Static Structures and Finite Element Analysis

Brian Mercer
Mechanical Science and Engineering

ILLINOIS
Mechanical Science & Engineering
GRAINGER COLLEGE OF ENGINEERING

What is the finite element method?

- The **F**inite **E**lement **M**ethod (FEM) is a mathematical framework that can be used to find approximate solutions to **p**artial **d**ifferential **e**quations (PDEs)

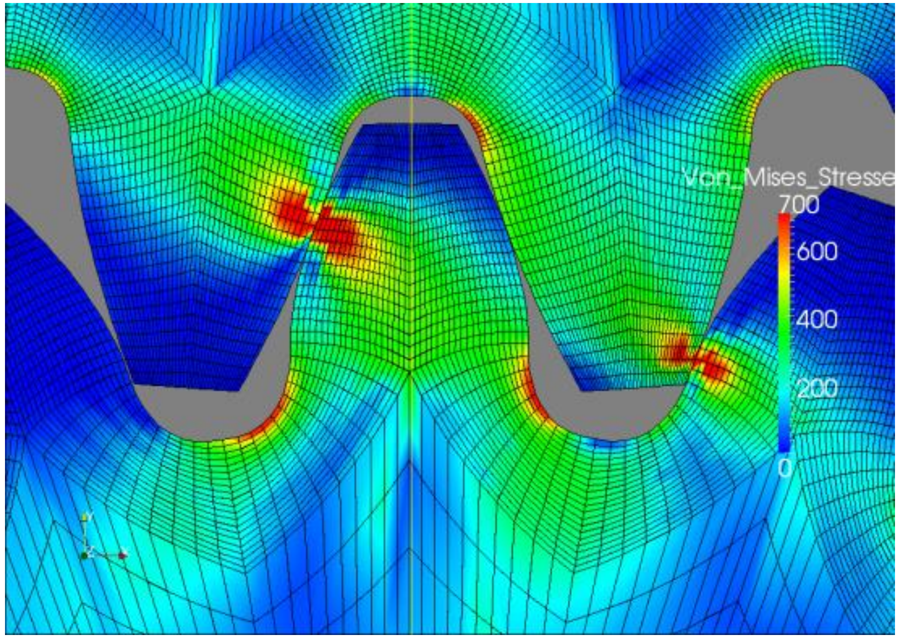
- PDEs describe lots of important physics, including
  - Solid mechanics (e.g. linear elasticity)
  - Fluid mechanics
  - Heat transfer
  - Electromagnetism

- **F**inite **E**lement **A**nalysis (FEA) involves the practical application of the finite element method to solve a real-world problem.

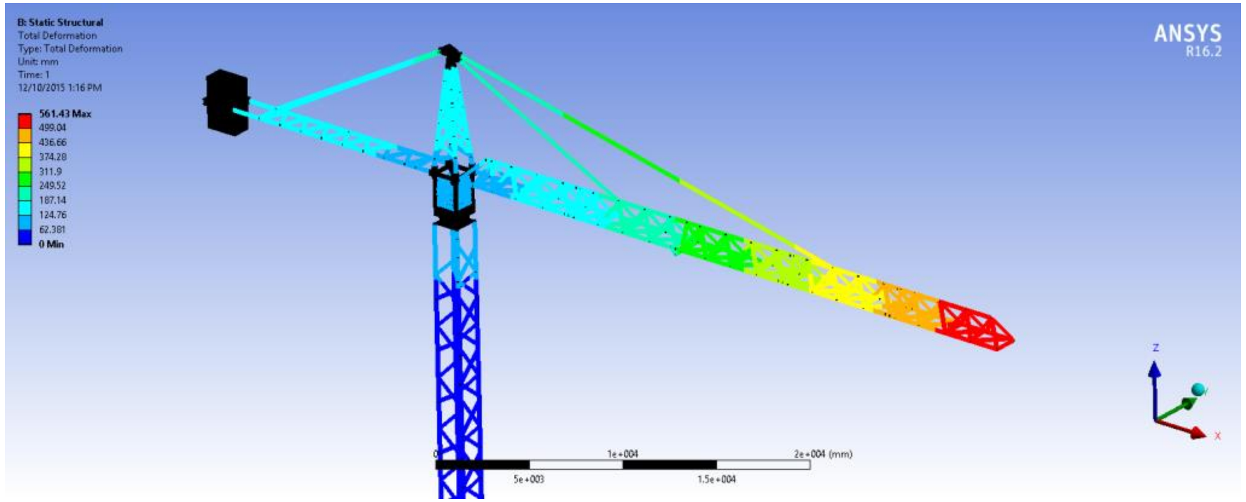
Why is FEA helpful?

- Most real-world problems are too complex to analyze by hand

- But, we'd like a way to estimate stresses and deformations in engineering components and structures. FEA, when used correctly, can help!



Gears in contact: what is the stress field in each of the gear teeth?
https://www.thetruthaboutcars.com/2014/07/saturation-dive-manual-transmission-gear-design/

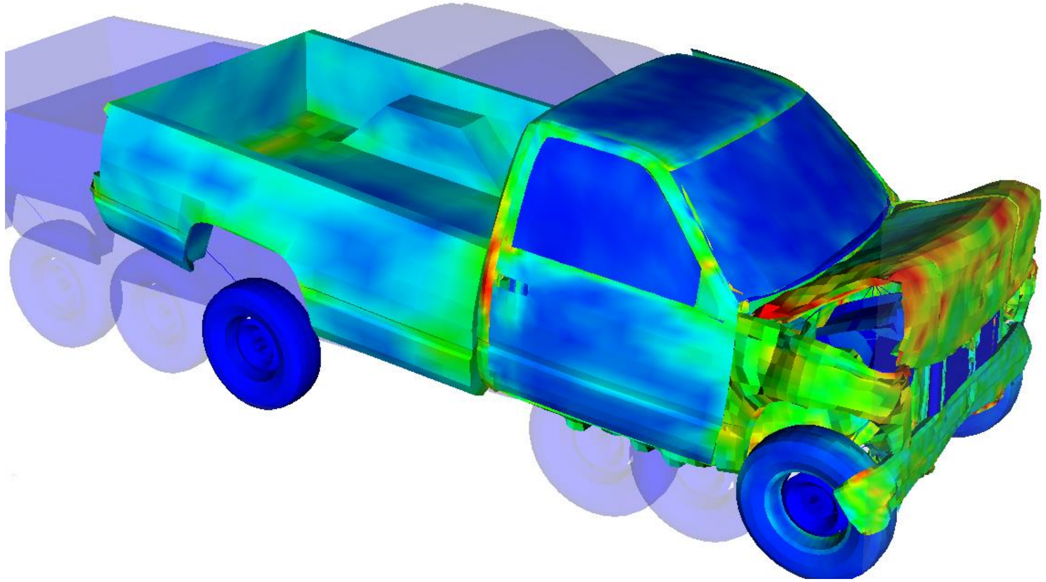


Large crane structure: how much deformation and stress does this structure experience under a given load?
https://www.finiteelementanalysis.com.au/featured/analysing-large-fabricated-structures/

Why is FEA helpful?

- Benefits of FEA:
  - ❑ Analyze complex geometries based on CAD models of parts to be designed/manufactured
  - ❑ Efficient for planning and prototyping
  - ❑ Quicker to test more variables under more situations
  - ❑ Is your test destructive? Destroy it in a simulation instead! It's cheaper
  - ❑ Bottom line: **saves time/money** when done well

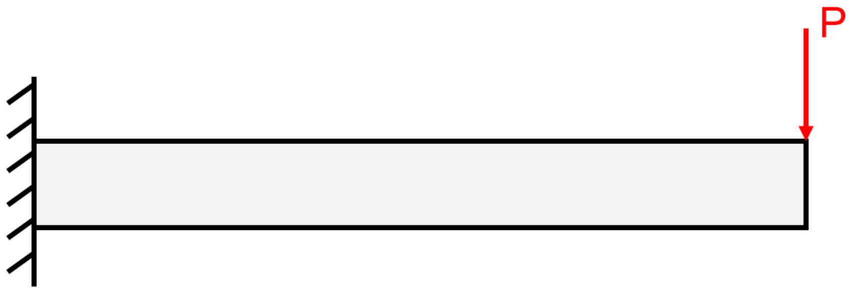
- Limitations include:
  - ❑ Models are always an **approximation** of reality
  - ❑ Complexity and learning curve of FEA software
  - ❑ Cost: very large models take time (days, or longer!) for the computer to solve



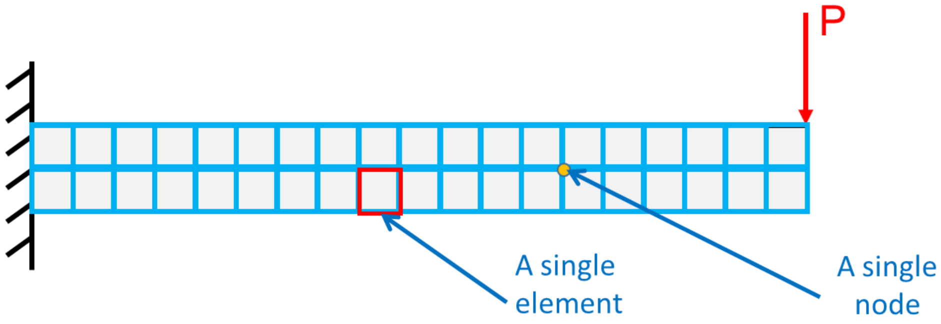
https://enteknograte.com/

FEM for Structural Analysis

Step 1: Define the problem: geometry, boundary conditions, material properties

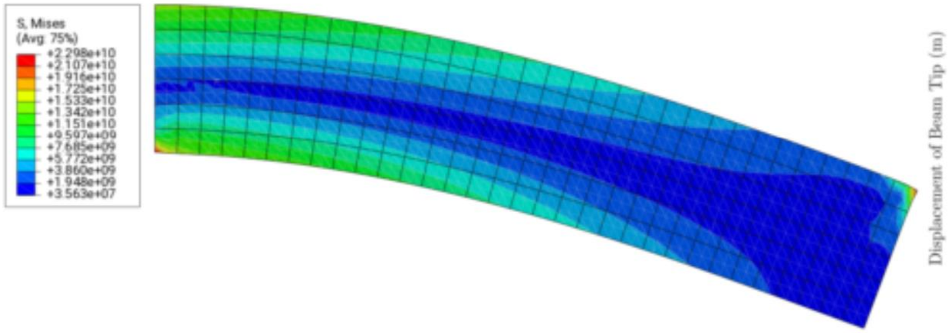


Step 2: Mesh (discretize) the domain into **nodes** and **elements**



Step 3: Solve the model to obtain nodal **displacements**

Step 4: Post-process the model to obtain element **stresses**

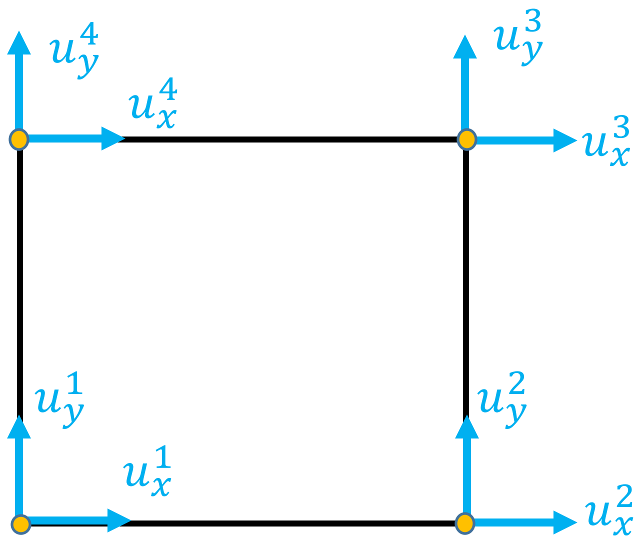


https://www.simscale.com/blog/2017/01/convergence-finite-element-analysis/

Anatomy of an Element

Below is an example of a 4-node quadrilateral element that can be used to model a 2D structural problem.

Each quadrilateral element has 4 nodes, and two **degrees of freedom** per node (u_x^i and u_y^i, the horizontal and vertical displacement of node i)



$$\boldsymbol{K^e u^e} = \boldsymbol{F^e} \Rightarrow \begin{pmatrix} K_{11} & \cdots & K_{18} \\ \vdots & \ddots & \vdots \\ K_{81} & \cdots & K_{88} \end{pmatrix} \begin{pmatrix} u_x^1 \\ u_y^1 \\ u_x^2 \\ u_y^2 \\ u_x^3 \\ u_y^3 \\ u_x^4 \\ u_y^4 \end{pmatrix} = \begin{pmatrix} F_x^1 \\ F_y^1 \\ F_x^2 \\ F_y^2 \\ F_x^3 \\ F_y^3 \\ F_x^4 \\ F_y^4 \end{pmatrix}$$

The element **stiffness matrix** and **load vector** relate the nodal displacements to forces applied on the nodes.

There are various techniques to derive the equations for the element stiffness and load vector for a given element, but all types of finite elements are defined by these two critical quantities

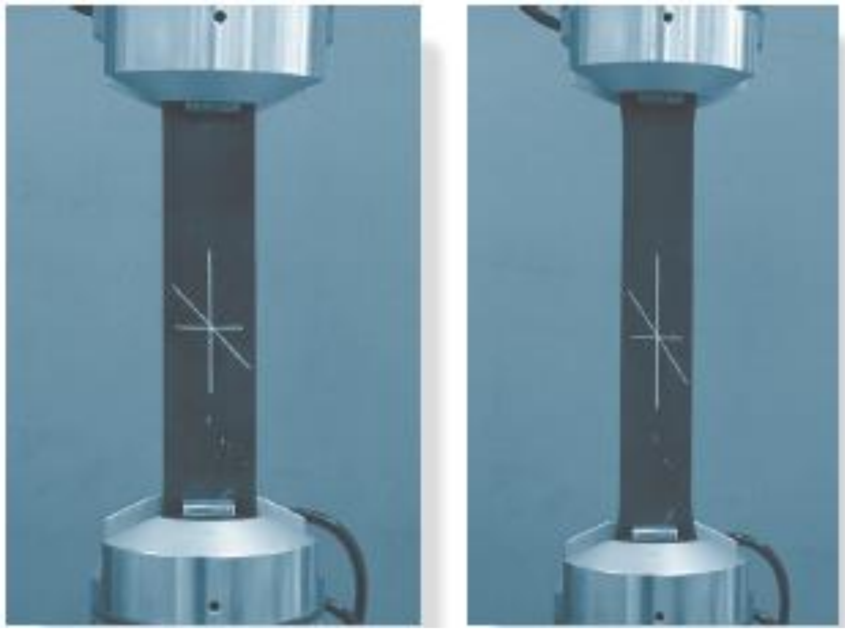
Material Properties

For **isotropic** materials, there are 3 key material properties to be aware of:

1. E= elastic modulus (Young's modulus): Describes the stiffness of a material under axial loading. Analagous to a spring constant for a linear spring
2. ν = Poisson's ratio: governs length change in the transverse direction when a structure is loaded axially
3. G = Shear modulus: like elastic modulus, but for shear stresses and shear deformation

Need only 2 constants to describe an isotropic material because $E = 2G(1+\nu)$. You should never define all 3 constants in FEA software (most software won't let you).

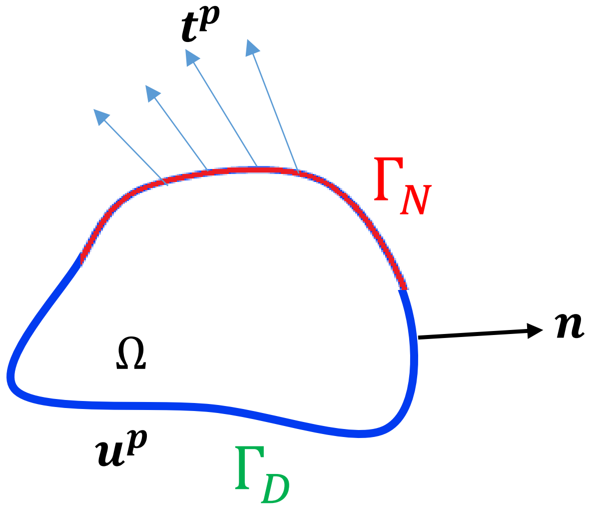
Isotropic are the simplest, but most common, materials you might encounter. Selection of an appropriate material model and constants is critical to performing a meaningful analysis.



Boundary Conditions

A key component of any structural analysis is specification of **boundary conditions (BCs).** BCs describe how the structure interacts with its surroundings. There are two general types to be aware of:

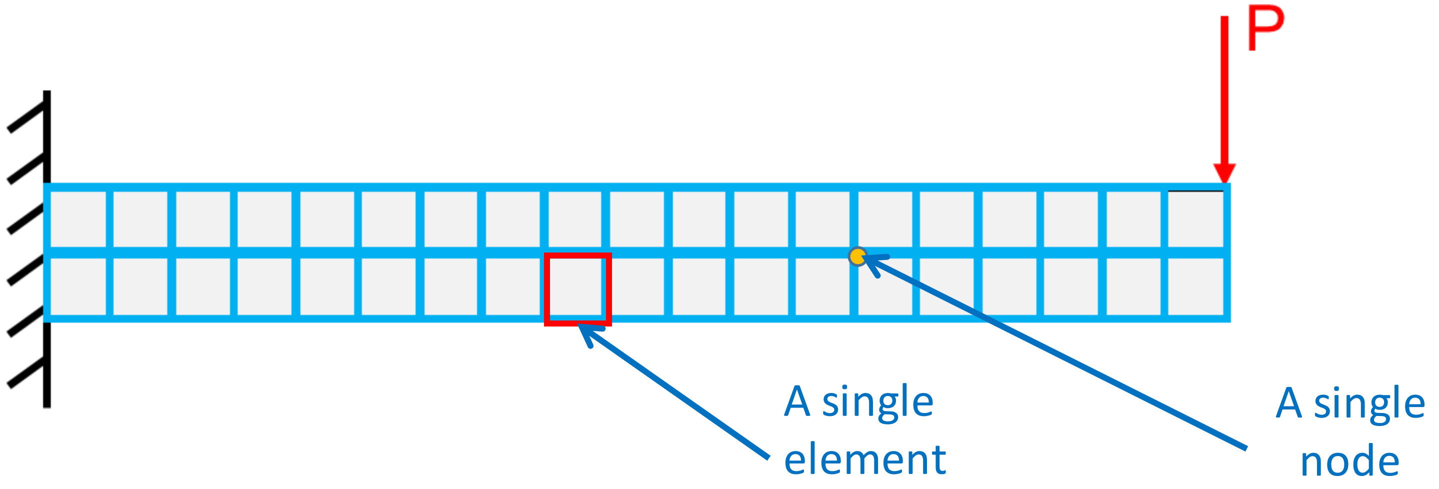
1. **Displacement BC (Γ_D):** Specify displacement on a portion of the boundary.
   - Often, $\boldsymbol{u}^p = \mathbf{0}$ (fixed boundary)
2. **Traction BC (Γ_N):** Traction is a force per unit area. It is represented by a vector.
   - Often simplified as a point load
   - Note: A "free" boundary (no loads, no constraints) is equivalent to prescribing a zero-traction condition



More complex boundary conditions (e.g. contact with another deformable body) are essentially special cases or combinations of the above conditions.

FEA Boundary Conditions: Example 1

The domain below has been broken into 38 total elements and 60 total nodes. The 60 total nodes represent $60 \times 2 = 120$ nodal displacement values in the model, 60 in the x direction and another 60 in the y direction



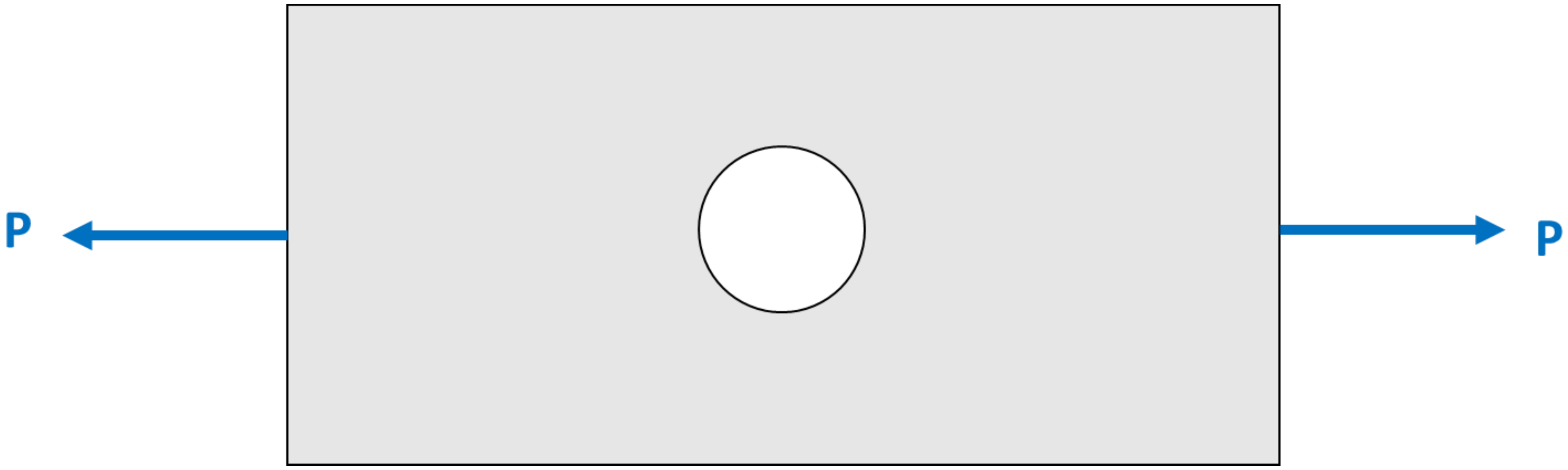
Some questions:

1. What are the boundary conditions used in this FEA model?
2. The number of **degrees of freedom,** n_{dof}, is the number of nodal displacement values that must be solved for. What is n_{dof} for this model?

FEA Boundary Conditions: Example 2

Consider a plate with a hole subjected to tensile loading. Sketch a model of the boundary conditions you could use to model this problem.

Hint: FEA Boundary conditions must restrain rigid body motions in order to find a unique solution to the displacement field. This means you **must** use displacement boundary conditions somewhere in the model to keep the body fixed in space.

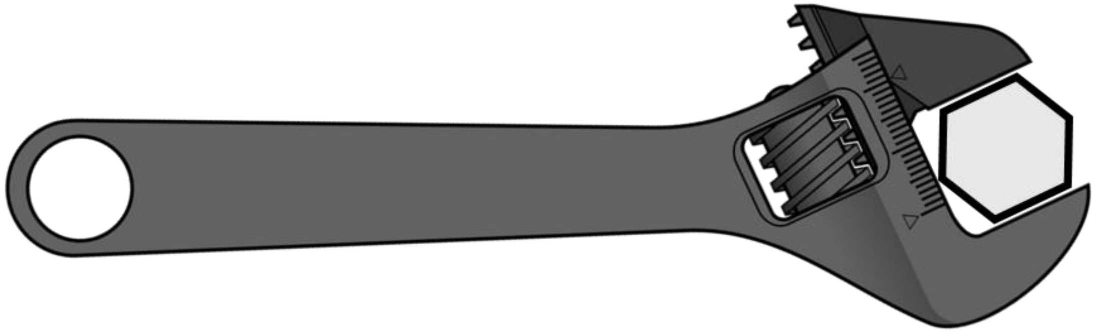


FEA Boundary Conditions: Example 3

Consider a wrench applying torque to a bolt head. Suppose we wish to use FEA to calculate the stresses in the wrench in this situation.

Sketch a model of the boundary conditions you could use to model this problem. Assume the user applies load along the handle, but the torque is low enough that the bolt doesn't begin to turn.

<u>**Hint**</u>: It's best not to include the bolt in the FEA model and mesh. How could you represent the presence of the bolt with boundary conditions?



Types of Elements

The most common types of finite elements are summarized in the figure below.

- Elements interpolate the displacement field between nodes.
- The displacement field is typically either modeled with a linear or quadratic polynomial.

What type of element should you use? Several factors

- **Cost**: linear elements are computationally cheaper/simpler to work with
- **Geometry**: Triangular elements easily mesh a domain; quadratic elements can have curved edges
- **Type of problem being solved**: E.g., linear triangles/tetrahedra don't capture bending deformation very accurately

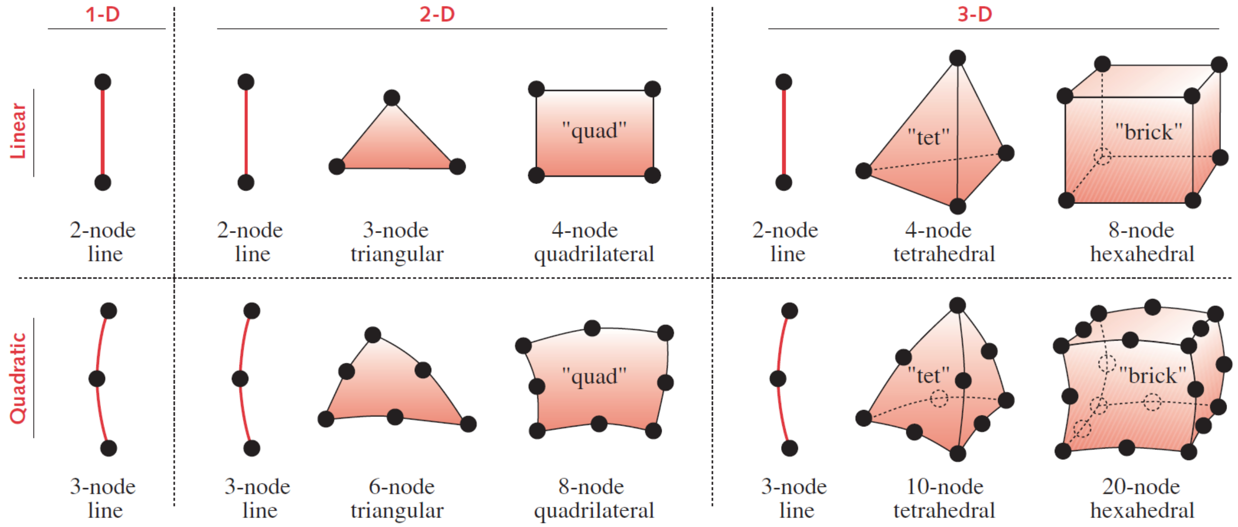
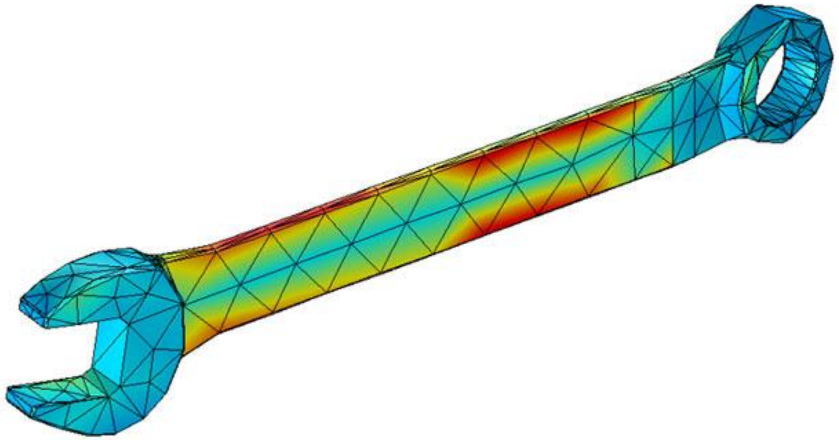


FIGURE 8-3

Some common finite elements



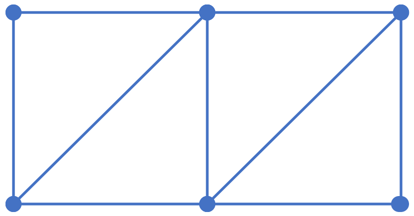
Influence of Mesh on FEA Solution

Remember that FEA results in an **approximation** of the exact solution to a PDE with specified boundary conditions.

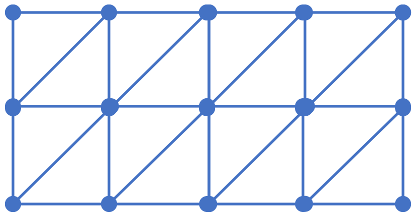
In general, **more nodes** and/or a **higher degree** of polynomial interpolation results in a **more accurate** FEA solution.

Starting from a given mesh, there are two ways to increase the accuracy of a solution:

1) Add more elements (**h-refinement**)
2) Use a higher-order polynomial for interpolation (**p-refinement**)

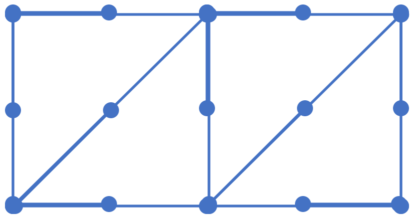


Original mesh
4 linear triangle elements
6 nodes



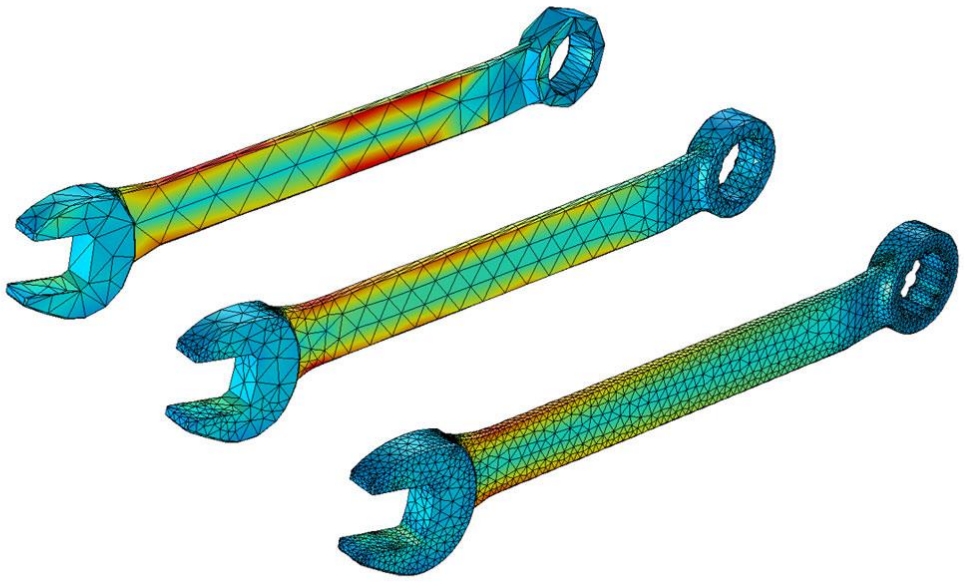
h-refinement
16 linear triangle elements
15 nodes
(**Option 1**)

Or

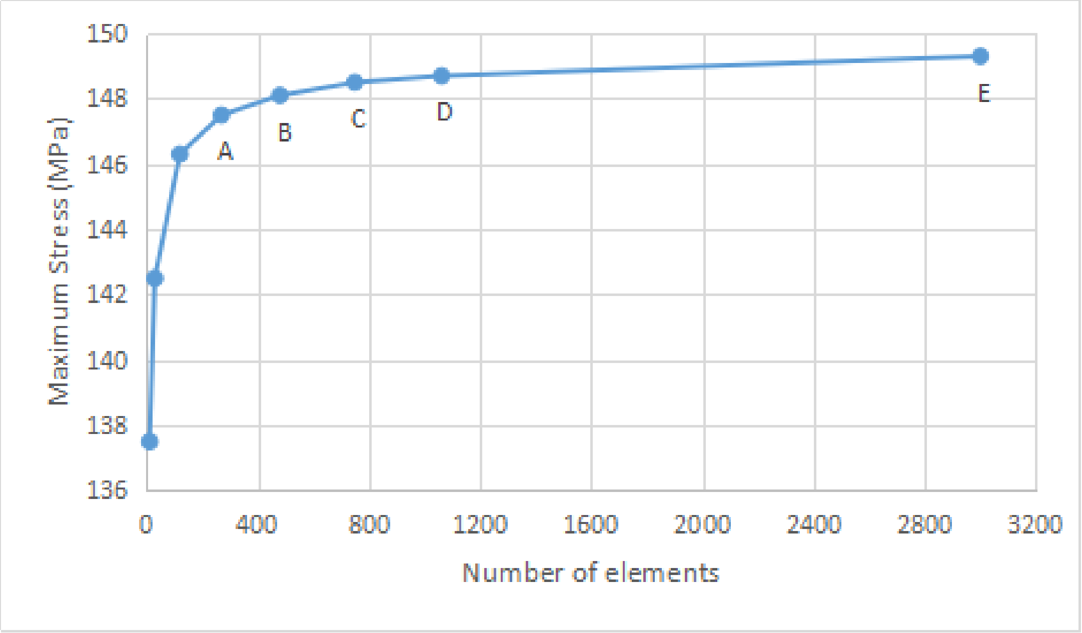


p-refinement
4 quadratic triangle elements
15 nodes
(**Option 2**)

FEA Solution Convergence



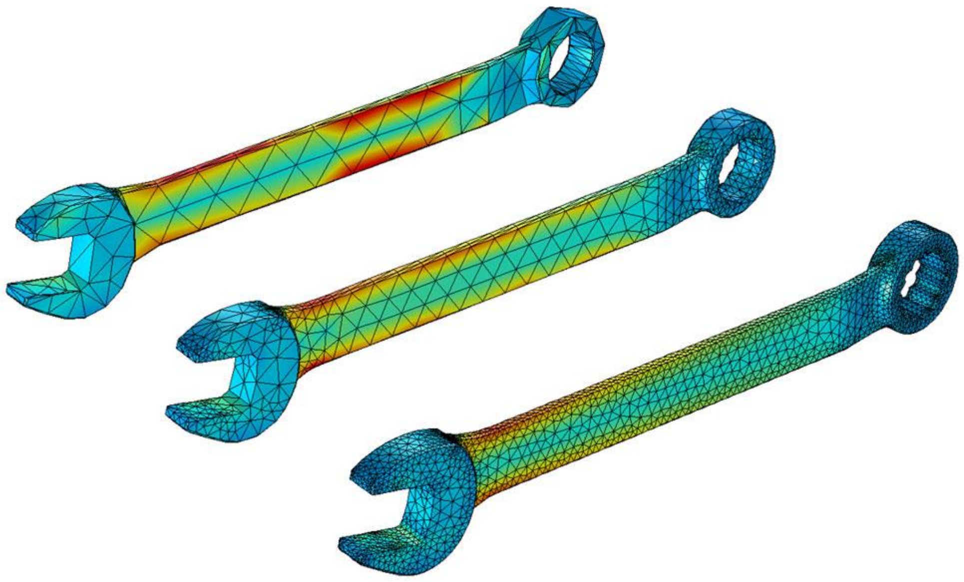
An example of h-refinement of an FEA model of a wrench.
https://www.comsol.com/multiphysics/mesh-refinement



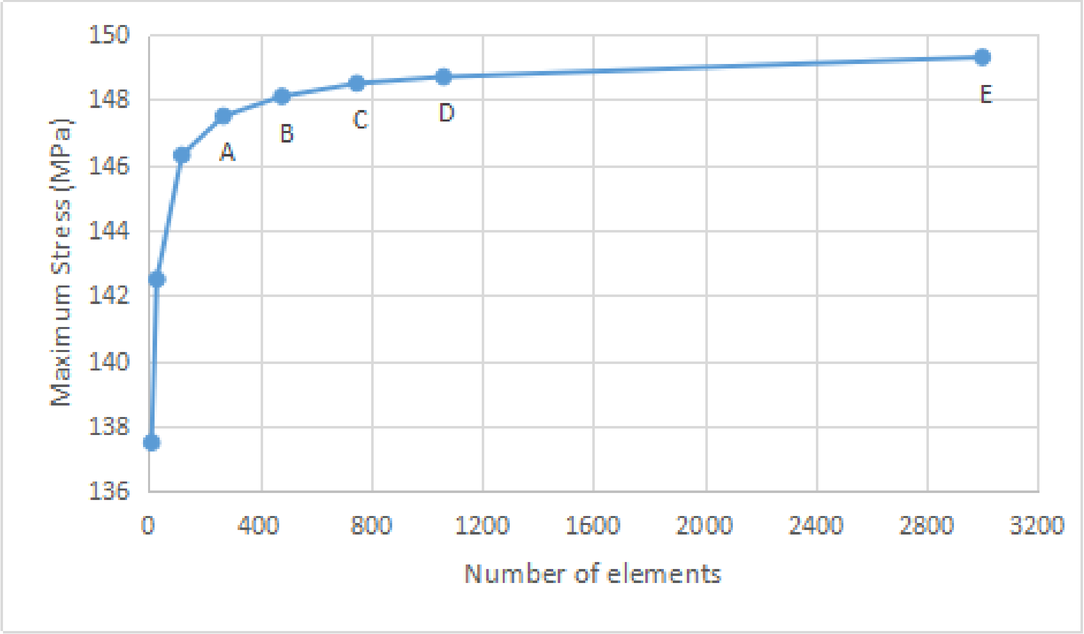
Example of a plot to detect solution convergence
https://caeuniversity.com/mesh-convergence-modern/

- When two successive refinements result in only a small difference in the FEA solution, we say the solution has **converged**
- Typically, you will need more elements where stress is changing quickly, or to adequately capture part geometry
- **You should <u>always</u> ensure your FEA model has converged!**

FEA Solution Convergence



An example of h-refinement of an FEA model of a wrench.
https://www.comsol.com/multiphysics/mesh-refinement



Example of a plot to detect solution convergence
https://caeuniversity.com/mesh-convergence-modern/

- For most FEA problems, **solving the linear system** of equations **Ku** = **F** consumes the **vast majority of computing time and power.**

- Because of the rapid scaling of the cost of solving a model, *it's best to use only as many elements/nodes as you truly need for a given model*

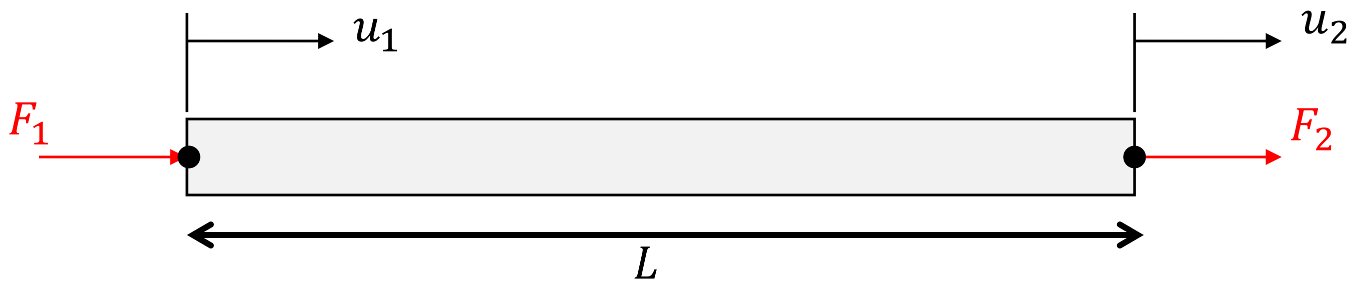
Model Validation

Model **validation** involves checking the results of your model against reality, e.g. documented experiments, performing your own tests or experiments, etc.

In our wrench example, if we observed the failure mode below, but the FEA model predicted the highest stresses to be in another part of the structure, it would warrant double-checking problem inputs like material properties, revisiting the boundary conditions to determine if they are adequate for the true loading scenario, etc.

FEA with Linear Rod Elements

The linear rod element is one of the simplest possible structural elements.



A linear rod has 2 nodes with nodal displacements u_1 and u_2 and nodal forces F_1 and F_2. The rod has constant cross-sectional area A and elastic modulus E, and the length of the rod is L. It can only take axial loads.

From equilibrium analysis, we know that $F_{int} = -F_1 = F_2 = \frac{EA}{L}(u_2 - u_1)$. Therefore, we can write two equations:

$$F_1 = \frac{EA}{L}(u_1 - u_2), \qquad F_2 = \frac{EA}{L}(-u_1 + u_2)$$

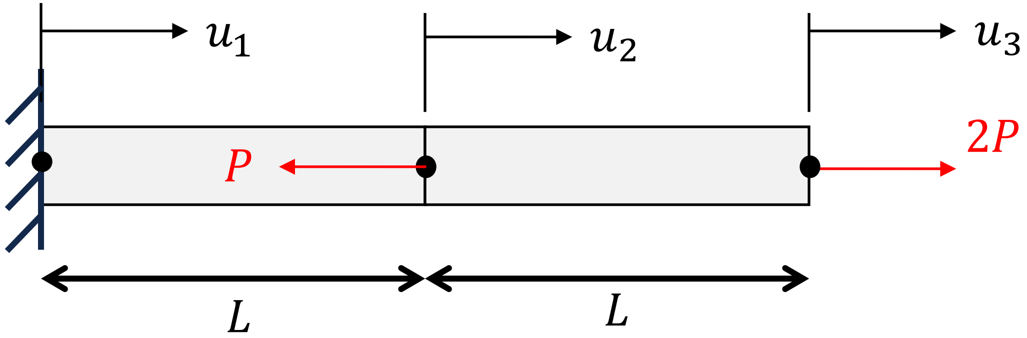
Or in matrix form:

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$$

This gives us the **stiffness matrix** and **load vector** for a 1D rod element: $\boldsymbol{K}^e = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$, and $\boldsymbol{F}^e = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$ (applied loads at the nodes)

FEA with Linear Rod Elements

Suppose we wish to find the total elongation of the structure below, which consists of two rod elements of the same length L, modulus E, and cross-sectional area A, connected in series. There are two elements and a total of 3 nodes



To find the stiffness matrix and load vector for this system, we can use **assembly**, which is a key feature of FEA solution strategies.

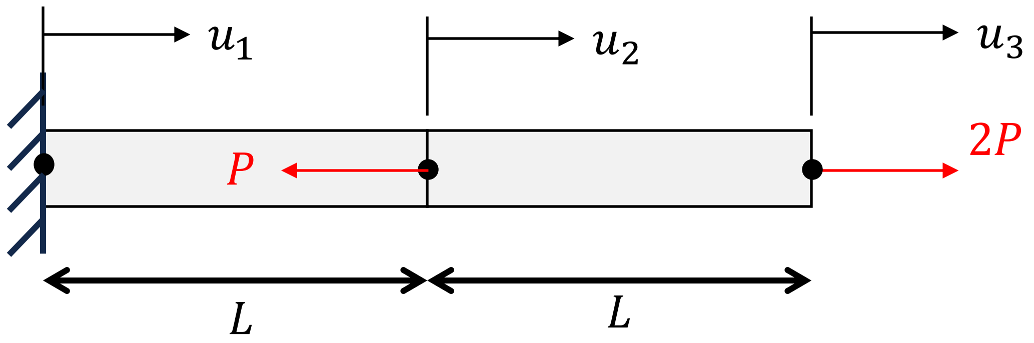
To assemble the global stiffness matrix, we compute each element stiffness and note its contribution to the global stiffness matrix, which is 3x3 in size due to the 3 total displacement DOF in the system:

$$\boldsymbol{K}_1^{\boldsymbol{e}} = \frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 & 0 \\ 0 & 0 & 0 \end{bmatrix}, \qquad \boldsymbol{K}_2^{\boldsymbol{e}} = \frac{EA}{L}\begin{bmatrix} 0 & 0 & 0 \\ 0 & 1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \rightarrow \boldsymbol{K} = \boldsymbol{K}_1^{\boldsymbol{e}} + \boldsymbol{K}_2^{\boldsymbol{e}} = \frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

$\boldsymbol{F}$ contains the loads applied to each node: (R is an unknown placeholder for the wall reaction force)

$$\boldsymbol{F} = \begin{bmatrix} R \\ -P \\ 2P \end{bmatrix}$$

FEA with Linear Rod Elements



Hence the global assembled system of equations is $\boldsymbol{Ku} = \boldsymbol{F}$:

$$\frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} R \\ -P \\ 2P \end{bmatrix}$$

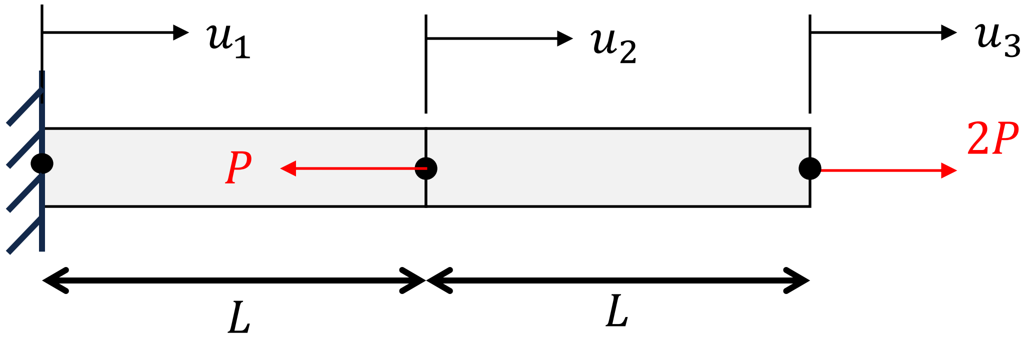
Because the rod is fixed at the wall, $u_1 = 0$, which allows us to modify the system of equations to read:

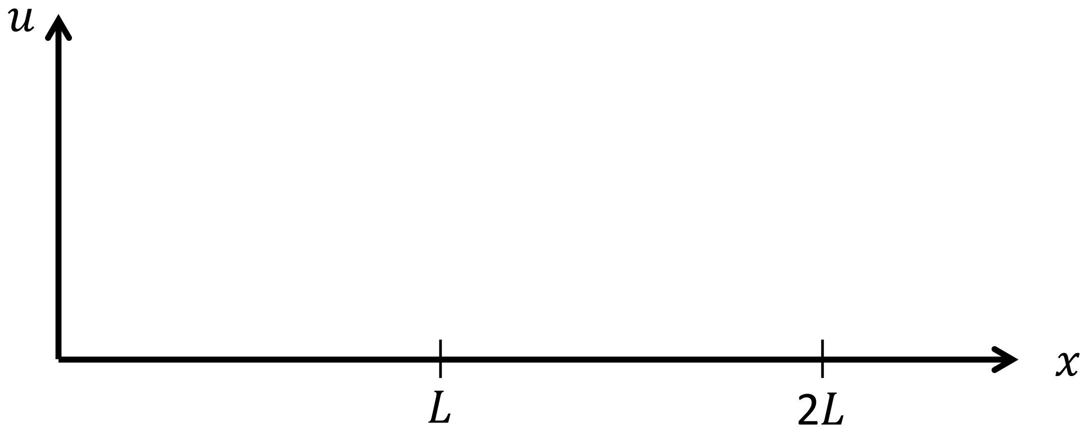
$$\frac{EA}{L}\begin{bmatrix} 2 & -1 \\ -1 & 1 \end{bmatrix}\begin{bmatrix} u_2 \\ u_3 \end{bmatrix} = \begin{bmatrix} -P \\ 2P \end{bmatrix}$$

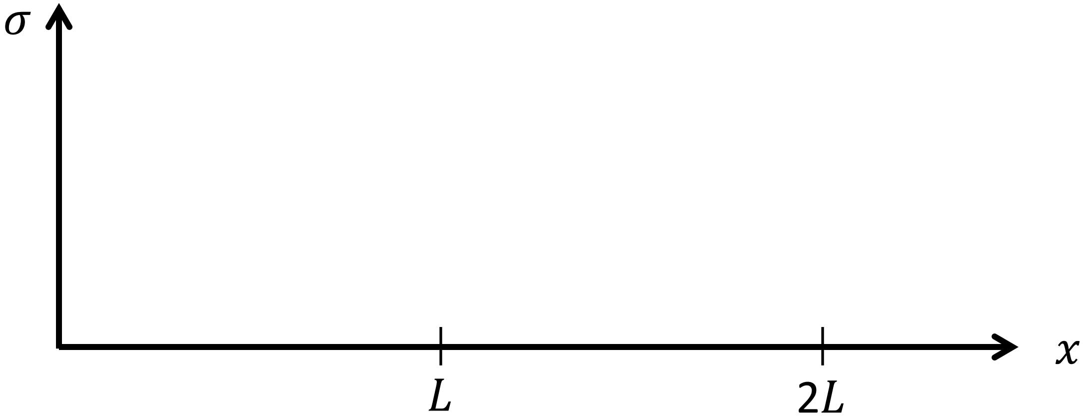
We can solve this to obtain $u_2 = \frac{PL}{EA}, u_3 = 3\frac{PL}{EA}$

We can revisit the first row of the original equation if we want to compute the wall reaction: $R = \frac{EA}{L}(u_1 - u_2) = -P$

FEA with Linear Rod Elements







In the linear rod element, the strain in each element is **constant**, and since $\sigma = E\epsilon$ for linear elasticity, so is the stress

Displacement is the integral of strain $\left(\epsilon = \frac{du}{dx} = \frac{\Delta u}{\Delta x}\right)$, so the displacement field is **linear** in each element.

Let's make plots of the FEA displacement field and stress field on the axes below.

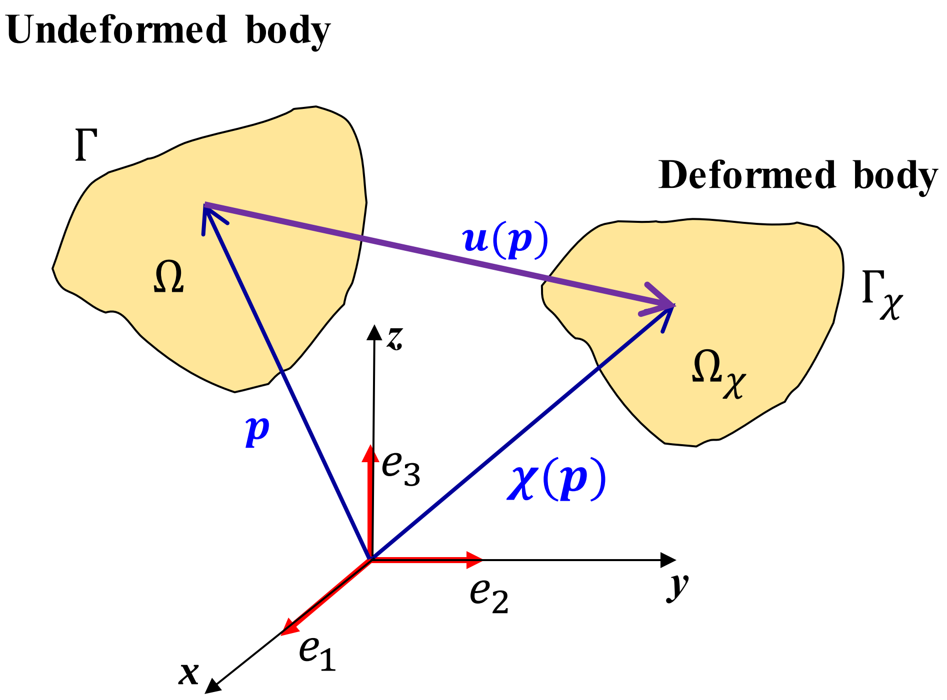
Recall the solution $(u_1, u_2, u_3) = \left(0, \frac{PL}{EA}, 3\frac{PL}{EA}\right)$ for this problem:

Appendix

The following slides give some more background and key equations from the theory of linear elasticity, which may be helpful in setting up FEA simulations and interpreting results.

This information can also be found in most introductory texts on solid mechanics or the finite element method.

Kinematics of a deforming body



Undeformed (or initial) configuration:

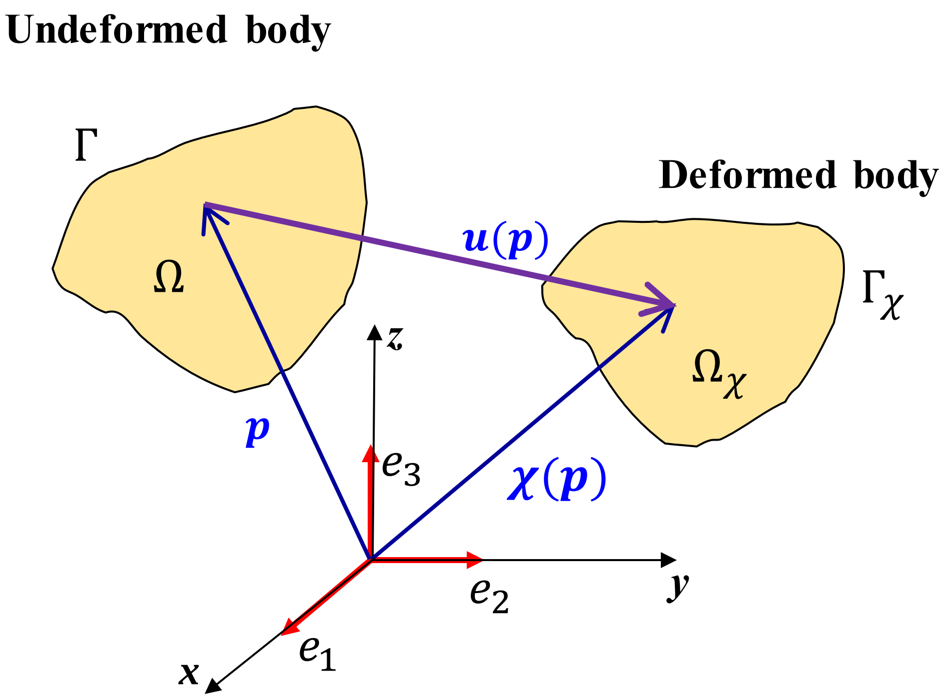
- Volume Ω
- Boundary $\Gamma = \Gamma_D \cup \Gamma_N$
- Γ_D: boundary where displacement is prescribed (essential BC)
- Γ_N: boundary where traction is prescribed (natural BC)
- Points in the body are defined by the **position vector** $\boldsymbol{p}$

Deformed configuration:
After forces are applied, the body assumes a new configuration defined by the **deformation** $\boldsymbol{\chi}(\boldsymbol{p})$

The **displacement** $\boldsymbol{u}(\boldsymbol{p})$ is defined such that $\boldsymbol{u}(\boldsymbol{p}) = \boldsymbol{\chi}(\boldsymbol{p}) - \boldsymbol{p}$

Note that so far no assumptions were made regarding the magnitude of the deformation

Kinematics for linear elasticity



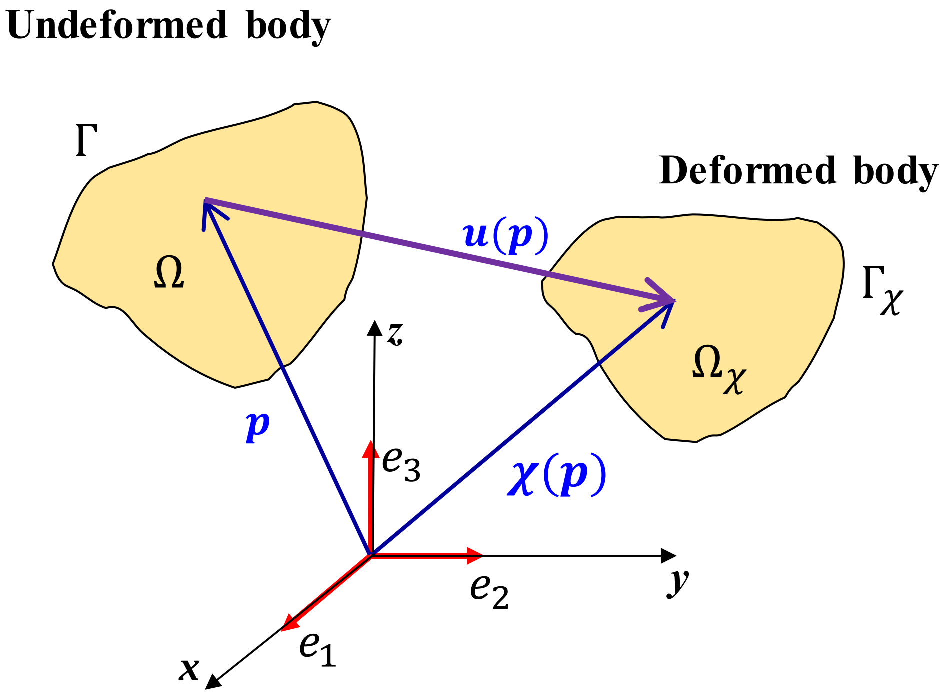
In linear elasticity, we assume the deformed configuration involves only small deformations (strains)

Therefore, we do not differentiate between the deformed and undeformed configuration. This simplifies many things:

- Strains and stresses are measured relative to the original geometry (i.e. engineering stress/strain is applicable)
- Applied loads don't change direction as a body deforms
- **Linear** constitutive law linking stress and strain

The displacement field $\boldsymbol{u}(\boldsymbol{p}) = \boldsymbol{\chi}(\boldsymbol{p}) - \boldsymbol{p}$ might have large magnitudes, but its gradient $\nabla \boldsymbol{u}$ is "small" in this treatment.

Kinematics for linear elasticity



These lectures notes will use (x, y, z) interchangeably with (x_1, x_2, x_3) to express the Cartesian coordinate variables

x_i is the coordinate associated with the unit vector $\boldsymbol{e}_\mathbf{i}$

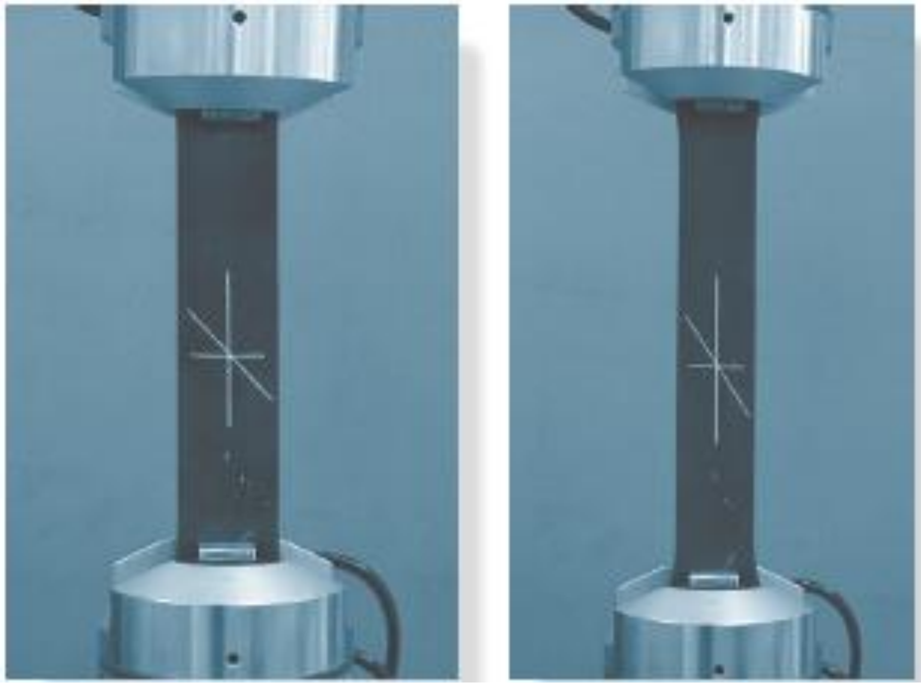
Using numbers allows for some more convenient notation, for example

$$(\nabla \boldsymbol{u})_{ij} = \frac{\partial u_i}{\partial x_j}$$

Clearly communicates the components of the gradient of the vector $\boldsymbol{u}$

Small-strain (linear) elasticity

Engineering normal strain: length change



Rubber membrane subject to tension

Normal strains characterize length change along a specific direction. There are 3 normal strain components:

$$\epsilon_{11} = \frac{\partial u_1}{\partial x_1}, \qquad \epsilon_{22} = \frac{\partial u_2}{\partial x_2}, \qquad \epsilon_{33} = \frac{\partial u_3}{\partial x_3}$$

For uniaxial stretching of a body with constant cross-sectional area (constant stress), we can write

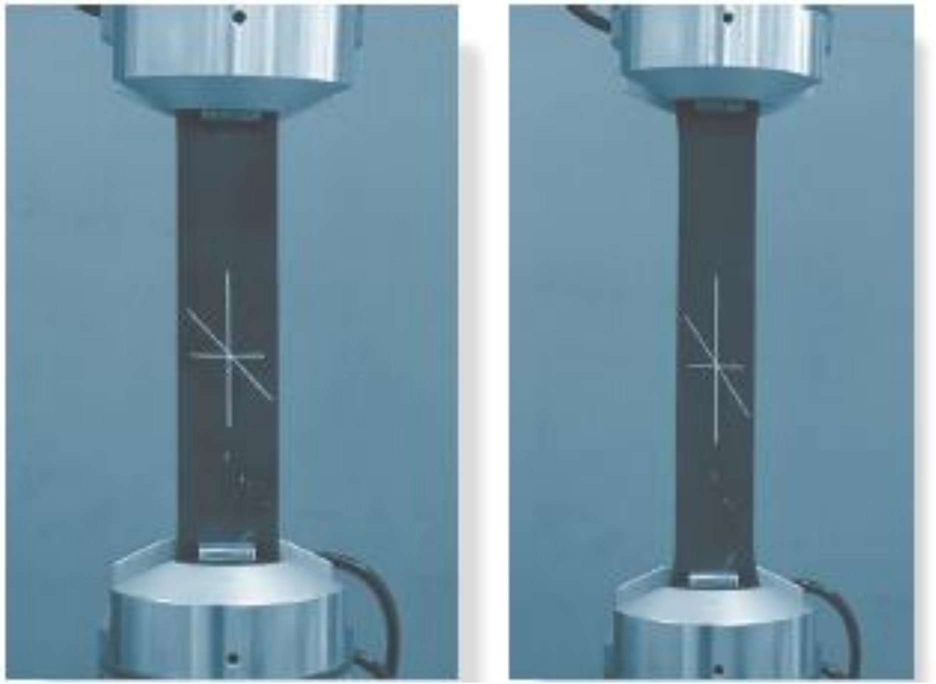
$$\epsilon = \frac{\Delta L}{L_0}$$

ΔL = change in length
L_0 = original length

Normal strains are positive when the material length is increased, and negative when the material length is decreased

Small-strain (linear) elasticity

Engineering shear strain: angle change



Rubber membrane subject to tension

Shear strain measures the angle change between two material line segments that were originally at 90 degrees to one another.

For example, the angle change between two material line segments originally aligned with the $x_1 - x_2$ axes can be shown to be

$$\gamma_{12} = \frac{\partial u_1}{\partial x_2} + \frac{\partial u_2}{\partial x_1}$$

Shear strains are symmetric in their indices, i.e. $\gamma_{12} = \gamma_{21}$. Therefore there are only 3 unique shear strains we need to be concerned with: $\gamma_{12}, \gamma_{23}, \gamma_{31}$

Shear strains are positive if the angle decreases (closes) and negative of the angle increases (opens)

State of stress in a body

The state of stress at a point in a solid is fully described by the three normal stresses σ_x, σ_y, and σ_z and three shear stresses τ_{xy}, τ_{yz}, and τ_{xz}; We only need three shear stress components because they are symmetric in their components as a requirement for moment equilibrium in the stress element.

→ $\tau_{xy} = \tau_{yx}$

→ $\tau_{yz} = \tau_{zy}$

→ $\tau_{xz} = \tau_{zx}$

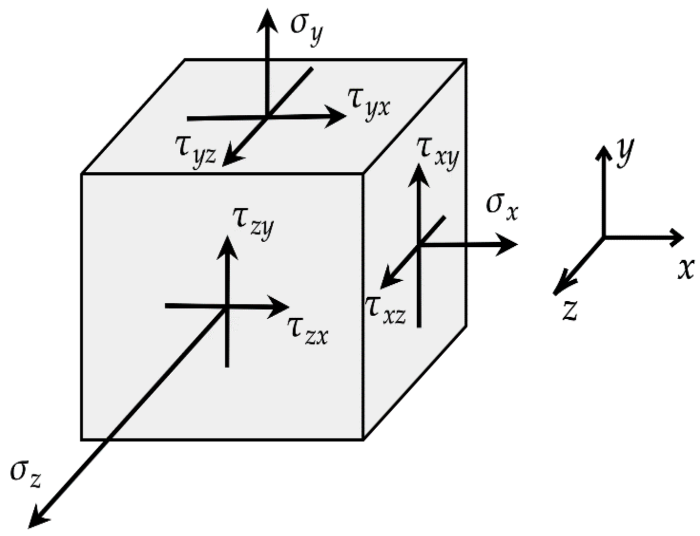
Using index notation, we instead write shear stresses using σ_{ij}, for example, $\tau_{xy} = \sigma_{12}$ in this notation

In index, notation, we write the stress tensor as

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \sigma_{12} & \sigma_{13} \\ \sigma_{21} & \sigma_{22} & \sigma_{23} \\ \sigma_{31} & \sigma_{32} & \sigma_{33} \end{bmatrix}$$

This is equivalent to writing, for a typical x-y-z system:

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



Stress-strain Relations

For **isotropic** materials, there are 3 material properties to be aware of:

1. E = elastic modulus (Young's modulus)
2. ν = Poisson's ratio
3. G = Shear modulus

Need **only 2 constants** to describe an isotropic material because $E = 2G(1+\nu)$

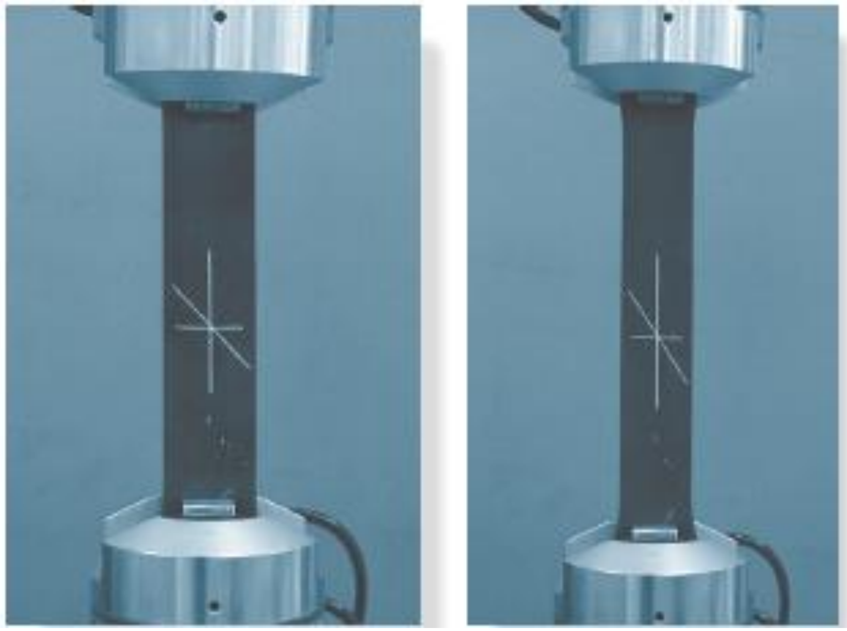
Axial loading:

Longitudinal stress-strain relation: $\sigma = E\epsilon$

Transverse strain: $\epsilon_{trans} = -\nu\epsilon_{long}$

If there are shear stresses:

$$\tau_{ij} = G\gamma_{ij} = 2G\epsilon_{ij}$$



Stress-strain relations (3D Hooke's law)

In general, all six stress and strain components may be non-zero due to a complex loading of a fully 3D structure. For **any** linearly elastic material, the relationship can be written as $\{\boldsymbol{\sigma}\} = [\boldsymbol{C}]\{\boldsymbol{\epsilon}\}$. The formulae below are for **isotropic** materials.

$$\begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1-2\nu}{2} \end{bmatrix} \begin{Bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ 2\epsilon_{12} \\ 2\epsilon_{13} \\ 2\epsilon_{23} \end{Bmatrix}$$

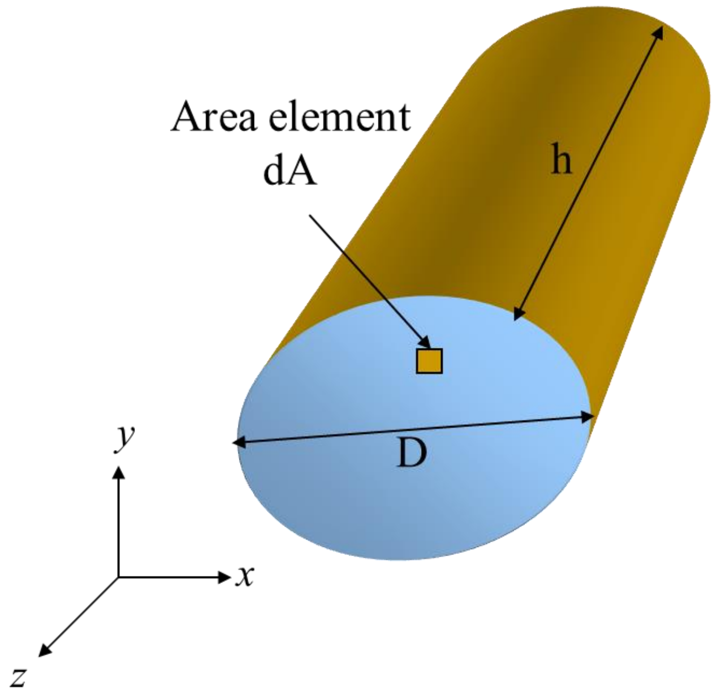
We can also invert the relationship to write $\{\boldsymbol{\epsilon}\} = [\boldsymbol{C}]^{-1}\{\boldsymbol{\sigma}\}$

$$\begin{Bmatrix} \epsilon_{11} \\ \epsilon_{22} \\ \epsilon_{33} \\ 2\epsilon_{12} \\ 2\epsilon_{13} \\ 2\epsilon_{23} \end{Bmatrix} = \frac{1}{E} \begin{bmatrix} 1 & -\nu & -\nu & 0 & 0 & 0 \\ -\nu & 1 & -\nu & 0 & 0 & 0 \\ -\nu & -\nu & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 2+2\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 2+2\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 2+2\nu \end{bmatrix} \begin{Bmatrix} \sigma_{11} \\ \sigma_{22} \\ \sigma_{33} \\ \sigma_{12} \\ \sigma_{13} \\ \sigma_{23} \end{Bmatrix}$$

Plane Strain

- In the plane strain case, only in-plane strain components are non-zero:

$$\boldsymbol{\epsilon} = \begin{bmatrix} \epsilon_{11} & \epsilon_{12} & 0 \\ \epsilon_{12} & \epsilon_{22} & 0 \\ 0 & 0 & 0 \end{bmatrix} \qquad \boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \sigma_{12} & 0 \\ \sigma_{21} & \sigma_{22} & 0 \\ 0 & 0 & \sigma_{33} \end{bmatrix}$$

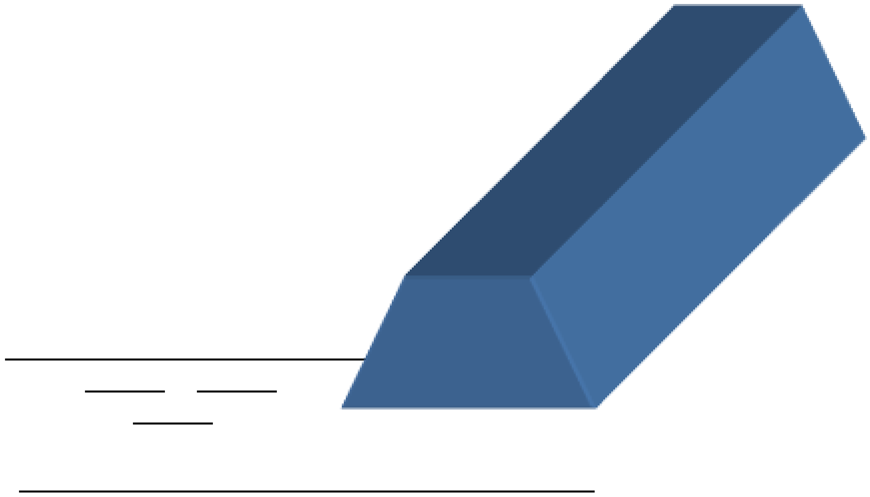


Assumptions:

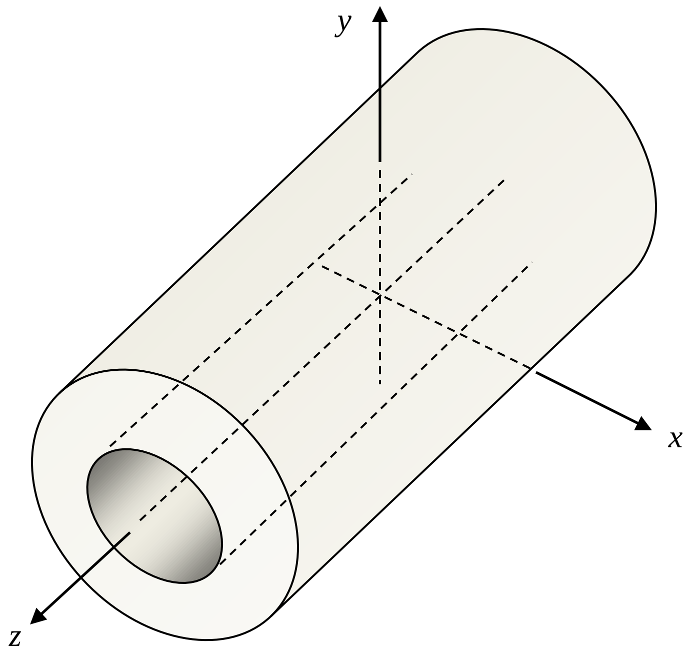
1. $h \gg D$
2. No body forces or tractions in the out-of-plane (z) direction
3. Applied tractions do not vary in the out-of-plane (z) direction

Plane Strain

Examples of plane strain problems



Dam



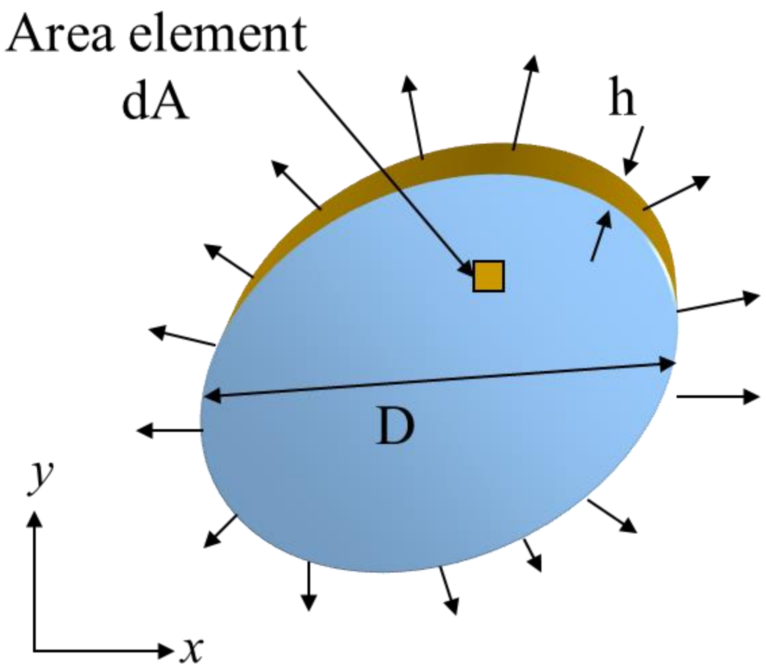
Long cylinders under uniform loading

Plane Stress

- In the plane stress case, only in-plane stress components are non-zero:

$$\boldsymbol{\sigma} = \begin{bmatrix} \sigma_{11} & \sigma_{12} & 0 \\ \sigma_{21} & \sigma_{22} & 0 \\ 0 & 0 & 0 \end{bmatrix}$$

$$\boldsymbol{\epsilon} = \begin{bmatrix} \epsilon_{11} & \epsilon_{12} & 0 \\ \epsilon_{21} & \epsilon_{22} & 0 \\ 0 & 0 & \epsilon_{33} \end{bmatrix}$$

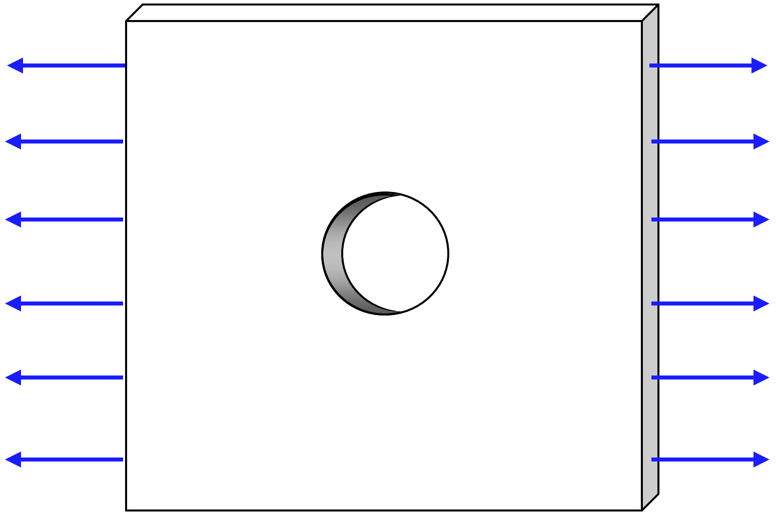


Assumptions:

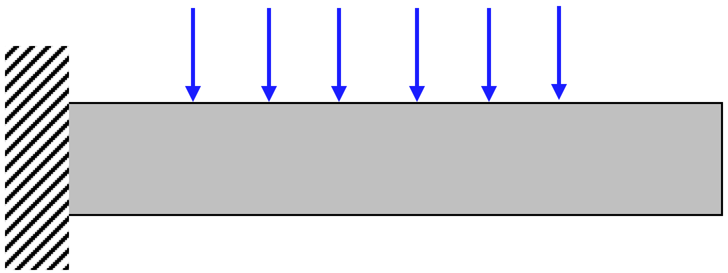
1. h<<D
2. Out-of-plane (z) surfaces are traction-free
3. No loads applied in the z direction

Plane Stress

Examples of plane stress problems



Thin plate with central hole



Thin cantilever plate

Post-processing

- **Principal stresses:** max/min normal stresses on any plane
- **Von Mises stress:** Useful for evaluating safety factor on ductile materials; plastic yield is predicted when the von Mises stress exceeds the yield strength of the material

In two dimensions:

$$\sigma_{p1,p2} = \frac{\sigma_{11} + \sigma_{22}}{2} \pm \sqrt{\left(\frac{\sigma_{11} - \sigma_{22}}{2}\right)^2 + (\sigma_{12})^2}$$

Plane Stress: $\sigma_{p3} = 0$

Plane Strain: $\sigma_{p3} = \sigma_{33} = \dfrac{E\,\nu}{(1+\nu)(1-2\nu)}(\epsilon_{11} + \epsilon_{22})$

Von Mises Stress

$$\sigma_{VM} = \sqrt{\frac{1}{2}\left[(\sigma_{p1} - \sigma_{p2})^2 + (\sigma_{p1} - \sigma_{p3})^2 + (\sigma_{p3} - \sigma_{p2})^2\right]}$$