Lecture

Tuesday, November 7, 2023 8:43 AM

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?

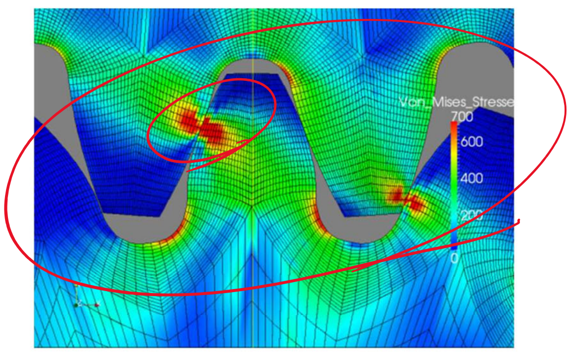
- The **F**inite **E**lement **M**ethod (FEM) is a mathematical framework that can be used to find approximate solutions to **p**artial **d**ifferential **e**quations (PDEs)
- PDEs describe lots of important physics, including
  - Solid mechanics (e.g. linear elasticity)
  - Fluid mechanics
  - Heat transfer
  - Electromagnetism
- **F**inite **E**lement **A**nalysis (FEA) involves the practical application of the finite element method to solve a real-world problem.

Page 3

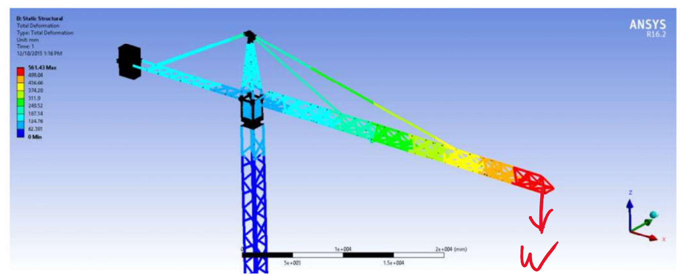
Tuesday, November 7, 2023 8:43 AM

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/



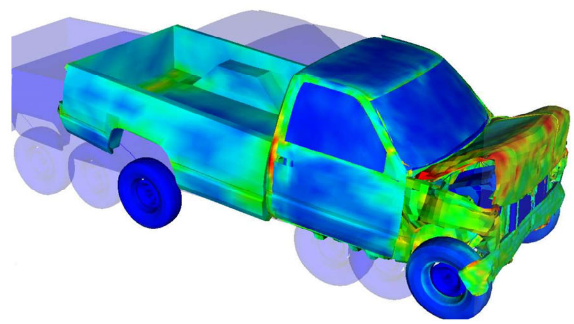
Large crane structure: how much deformation and stress does this structure experience under a given load?
https://www.finiteelementanalysis.com.au/featured/analysing-large-fabricated-structures/

Tuesday, November 7, 2023 8:43 AM

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/

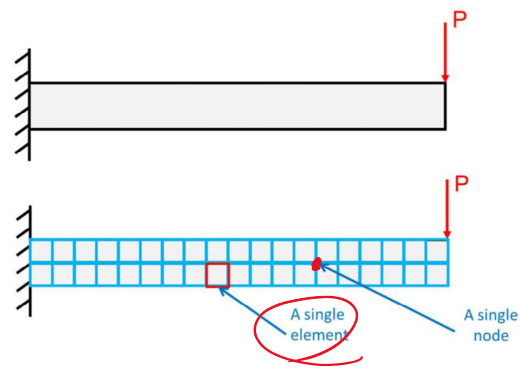
Page 5

Tuesday, November 7, 2023 8:43 AM

FEM for Structural Analysis

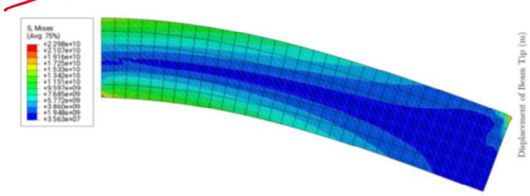
Step 1: Define the problem: geometry, boundary conditions, material properties

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/

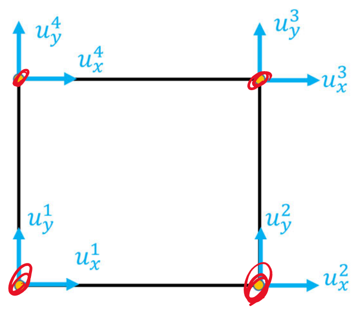
Page 6

Tuesday, November 7, 2023 8:43 AM

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)



stiffness F^e

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

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

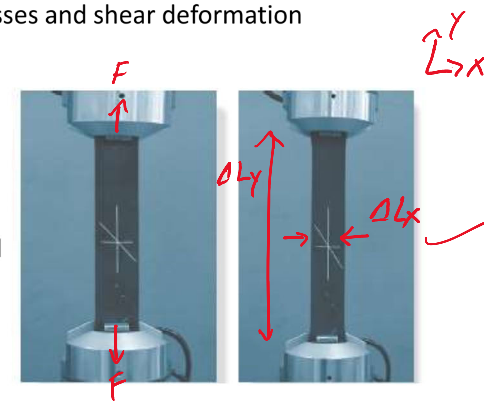
For **isotropic** materials, there are 3 key material properties to be aware of:

like spring const → τ $\sigma = E\varepsilon$

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).

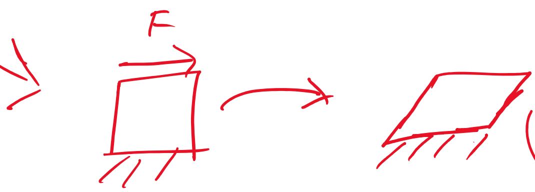
Isotropic are the simplest, but most common, materials you might encounter. Selection of an appropriate material model and constants is critical to performing a meaningful analysis.



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

$\varepsilon_{trans} = -\nu\,\varepsilon_{long}$

ε_x ε_y



$\tau = G\gamma$

shear stress

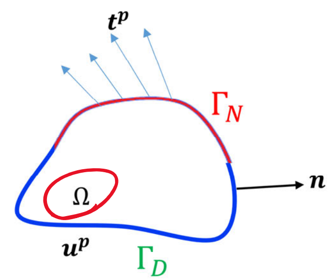
shear modulus

shear strain

Tuesday, November 7, 2023 8:43 AM

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, $\boldsymbol{u}^p = \boldsymbol{0}$ (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.

Tuesday, November 7, 2023 8:43 AM

FEA Boundary Conditions: Example 1

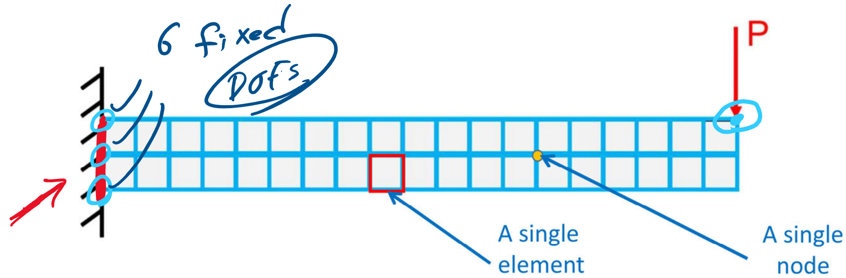
The domain below has been broken into 38 total elements and 60 total nodes. The 60 total nodes represent $60 \times 2 = 120$ nodal displacement values in the model, 60 in the x direction and another 60 in the y direction

$\underline{u} = \begin{bmatrix} u_x \\ u_y \end{bmatrix}$

6 fixed DOFs

$\underline{u} = \underline{0}$

$u_x, u_y = 0$



Some questions:

1. What are the boundary conditions used in this FEA model? → $\underline{u}$, P
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?

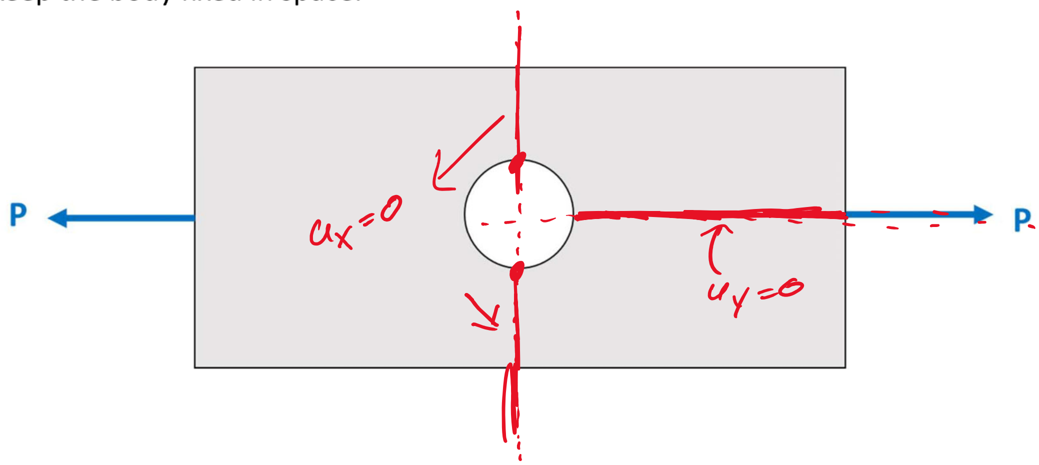
$n_{dof} = 120 - 6 = \boxed{114}$

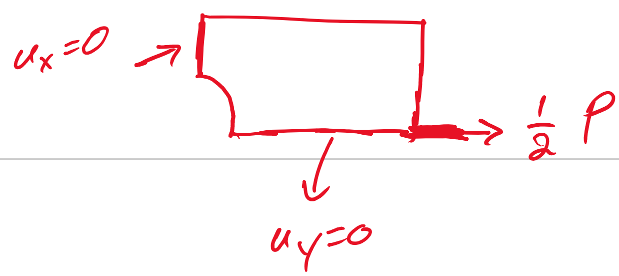
Mechanical Science & Engineering

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.



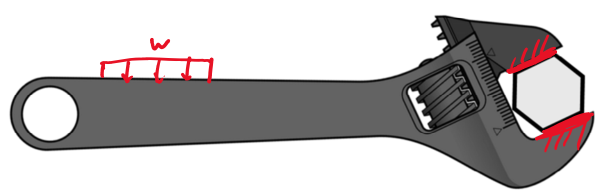


FEA Boundary Conditions: Example 3

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.

Hint: 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

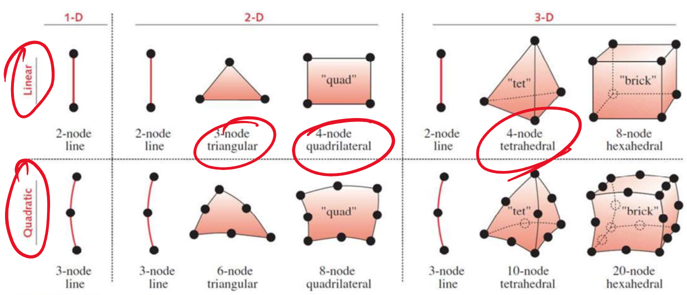
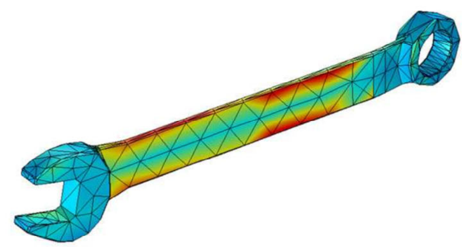


FIGURE 8-3

Some common finite elements



Page 13

Tuesday, November 7, 2023 8:43 AM

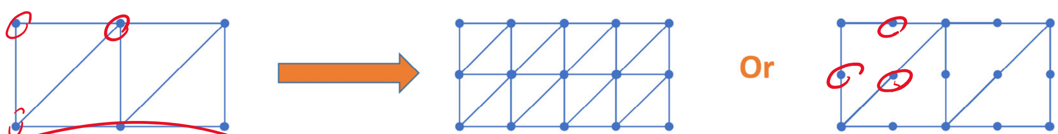
Influence of Mesh on FEA Solution

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

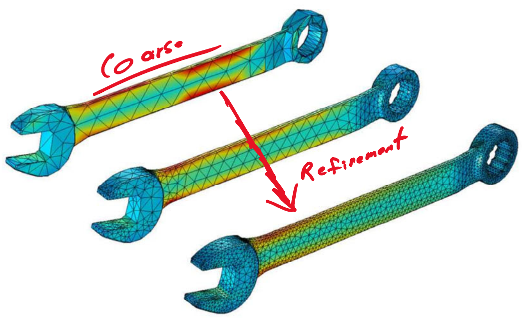
h-refinement
16 linear triangle elements
15 nodes
(**Option 1**)

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

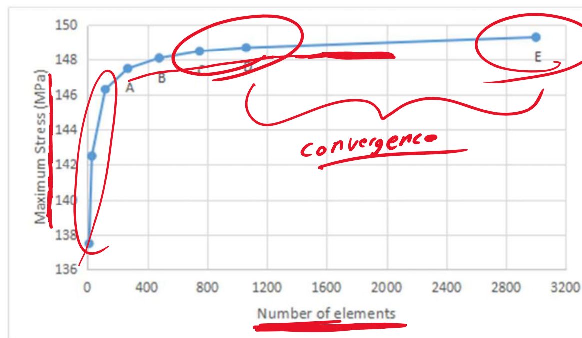
Page 14

Tuesday, November 7, 2023 8:43 AM

FEA Solution Convergence



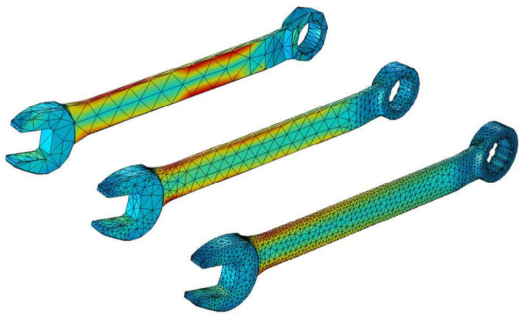
An example of h-refinement of an FEA model of a wrench.
https://www.comsol.com/multiphysics/mesh-refinement



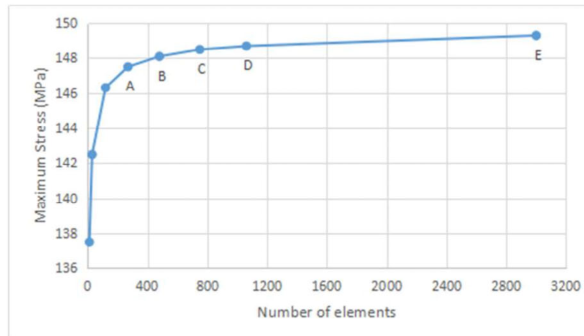
Example of a plot to detect solution convergence
https://caeuniversity.com/mesh-convergence-modern/

- 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*

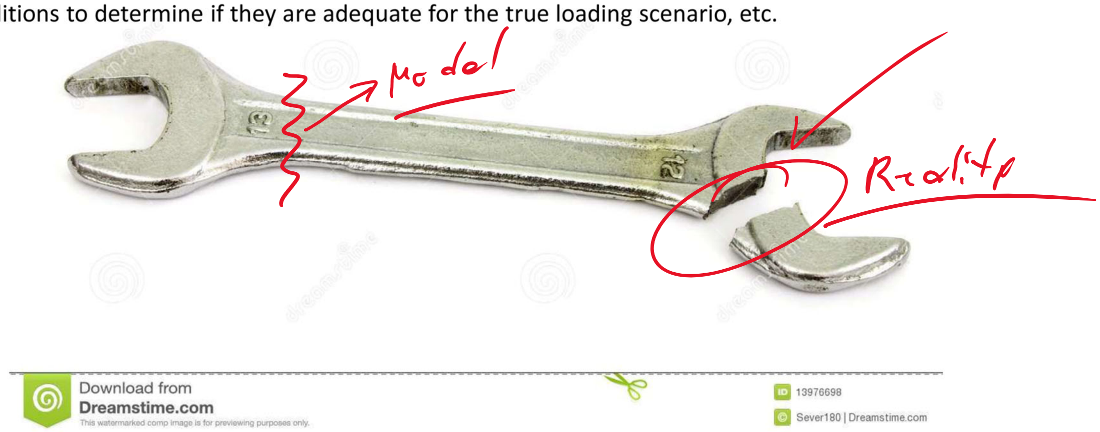
Page 16

Tuesday, November 7, 2023 8:43 AM

Model Validation

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

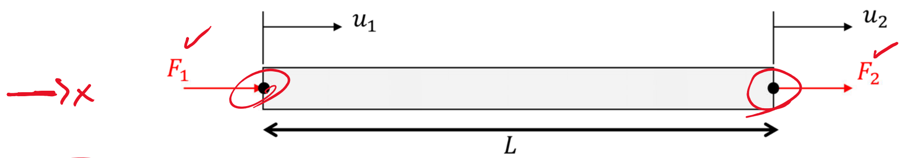
In our wrench example, if we observed the failure mode below, but the FEA model predicted the highest stresses to be in another part of the structure, it would warrant double-checking problem inputs like material properties, revisiting the boundary conditions to determine if they are adequate for the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



Tuesday, November 7, 2023 8:43 AM

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 $F_{int} = -F_1 = F_2 = \frac{EA}{L}\underbrace{(u_2 - u_1)}_{\Delta u}$. Therefore, we can write two equations:

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

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: $\boxed{\boldsymbol{K}^e = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}}$ and $\boldsymbol{F}^e = \begin{bmatrix} F_1 \\ F_2 \end{bmatrix}$ (applied loads at the nodes)

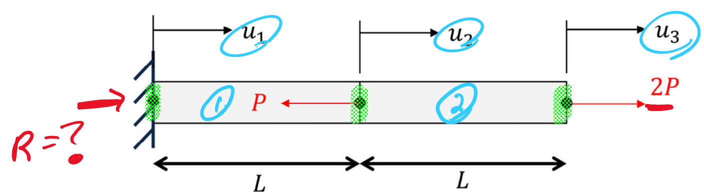
stiffness Matrix

Load vector

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

$$K^e = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$



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:

$$\boldsymbol{K}_1^e = \frac{EA}{L}\begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 1 & -1 & 0 \\ -1 & 1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \\ 3 \end{array}, \quad \boldsymbol{K}_2^e = \frac{EA}{L}\begin{array}{c} \begin{array}{ccc} 1 & 2 & 3 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 \\ 0 & 1 & -1 \\ 0 & -1 & 1 \end{bmatrix} \end{array} \begin{array}{c} 1 \\ 2 \\ 3 \end{array} \rightarrow \quad \boldsymbol{K} = \underbrace{\boldsymbol{K}_1^e + \boldsymbol{K}_2^e}_{\text{sum}} = \frac{EA}{L}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

Global stiffness

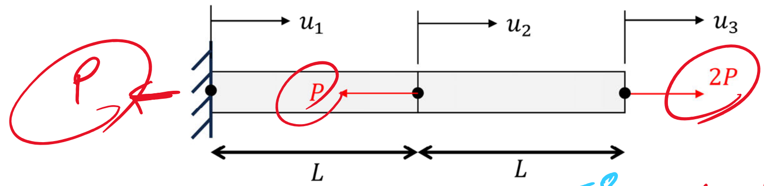
$\boldsymbol{F}$ contains the loads applied to each node: (R is an unknown placeholder for the wall reaction force)

$$\boldsymbol{F} = \begin{bmatrix} R \\ -P \\ 2P \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 3 \end{array}$$

Page 19

Tuesday, November 7, 2023 8:43 AM

FEA with Linear Rod Elements



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

=0

to solve

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

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

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

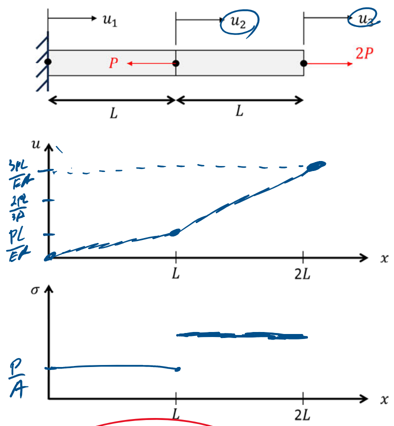
We can solve this to obtain $u_2 = \frac{PL}{EA}, u_3 = 3\frac{PL}{EA}$ ✓

$\underline{K}\,\underline{u} = \underline{F} \rightarrow \underline{u} = \underline{K}^{-1}\underline{F}$

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

1st Row: $\frac{EA}{L}\left(1 \cdot \underbrace{u_1}_{=0} - 1 \cdot \underbrace{u_2}_{\frac{PL}{EA}} + 0 \cdot u_3\right) = R = -P$

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:

$$\sigma_i = E_i \, \varepsilon_i$$

Element 1

$$\varepsilon_1 = \frac{\Delta L}{L} = \frac{u_2 - u_1}{L} = \frac{P}{EA}$$

$$\sigma_1 = E_1 \varepsilon_1 = \frac{P}{A}$$

Element 2

$$\varepsilon_2 = \frac{\Delta L}{L} = \frac{u_3 - u_2}{L} = \frac{2P}{EA}$$

$$\sigma_2 = E \varepsilon_2 = 2\frac{P}{A}$$

next class
Engineering Hall
406B1

OH
LUMEB 1028
12p - 1p