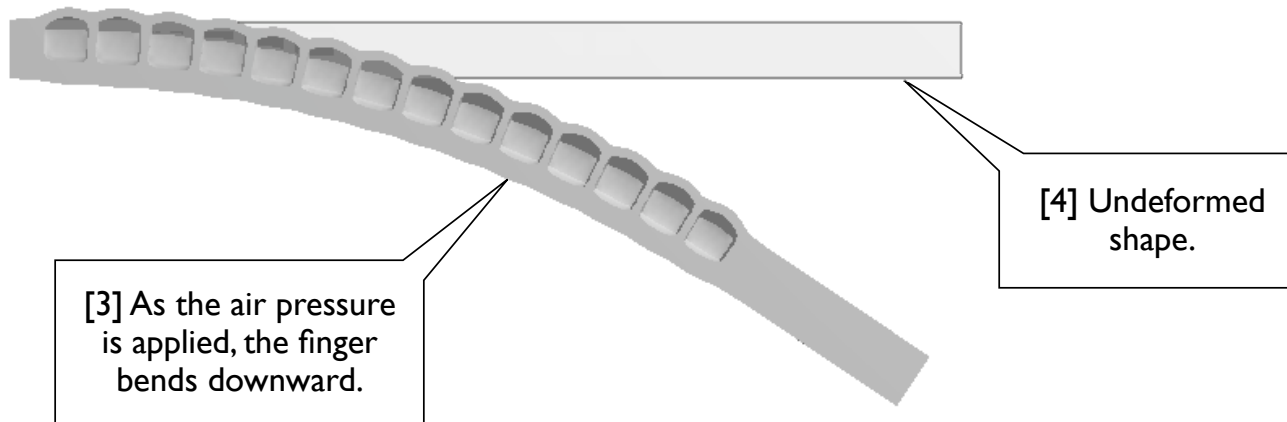
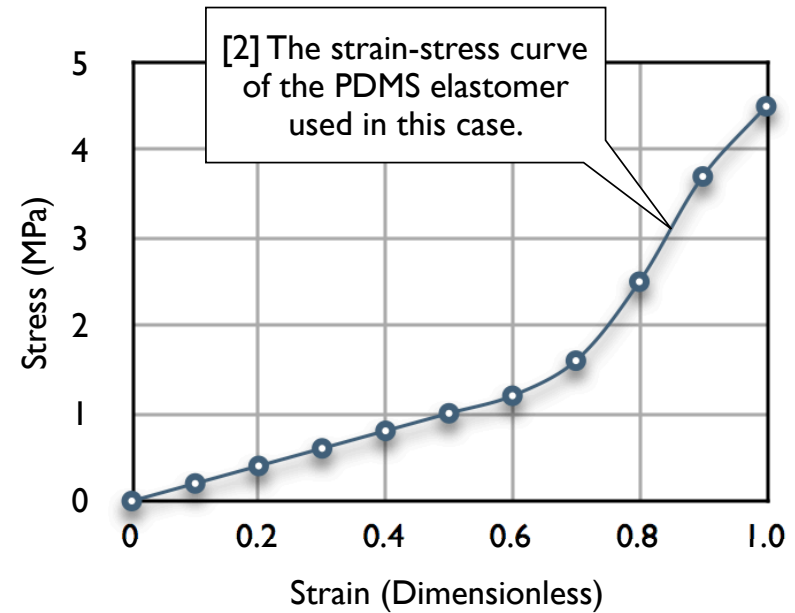
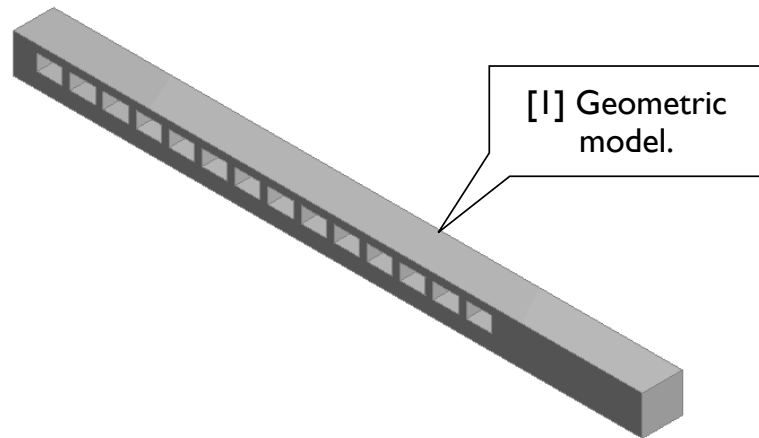
[1] A robot hand has five fingers, remotely controlled by a surgeon through internet.



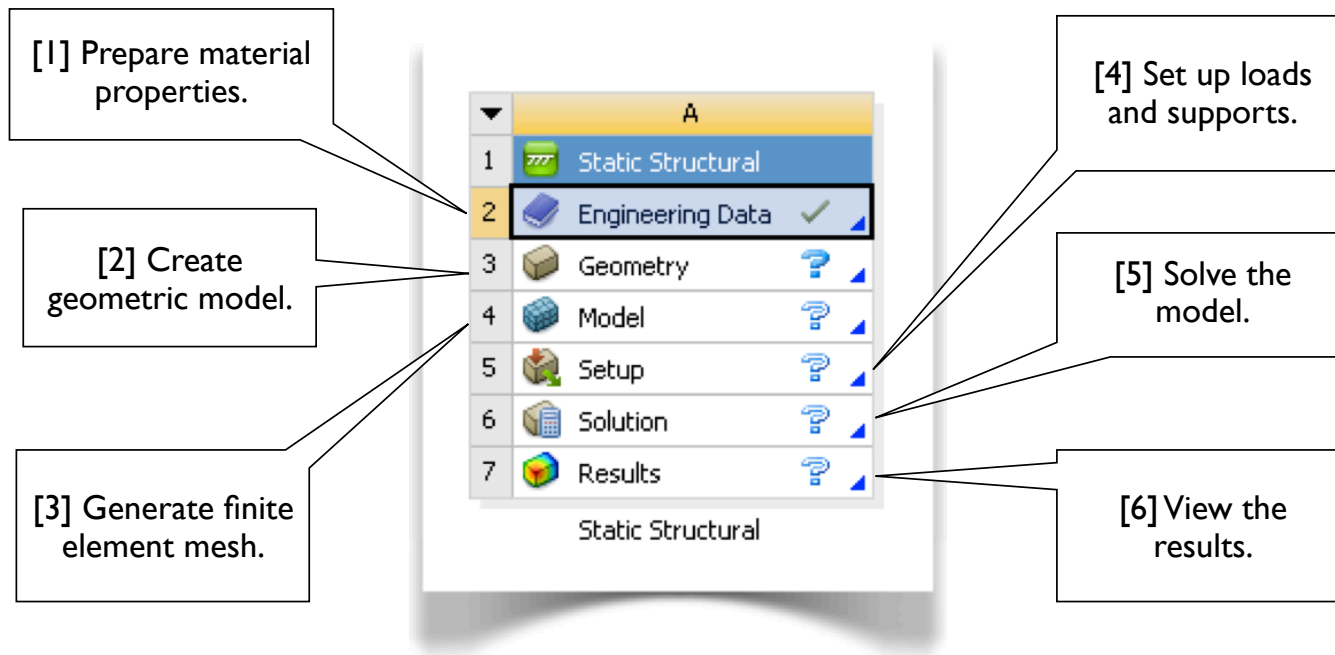
[2] The finger has a size of 80x5x10.2 (mm) and 14 air chambers, each 3.2x2x8 (mm).



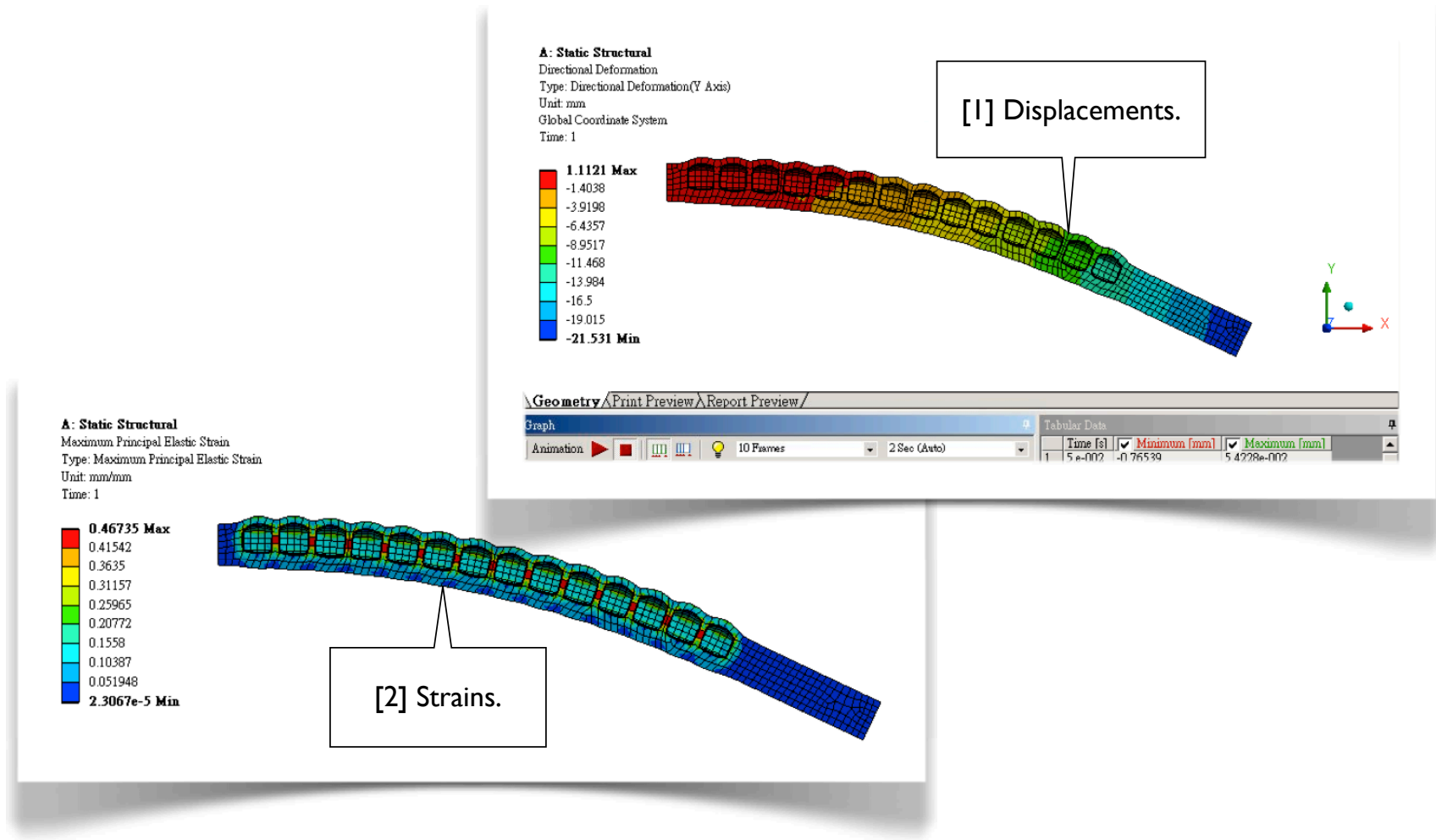
Problem Description



Static Structural Simulations

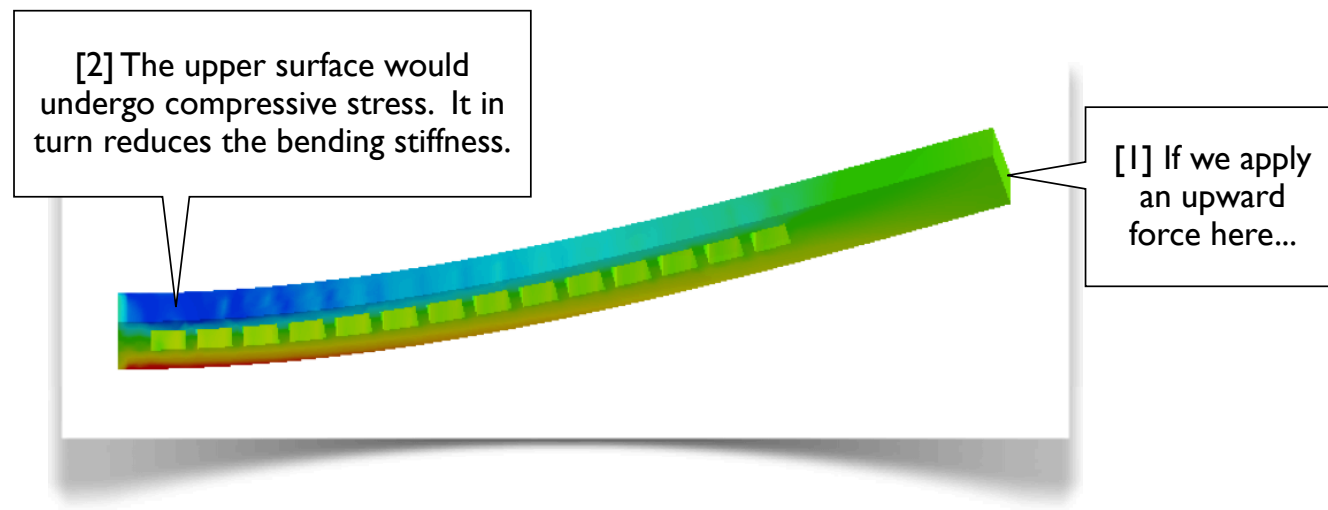


Static Structural Simulations



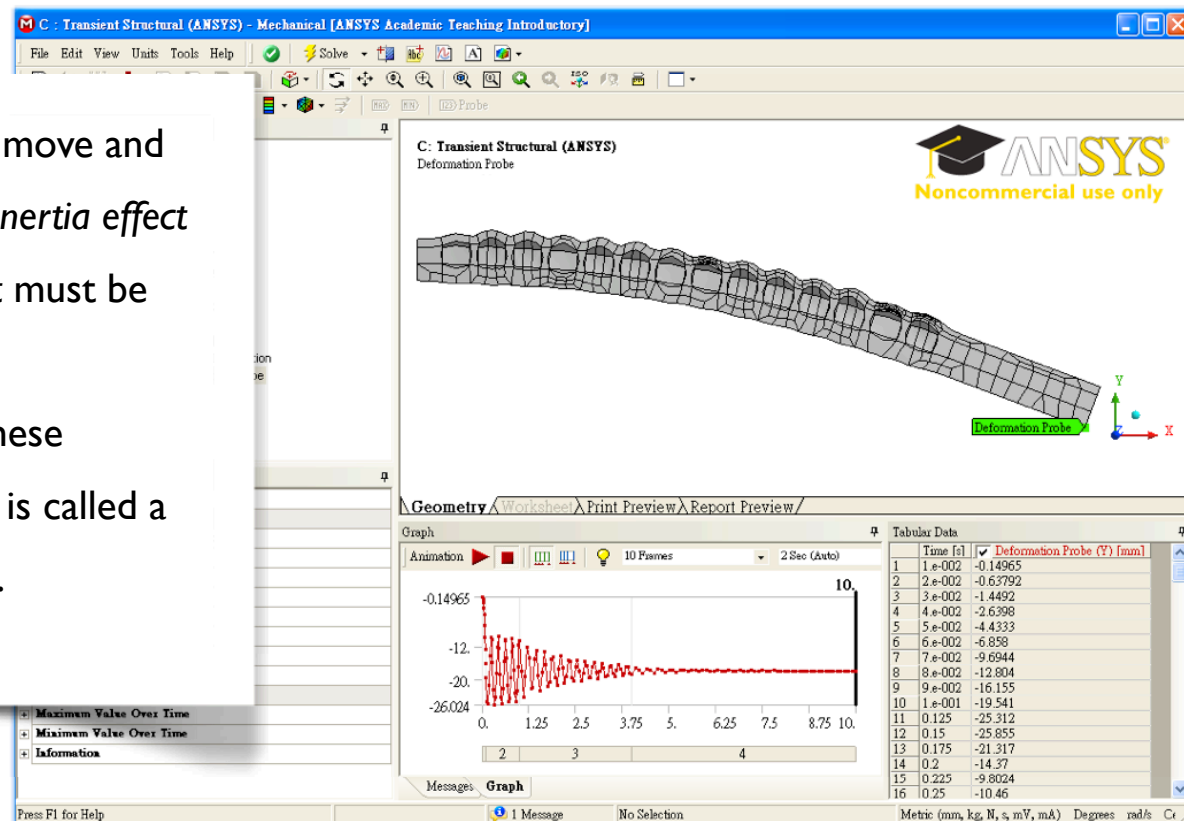
Buckling and Stress-Stiffening

- *Stress-stiffening*: bending stiffness increases with increasing axial tensile stress, e.g., guitar string.
- The opposite also holds: bending stiffness decreases with increasing axial compressive stress.
- *Buckling*: phenomenon when bending stiffness reduces to zero; i.e., the structure is unstable. Usually occurs in slender columns, thin walls, etc.
- Purpose of a buckling analysis is to find *buckling loads* and *buckling modes*.



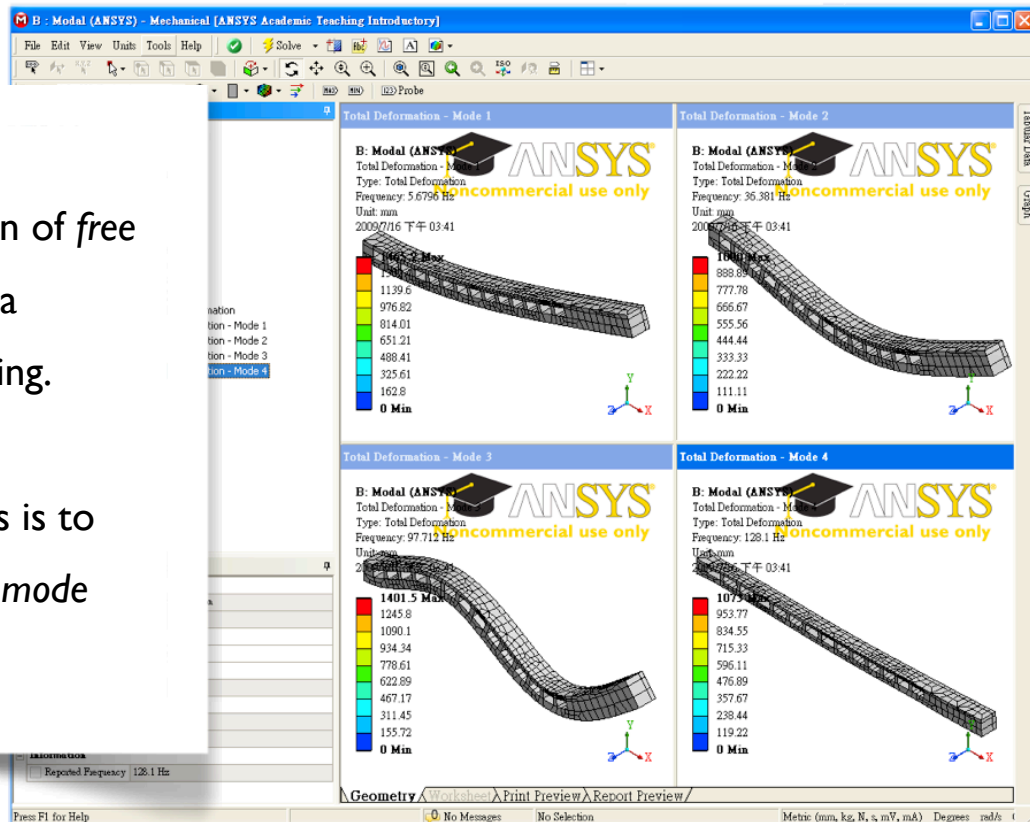
Dynamic Simulations

- When the bodies move and deform very fast, *inertia effect* and *damping effect* must be considered.
- When including these dynamic effects, it is called a *dynamic simulation*.



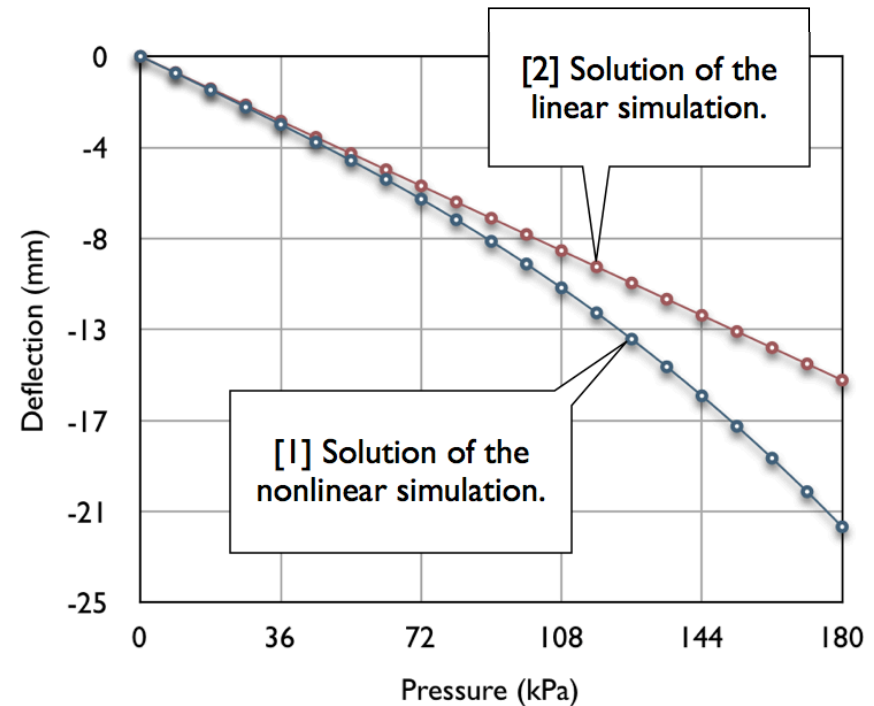
Modal Analysis

- A special case of dynamic simulations is the simulation of *free vibrations*, the vibrations of a structure without any loading.
- It is called a *modal analysis*.
- Purpose of a modal analysis is to find *natural frequencies* and *mode shapes*.



Structural Nonlinearities

- Linear simulations assume that the response is linearly proportional to the loading.
- When the solution deviates from the reality, a nonlinear simulation is needed.
- Structural nonlinearities come from large deformation, topology changes, nonlinear stress-strain relationship, etc.

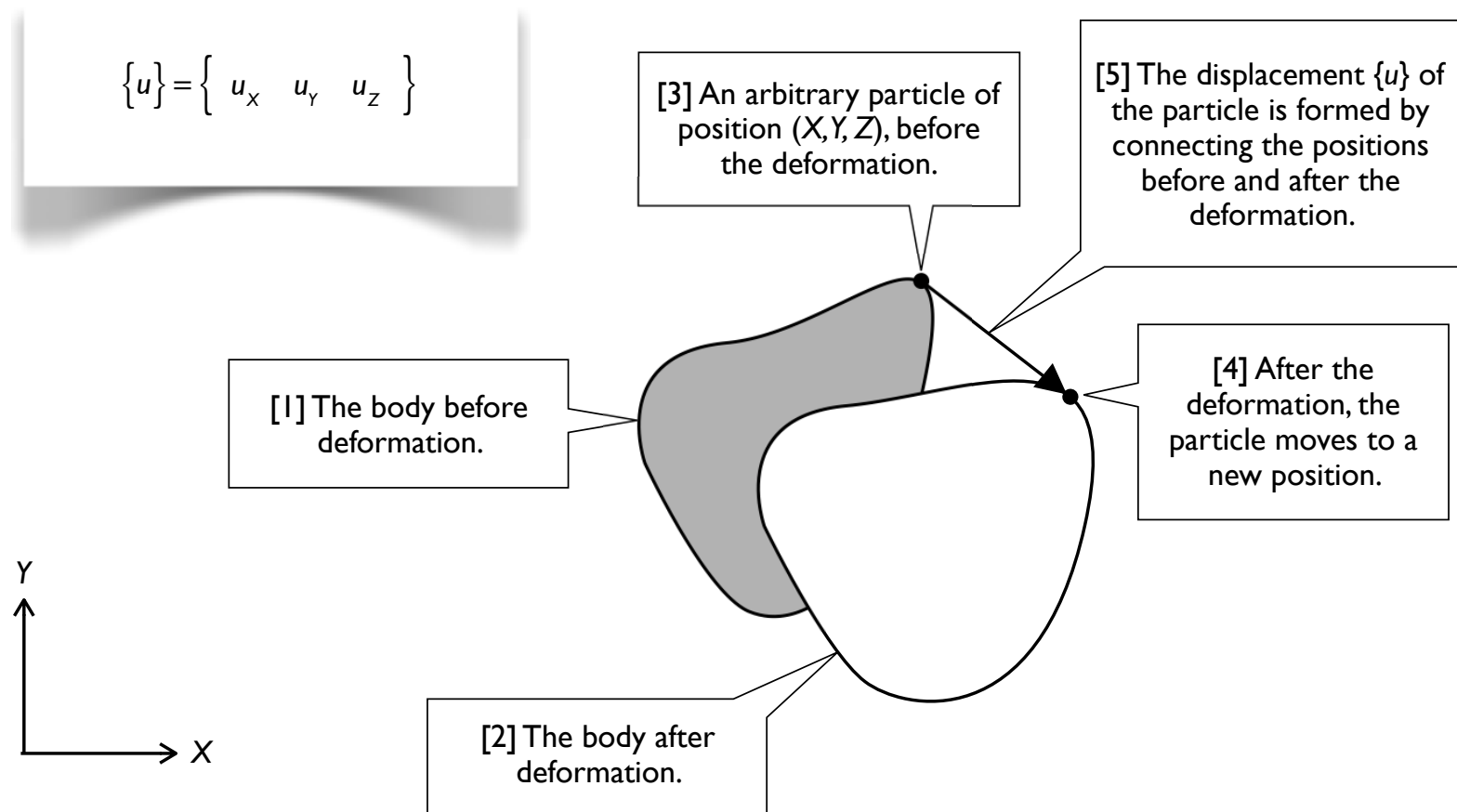


Section I.2

Structural Mechanics: A Quick Review

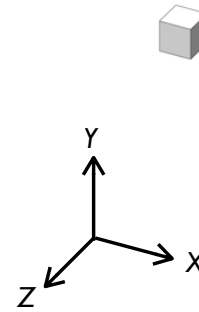
- *Engineering simulation*: finding the *responses* of a *problem domain* subject to *environmental conditions*.
- *Structural simulation*: finding the *responses* of *bodies* subject to *environmental conditions*.
 - The bodies are described by geometries and materials.
 - Environment conditions include supports and loading conditions.
 - Responses can be described by displacements, strains, or stresses.

Displacements



Stresses

- The stress at a certain point is *the force per unit area acting on the boundary faces of an infinitesimally small body centered at that point.*
- The stress values may be different at different faces.
- The small body can be any shape.
- we usually use an infinitesimally small cube of which each edge is parallel to a coordinate axis.
- If we can find the stresses on a small cube, we can calculate the stresses on any other shapes of small body.

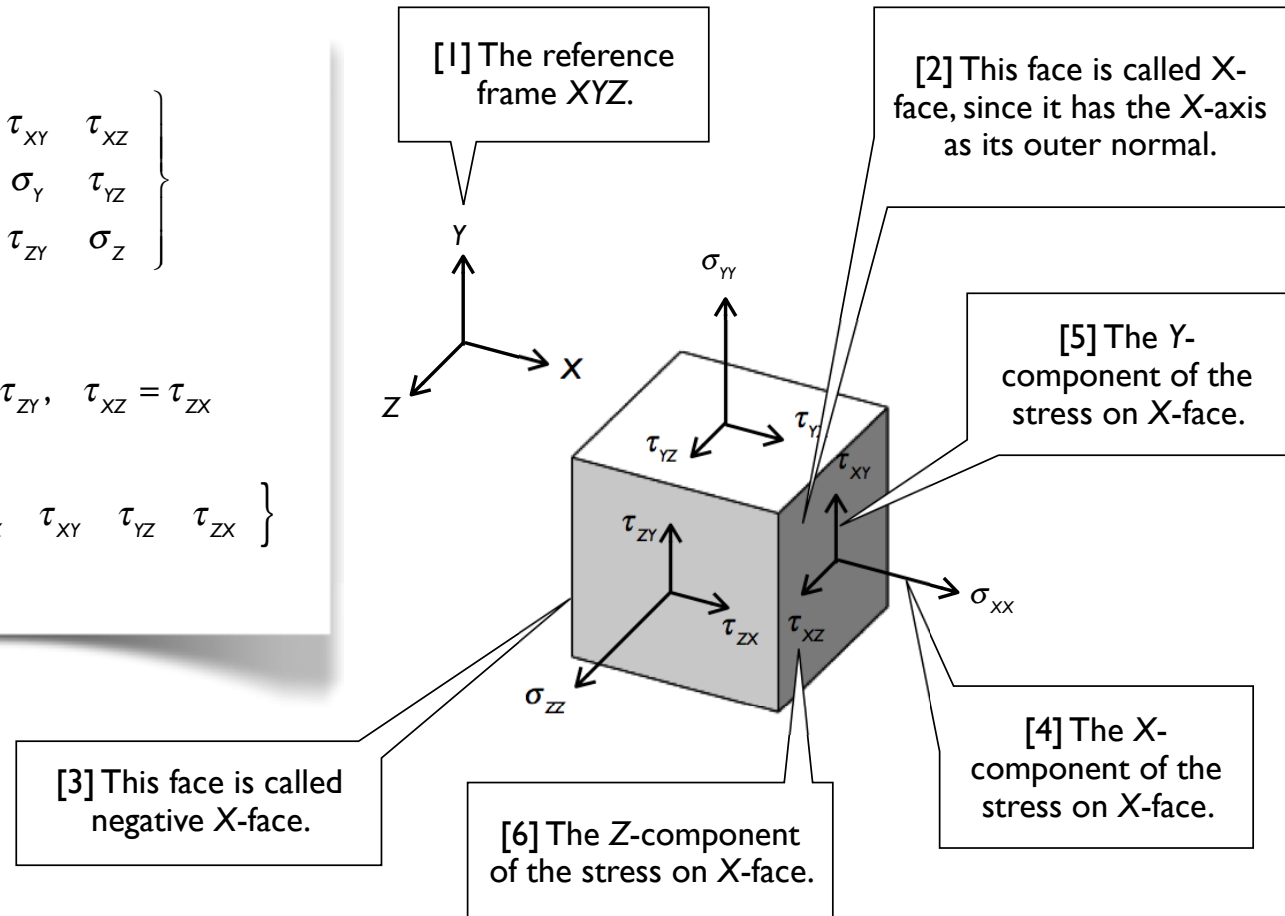


Stresses

$$\{\sigma\} = \begin{Bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \\ \tau_{zx} & \tau_{zy} & \sigma_z \end{Bmatrix}$$

$$\tau_{xy} = \tau_{yx}, \quad \tau_{yz} = \tau_{zy}, \quad \tau_{xz} = \tau_{zx}$$

$$\{\sigma\} = \begin{Bmatrix} \sigma_x & \sigma_y & \sigma_z & \tau_{xy} & \tau_{yz} & \tau_{zx} \end{Bmatrix}$$



Strains

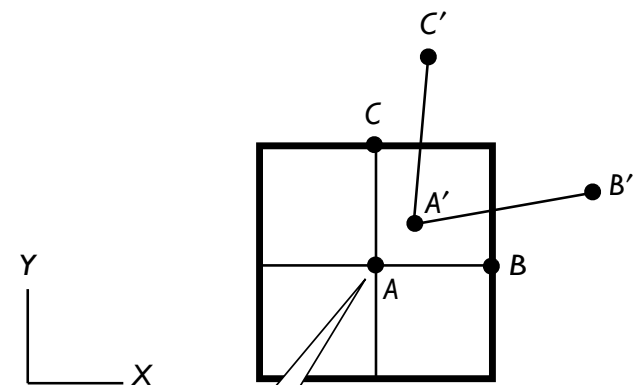
$$\varepsilon_x = \frac{A'B' - AB}{AB} \text{ (dimensionless)}$$

$$\varepsilon_y = \frac{A'C' - AC}{AC} \text{ (dimensionless)}$$

$$\gamma_{xy} = \angle CAB - \angle C'A'B' \text{ (rad)}$$

$$\{\varepsilon\} = \left\{ \varepsilon_x \quad \varepsilon_y \quad \varepsilon_z \quad \gamma_{xy} \quad \gamma_{yz} \quad \gamma_{zx} \right\}$$

[2] After deformation, ABC becomes A'B'C'. Assume the deformation is infinitesimally .

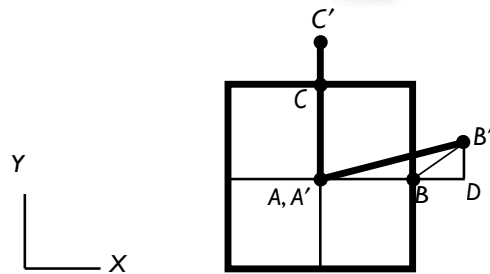


[1] Strains are defined as the displacements of a point relative to its neighboring points. To study the strain at A, consider its neighboring points B and C, which are along X-axis and Y-axis respectively.

Strains

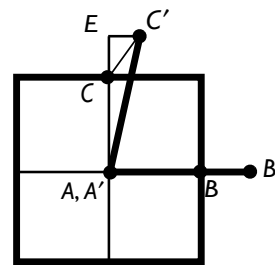
$$\epsilon_x = \frac{BD}{AB} \text{ (dimensionless)}$$

$$\gamma_{xy} = \frac{DB'}{AB} \text{ (rad)}$$



$$\epsilon_y = \frac{CE}{AC} \text{ (dimensionless)}$$

$$\gamma_{yx} = \frac{EC'}{AC} \text{ (rad)}$$



Governing Equations

$$\{u\} = \begin{Bmatrix} u_x & u_y & u_z \end{Bmatrix}$$

$$\{\sigma\} = \begin{Bmatrix} \sigma_x & \sigma_y & \sigma_z & \tau_{xy} & \tau_{yz} & \tau_{zx} \end{Bmatrix}$$

$$\{\varepsilon\} = \begin{Bmatrix} \varepsilon_x & \varepsilon_y & \varepsilon_z & \gamma_{xy} & \gamma_{yz} & \gamma_{zx} \end{Bmatrix}$$

15 quantities

- Equilibrium Equations (3 Equations)
- Strain-Displacement Relations (6 Equations)
- Stress-Strain Relations (6 Equations)

Stress-Strain Relations: Hooke's Law

$$\varepsilon_x = \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E}$$

$$\varepsilon_y = \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E}$$

$$\varepsilon_z = \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E}$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}$$

$$\gamma_{yz} = \frac{\tau_{yz}}{G}$$

$$\gamma_{zx} = \frac{\tau_{zx}}{G}$$

$$G = \frac{E}{2(1 + \nu)}$$

- For *isotropic, linearly elastic* materials, *Young's modulus* (E) and *Poisson's ratio* (ν) are used to fully describe the stress-strain relations.
- The Hooke's law is called a *material model*.
- The Young's modulus and the Poisson's ratio are called the *material parameters* of the material model.

Stress-Strain Relations: Hooke's Law

Thermal Effects Included

$$\varepsilon_x = \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} + \alpha \Delta T$$

$$\varepsilon_y = \frac{\sigma_y}{E} - \nu \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} + \alpha \Delta T$$

$$\varepsilon_z = \frac{\sigma_z}{E} - \nu \frac{\sigma_x}{E} - \nu \frac{\sigma_y}{E} + \alpha \Delta T$$

$$\gamma_{xy} = \frac{\tau_{xy}}{G}$$

$$\gamma_{yz} = \frac{\tau_{yz}}{G}$$

$$\gamma_{zx} = \frac{\tau_{zx}}{G}$$

- If temperature changes (thermal loads) are involved, the coefficient of thermal expansion, (CTE, α) must be specified in material properties.

Section I.3

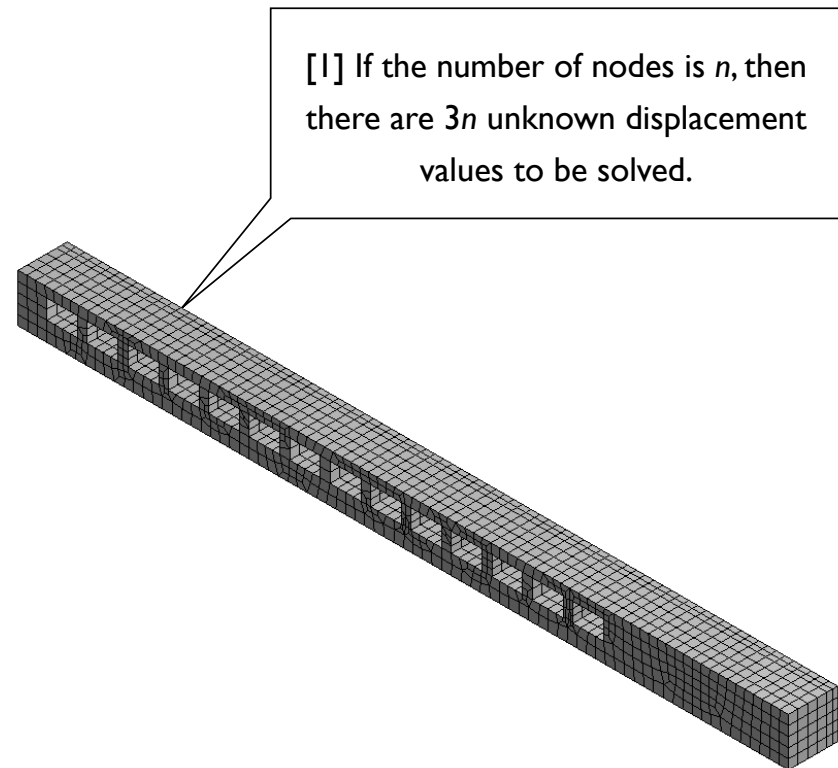
Finite Element Methods: A Conceptual Introduction

Basic Ideas

- A basic idea of finite element methods is to divide the structural body into small and geometrically simple bodies, called *elements*, so that equilibrium equations of each element can be written, and all the equilibrium equations are solved simultaneously
- The elements are assumed to be connected by *nodes* located on the elements' edges and vertices.

Basic Ideas

- Another idea is to solve unknown discrete values (displacements at the nodes) rather than to solve unknown functions (displacement fields).
- Since the displacement on each node is a vector and has three components (in 3D cases), the number of total unknown quantities to be solved is three times the number of nodes.
- The nodal displacement components are called the *degrees of freedom* (DOF's) of the structure.



Basic Ideas

- In static cases, the system of equilibrium equations has following form:

$$[K]\{D\} = \{F\}$$

- The displacement vector $\{D\}$ contains displacements of $3n$ degrees of freedom, where n is the number of nodes.
- The force vector $\{F\}$ contains forces acting on all degrees of freedom.
- The matrix $[K]$ is called the *stiffness matrix* of the structure. In a special case when the structure is a spring, then $\{F\}$ is the external force, $\{D\}$ is the deformation of the spring, and $[K]$ is the spring constant.

Procedure of Finite Element Method

1. Given the bodies' geometries, material properties, support conditions, and loading conditions.
2. Divide the bodies into elements.
3. Establish the equilibrium equation: $[K] \{D\} = \{F\}$
 - 3.1 Construct the $[K]$ matrix, according to the elements' geometries and the material properties.
 - 3.2 Most of components in $\{F\}$ can be calculated, according to the loading conditions.
 - 3.3 Most of components in $\{D\}$ are unknown. Some component, however, are known, according to the support conditions.
 - 3.4 The total number of unknowns in $\{D\}$ and $\{F\}$ should be equal to the total number of degrees of freedom of the structure.

Procedure of Finite Element Method

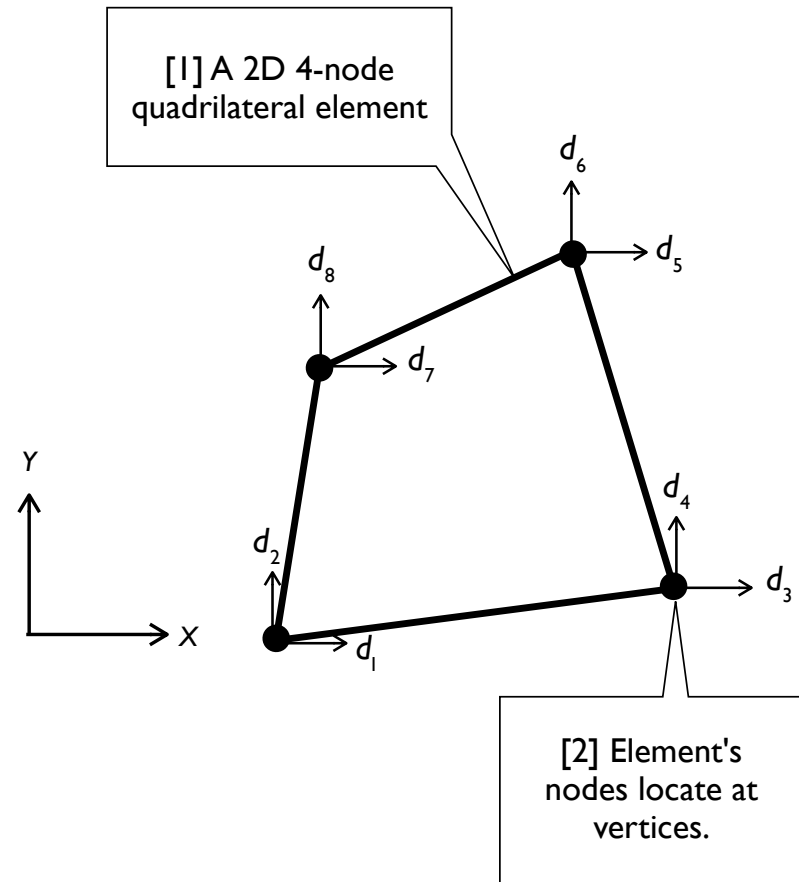
4. Solve the equilibrium equation. Now, the nodal displacements $\{d\}$ of each element are known.
5. For each element:
 - 5.1 Calculate displacement fields $\{u\}$, using an interpolating method, $\{u\} = [N] \{d\}$. The interpolating functions in $[N]$ are called the *shape functions*.
 - 5.2 Calculate strain fields according to the strain-displacement relations.
 - 5.3 Calculate stress fields according to the stress-strain relations (Hooke's law).

Shape Functions

- Shape functions serve as interpolating functions, allowing the calculation of displacement fields (functions of X, Y, Z) from nodal displacements (discrete values).

$$\{u\} = [N]\{d\}$$

- For elements with nodes at vertices, the interpolation must be linear and thus the shape functions are linear (of X, Y, Z).



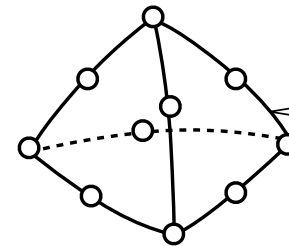
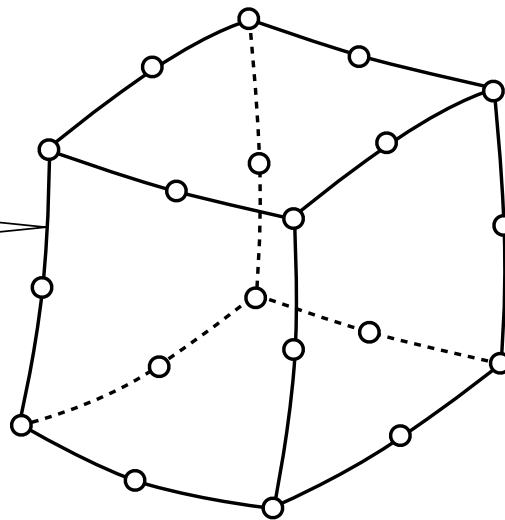
Shape Functions

- For elements with nodes at vertices as well as at middles of edges, the interpolation must be quadratic and thus the shape functions are quadratic (of X, Y, Z).
- Elements with linear shape functions are called *linear elements*, *first-order elements*, or *lower-order elements*.
- Elements with quadratic shape functions are called *quadratic elements*, *second-order elements*, or *higher-order elements*.
- ANSYS Workbench supports only first-order and second-order elements.

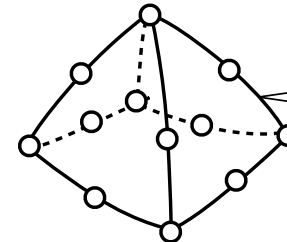
Workbench Elements

3D Solid Bodies

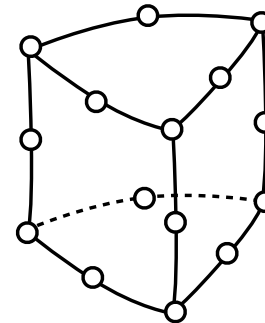
[1] 3D 20-node
structural solid.
Each node has 3
translational
degrees of
freedom: D_x , D_y ,
and D_z .



[4] Tetrahedron.



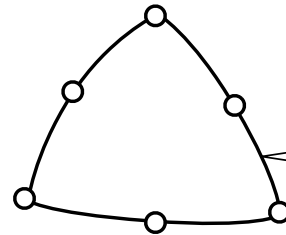
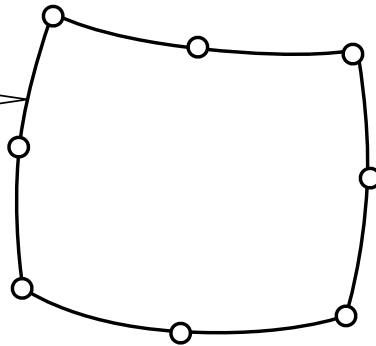
[3] Quadrilateral-
based pyramid.



[2] Triangle-based
prism.

2D Solid Bodies

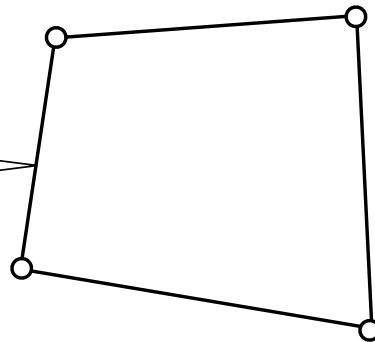
[5] 2D 8-node
structural solid.
Each node has 2
translational
degrees of
freedom: D_x and
 D_y .



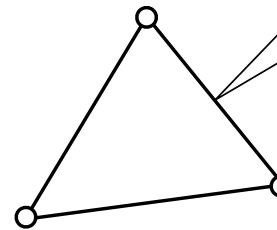
[6] Degenerated
Triangle.

3D Surface Bodies

[7] 3D 4-node *structural shell*. Each node has 3 translational and 3 rotational degrees of freedom: D_X , D_Y , D_Z , R_X , R_Y , and R_Z .

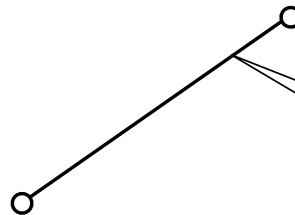


[8] Degenerated Triangle



3D Line Bodies

[9] 3D 2-Node *beam*. Each node has 3 translational and 3 rotational degrees of freedom: D_X , D_Y , D_Z , R_X , R_Y , R_Z .



Section I.4

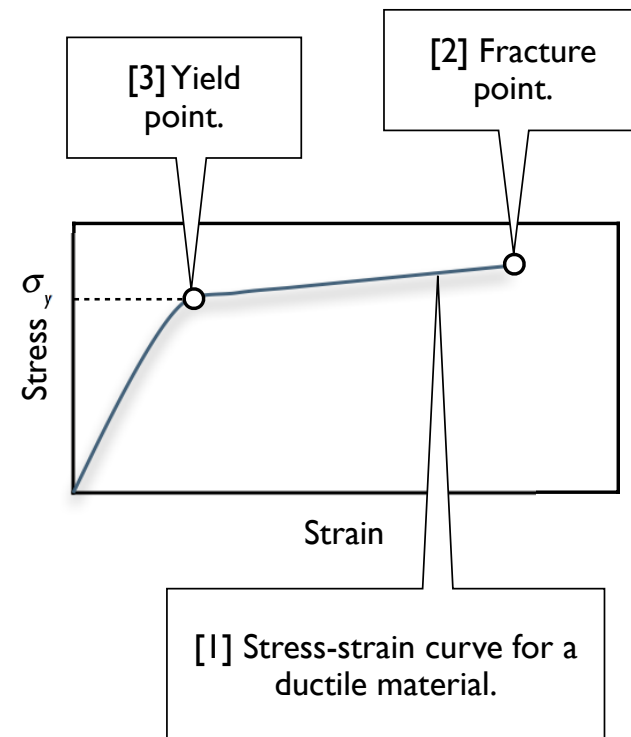
Failure Criteria of Materials

Ductile versus Brittle Materials

- A *Ductile material* exhibits a large amount of strain before it fractures.
- The fracture strain of a *brittle material* is relatively small.
- Fracture strain is a measure of ductility.

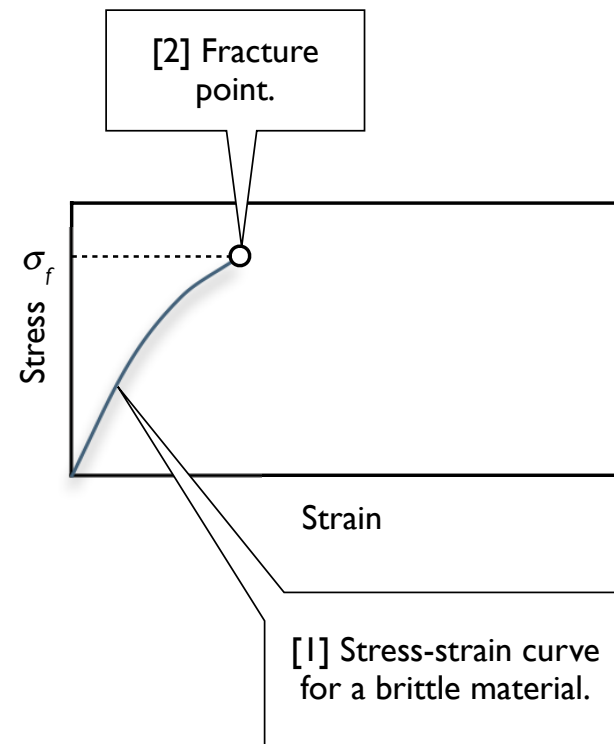
Failure Points for Ductile Materials

- Mild steel is a typical ductile material.
- For ductile materials, there often exists an obvious yield point, beyond which the deformation would be too large so that the material is no longer reliable or functional; the failure is accompanied by excess deformation.
- Therefore, for these materials, we are most concerned about whether the material reaches the yield point σ_y .



Failure Points for Brittle Materials

- Cast iron and ceramics are two examples of brittle materials.
- For brittle materials, there usually doesn't exist obvious yield point, and we are concerned about their fracture point σ_f .

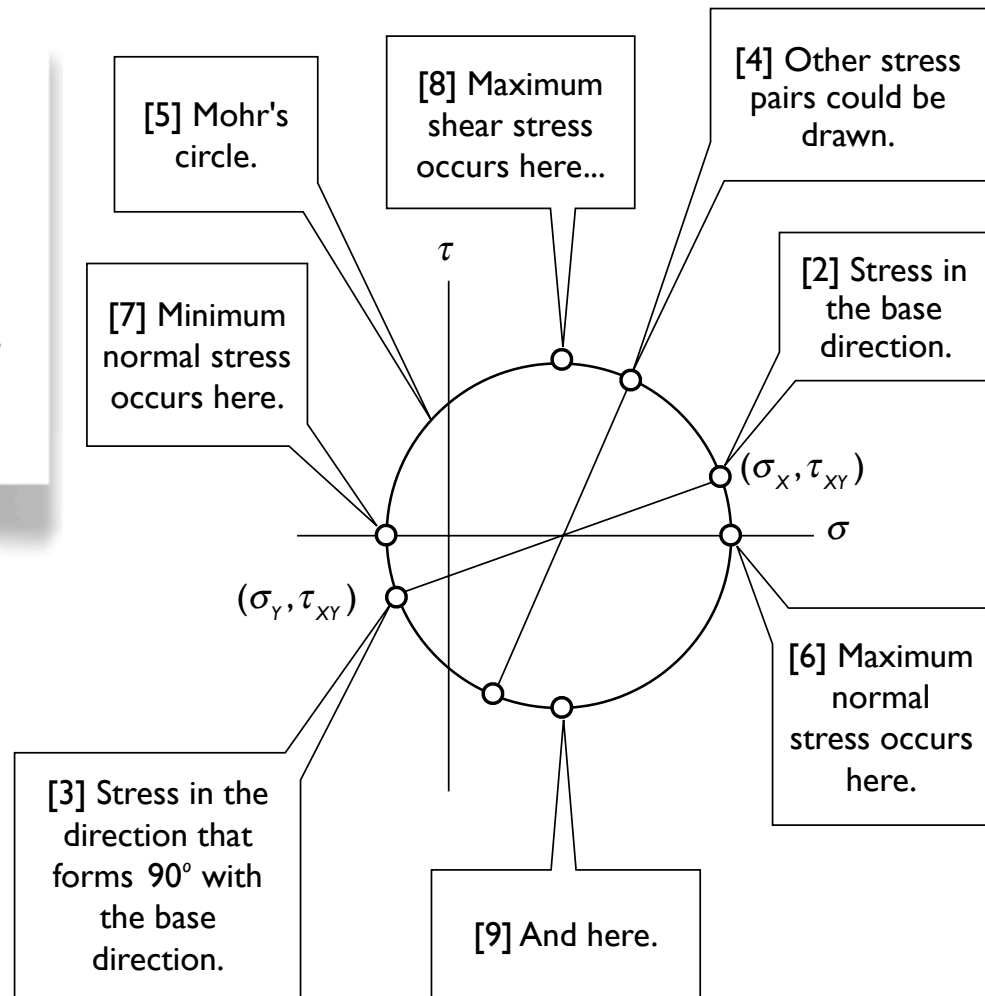
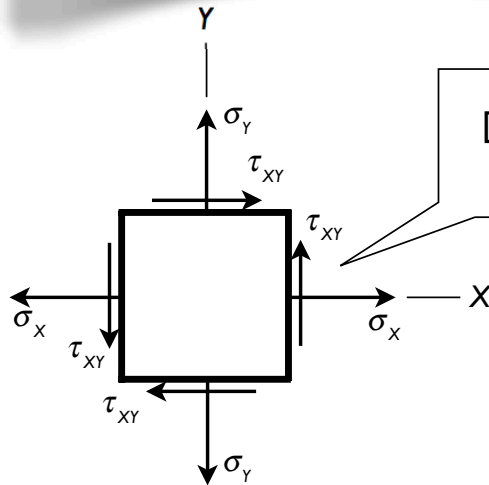


Failure Modes

- The fracture of brittle materials is mostly due to *tensile failure*.
- The yielding of ductile materials is mostly due to *shear failure*

Principal Stresses

- A direction in which the shear stress vanishes is called a *principal direction*.
- The corresponding normal stress is called a *principal stress*.



Principal Stresses

- At any point of a 3D solid, there are three principal directions and three principal stresses.
- The maximum normal stress is called the *maximum principal stress* and denoted by σ_1 .
- The minimum normal stress is called the *minimum principal stress* and denoted by σ_3 .
- The *medium principal stress* is denoted by σ_2 .
- The maximum principal stress is often a positive value, a tension; the minimum principal stress is often a negative value, a compression.

Failure Criterion for Brittle Materials

- The failure of brittle materials is a tensile failure. In other words, a brittle material fractures because its tensile stress reaches the fracture strength σ_f .
- We may state a failure criterion for brittle materials as follows: At a certain point of a body, if the maximum principal stress reaches the fracture strength of the material, it will fail.
- In short, a point of material fails if

$$\sigma_1 \geq \sigma_f$$

Tresca Criterion for Ductile Materials

- The failure of ductile materials is a shear failure. In other words, a ductile material yields because its shear stress reaches the shear strength τ_y of the material.
- We may state a failure criterion for ductile materials as follows: At a certain point of a body, if the maximum shear stress reaches the shear strength of the material, it will fail.
- In short, a point of material fails if

$$\tau_{\max} \geq \tau_y$$

- It is easy to show (using Mohr's circle) that

$$\tau_{\max} = \frac{\sigma_1 - \sigma_3}{2}$$

$$\tau_y = \frac{\sigma_y}{2}$$

- Thus, the material yields if

$$\sigma_1 - \sigma_3 \geq \sigma_y$$

- $(\sigma_1 - \sigma_3)$ is called the stress intensity.

Von Mises Criterion for Ductile Materials

- In 1913, Richard von Mises proposed a theory for predicting the yielding of ductile materials. The theory states that the yielding occurs when the deviatoric strain energy density reaches a critical value, i.e.,

$$w^d \geq w^{yd}$$

- It can be shown that the yielding deviatoric energy in uniaxial test is

$$w^{yd} = \frac{(1+\nu)\sigma_y^2}{3E}$$

- And the deviatoric energy in general 3D cases is

$$w^d = \frac{1+\nu}{6E} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]$$

Von Mises Criterion for Ductile Materials

- After substitution and simplification, the criterion reduces to that the yielding occurs when

$$\sqrt{\frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]} \geq \sigma_y$$

- The quantity on the left-hand-side is termed *von Mises stress* or *effective stress*, and denoted by σ_e ; in ANSYS, it is also referred to as *equivalent stress*,

$$\sigma_e = \sqrt{\frac{1}{2} \left[(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2 \right]}$$