Structures and Materials 3

Finite Element Analysis

Dr Giuliano Allegri

Course Contents

- Lecture 1 - Introduction to the finite element method
- Lecture 2 - Bar Elements
  - Definition/formulation
  - Stiffness matrices
- Lecture 3 - Bar Elements
  - Stiffness matrices cont.
  - Local/global coordinate transformations
- Lecture 4 - Beam Elements
  - Definition/formulation

Course Contents

- Lecture 5 - Triangular and Quadrilateral Elements
  - Definition/formulation
  - Stiffness matrices

- Lecture 6 - Convergence and meshing quality

- Lecture 7 – Practical Application of FEA

References and Prerequisites

Textbooks:-

Finite Element Modelling for Stress Analysis……………… R. D. Cook

The Finite Element Method ……… O. C. Zienkiewicz and R. L. Taylor

A First Course in Finite Elements…………….J.Fish and T. Belytschko

1st Year course notes

Stresses and deformations ……………………………….Dr Kawashita

2nd Year course notes

Beams with thin-walled sections……………………………Dr Belnoue

Objectives

At the end of this lecture series you should be able to:-

- Understand the basis of finite element idealization and matrix analysis of structures
- Carry out finite element matrix analysis for simple bar and beam structures by hand.
- Understand the formulation of 2D membrane elements (triangular and quadrilateral)
- Meshing models in order to achieve convergence and take advantage of structural symmetries

Lab Sessions

- Lab Sessions start in Week 5 - Monday (28th Oct)
- 10:00 – 13:00: Demonstrator assistance 10:00 – 12:00
- **Week 9**: **FE test 1** – 1D elements only, to be solved in class; hand-in and marking on Blackboard – 12.5% of StM3 mark
- **Week 15**: **FE test 2** – 2D elements only, to be solved at home; hand-in and marking on Blackboard – 12.5% of StM3 mark
- The two tests are worth 25% of StM3 in total
- No further FEA exam question in exam!

Structures and Materials 3

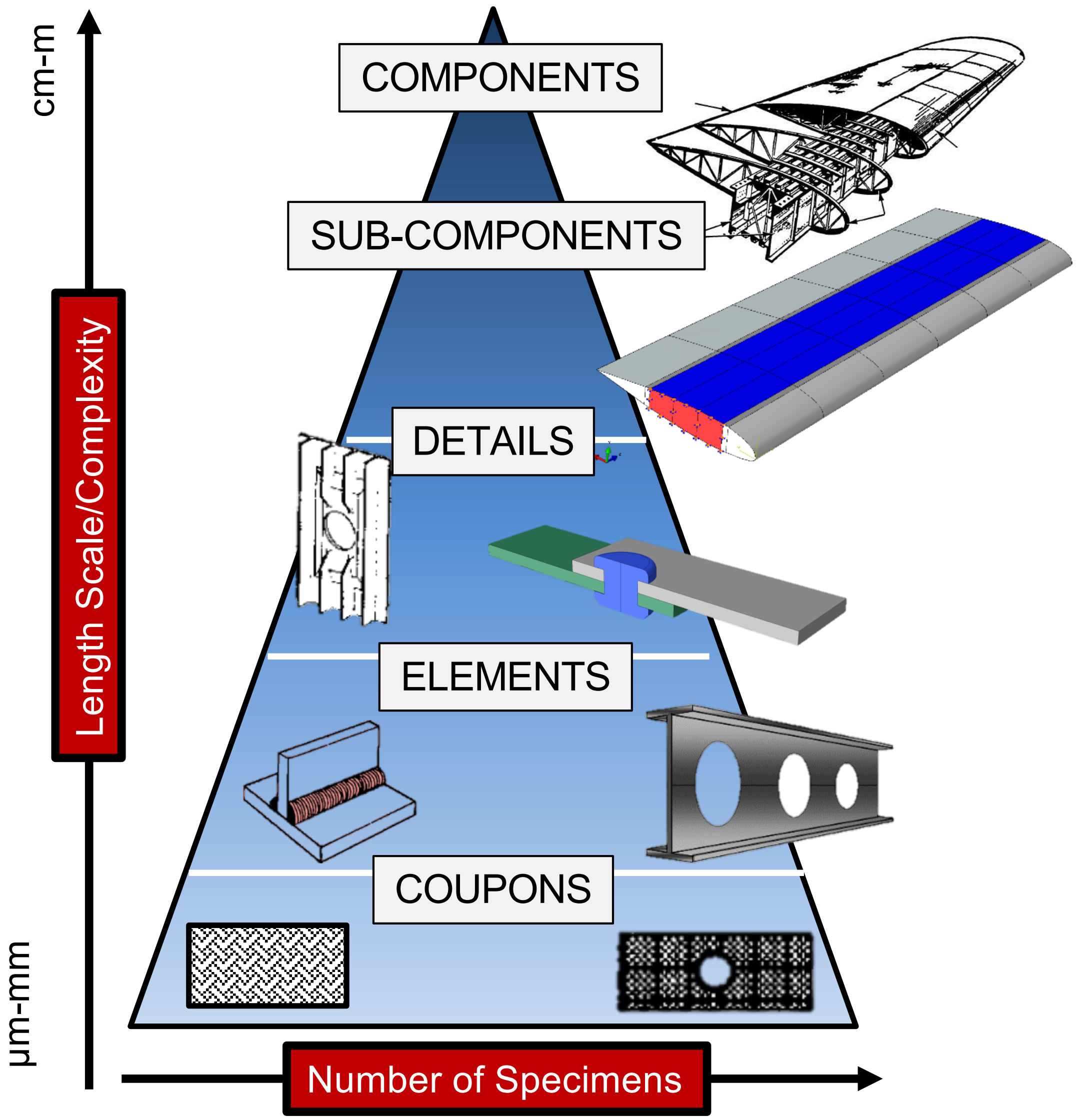
Introduction to FEA

Dr Giuliano Allegri

Lecture 1

- Introduction
- FE process
  - Idealisation
  - Discretisation
  - Solution
  - Post-processing
- Sources of Error
- Elements and Element Selection

Pyramid of Testing/Simulation

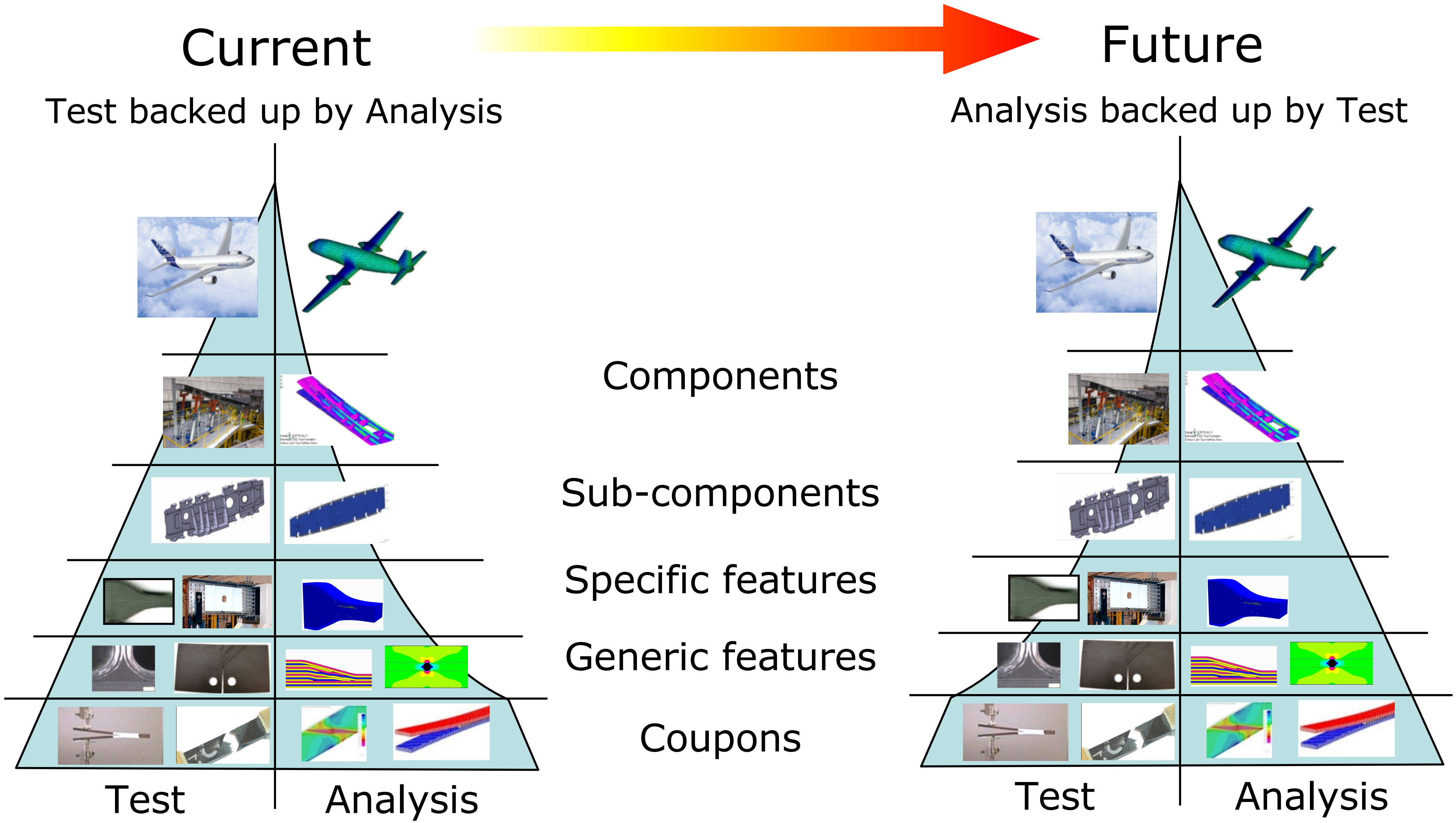


Virtual testing can:

- Identify key test data required for minimal model uncertainty
- Guide and inform the physical testing
- Augment the data with further combinations of test cases
- Replace physical tests at intermediate levels

Cost effective and optimised designs are a result of good synergy between testing and CAE

Virtual Testing of Composites



Introduction

- The finite element method is a numerical tool for solving engineering problems
- It is useful where analytical solutions do not exist
  e.g. complicated geometry, loading and material properties
- Many commercial codes exist for solving general problems
  e.g. Nastran, Ansys, Abaqus, Marc, Ideas
- Many applications - **Structural/Stress Analysis**, Fluid Flow, Heat Transfer, Electro-Magnetic Fields, Acoustics
- Small purpose written codes often used in research for specific tasks

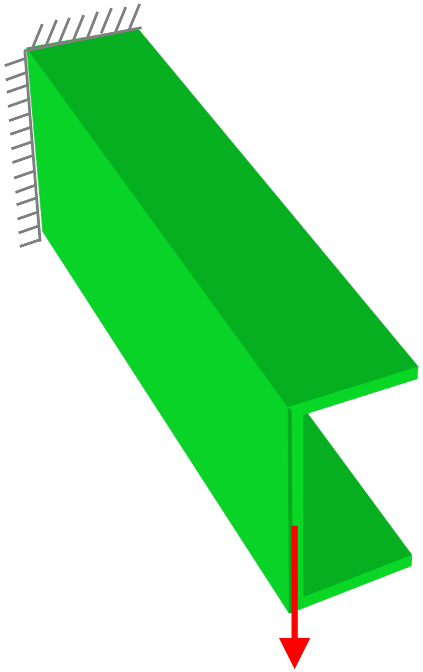
Recap

- Stress vs. strain?
- Hookes law?
- Modulus of Steel/Aluminium?
- Stress vs strength?
- Units?

Process

1. A complex physical entity is reduced to a simplified representation the captures the structural entities – IDEALISATION
2. A body is modelled by dividing it into an equivalent system of smaller bodies or units (finite elements) interconnected at points common to adjacent elements (nodes) – DISCRETISATION
3. Obtain a set of algebraic equations and solve for unknown nodal values (displacement) – SOLUTION
4. Stresses and strains are back calculated from displacements using constitutive relations – POST-PROCESSING

Step 1-Idealisation

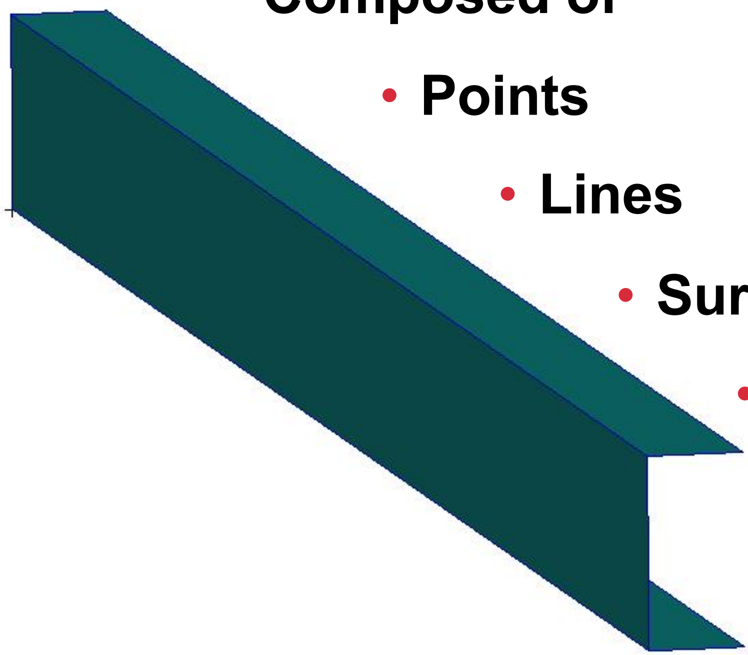


- Most important engineering practice
- Model to predict of behaviour of system
  - Aspects of interest
  - Solutions:
    - Analytical – regular shapes, simple BC, if exists
    - Numerical – approx.

Step 2-Discretisation (a) Mesh Generation

- No analytical solution ⇒ numerical ⇒ FE
- Discretisation ⇒ reduce no. of DOF
- Discrete model ⇒ elements interconnected

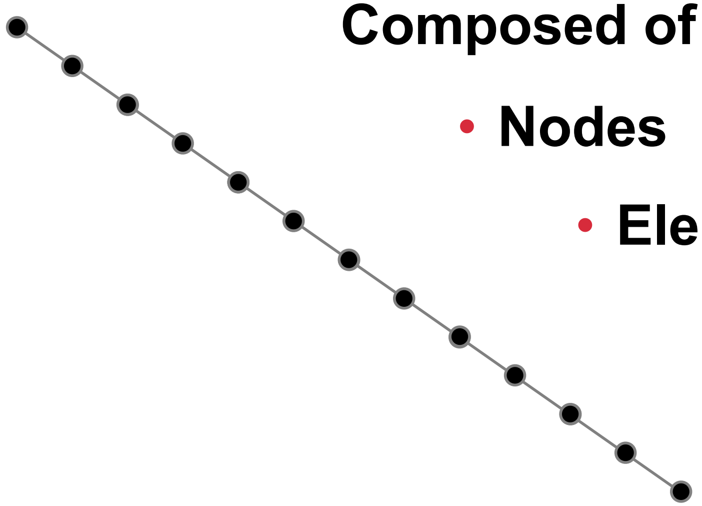
Geometry



Composed of

- **Points**
- **Lines**
- **Surfaces**
- **Solids**

FE Mesh

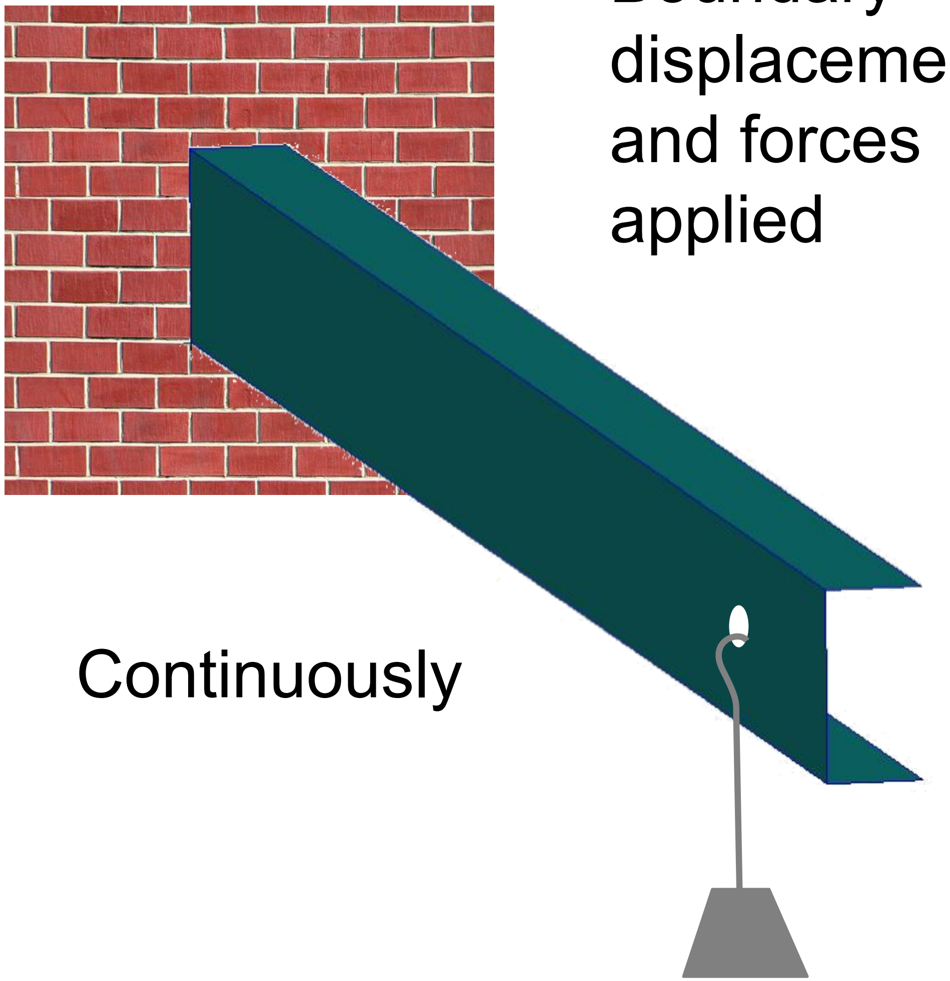


Composed of

- **Nodes**
- **Elements**

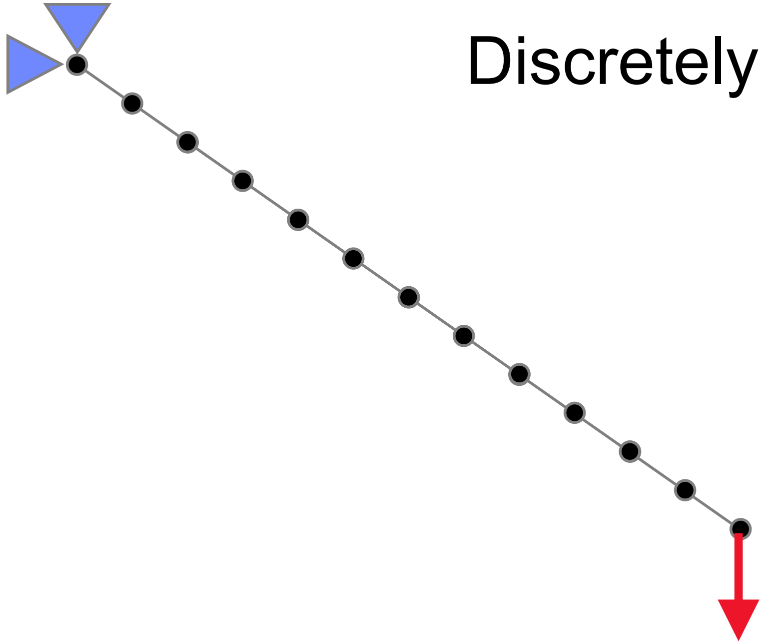
Step 2 - Discretisation (b) Boundary Conditions

Boundary displacements and forces applied



Continuously

Continuum



Discretely

FEA

Step 3-Solution

1. Form individual element stiffness matrices often in local element coordinate system

2. Transform *element stiffness matrix* into global coordinate system

3. Connect elements together, that is, assemble the element **[k]** matrices to obtain the structure or *“global” stiffness matrix* **[K]**

4. Assemble the loads into a global *load vector* **{F}**

5. Apply displacement boundary conditions

6. Solve equation set **{F} = [K] {U}** to give displacements **{U}**

Step 3-Solution

There are only 3 basic arguments that can be used to solve a structural problem (independent of type of structure/loading/material):

1. Equilibrium – relating external forces and moments (applied or reacted) to internal forces and moments (stress resultants).

2. Compatibility – relating strains to displacements

3. Constitutive – relating stress to strain
   e.g. linear elastic stress - strain law.

Step 3-Solution

Assumptions

e.g. pin jointed framework

Equilibrium

Relation between external/internal forces + moments

Compatibility

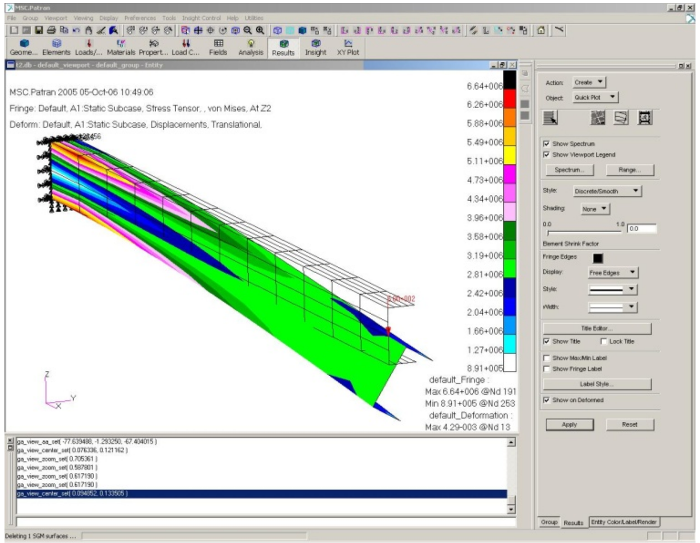
Relation between displacements + strains

Relation between stress + strains

Constitutive

We will only consider linear static analysis
i.e. small deformations, excluding plastic behaviour

Step 4-Postprocessing



- **{U}** used to back calculate elemental quantities, e.g. stresses and strains from displacements and element matrices
- A contour display shows the model and result
- Select display settings correctly

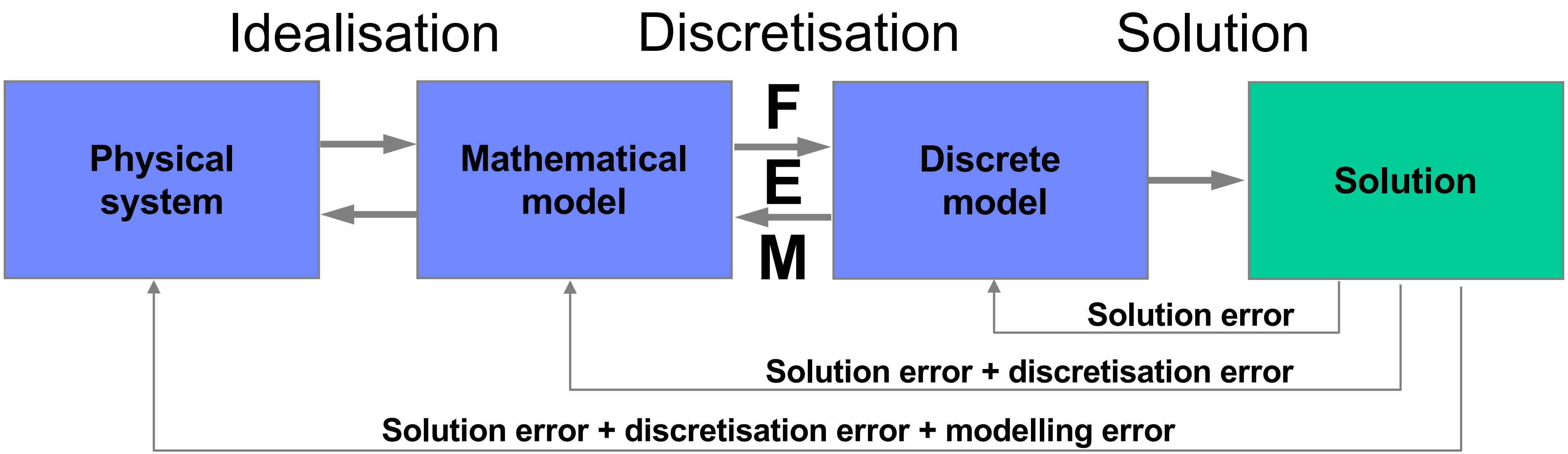
Variety of output:

- **stress components: helpful to understand load paths**
- **principal stresses: max./min. good for assessing fracture/buckling, directions useful to show load paths**
- **Von Mises equivalent stress: provides single number to locate hotspots and assess risk of yielding strains (useful for comparison with strain gauges)**

Advantages of FEA

- Any irregular or non-uniform boundary conditions
- Any irregular and non-uniform loading and geometry
- Size of the problem is not limited since it is ideally suitable for computers
- Solution of complex and non-linear problems is feasible

Sources of Error



- Modelling errors: difficult and expensive to evaluate, as requires tests

- Discretisation errors:
  - Mesh size $\rightarrow$ convergence
  - Mesh shape $\rightarrow$ regular & symmetric better
  - Assumption $\rightarrow$ eg displacement function chosen

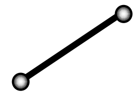
- Solution errors: numerical analysis, rounding, etc. Usually small

Validation

Verification

Typical Finite Elements

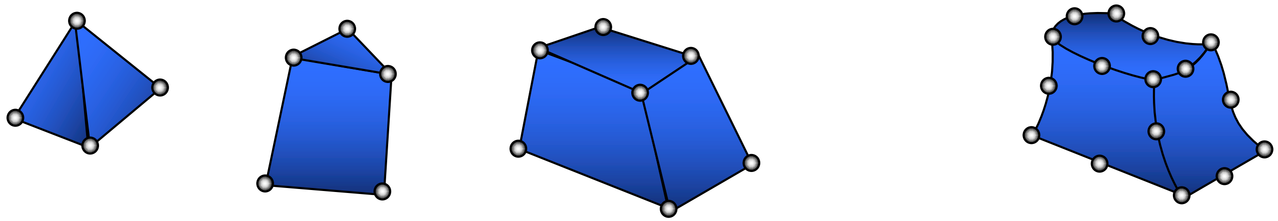
Linear (1D)



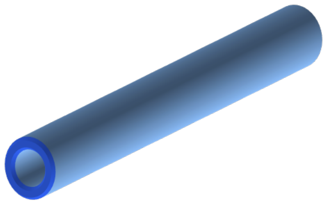
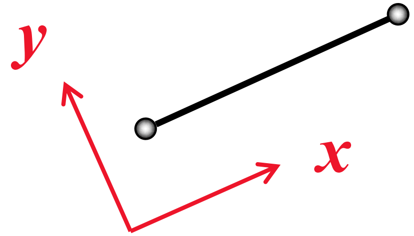
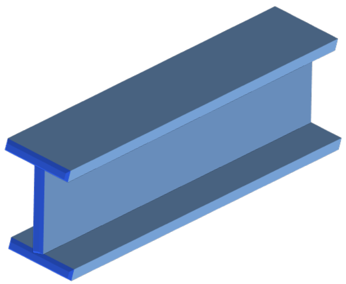
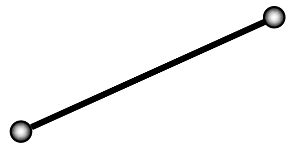
Higher order elements have mid-side nodes and can represent more complex shapes/stresses

Planar (2D)

Solid (3D)



Choosing Elements: 1D

Physical structural component	Mathematical model name	Finite element discretisation	Active Degrees of Freedom
	**Bar/Rod/ Cable/Truss**		Axial u_x, u_y, u_z
	**Beam/Bar**		Axial, bending, (torsion) u_x, u_y, u_z $\theta_x, \theta_y, \theta_z$

Condition for beam: cross-section dimensions << axial dimension

Limitations of Beam Elements 1

1. Assumes plane sections remain plane.
    - Means linear variation of strain through the depth.
    - Not valid for very highly curved beams, large axial tension/compression of non-solid sections (eg pipes, I-beams, U-beams).

2. No distortion of beam cross section
    - Section collapse may occur due to cross section change under applied loads
    - Results in very weak behaviour that is not predicted by beam theory, eg thin walled curved pipes.

Limitations of Beam Elements 2

3. Neglects effects of curvature
   - Through-the-depth Brazier stresses that develop in curved beams are ignored.
4. Ignores discontinuities/local effects
   - Stresses at kink in beam flange is ignored.
   - Stress concentrations due to rivet and bolt holes are ignored
5. Simple theory assumes linearity
   - Material stress-strain response is linear.
   - Deformation induced curvature is ignored.
   - Direction of loads does not change.

Structures and Materials 3

Bar Element

Dr Giuliano Allegri

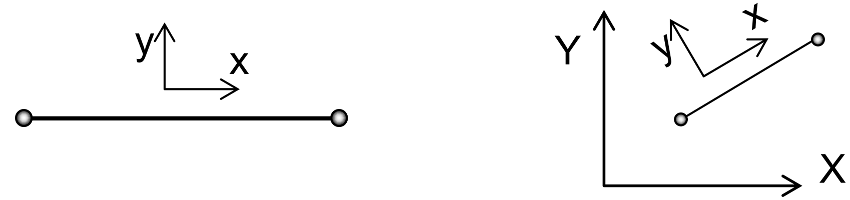
Lecture 2

- In-line 2D bar element/local axes
  - Definition and sign conventions
  - Example 1: Coincident 2 bar element structure - local element axes
    - Complete matrix formulation – *Method 1*

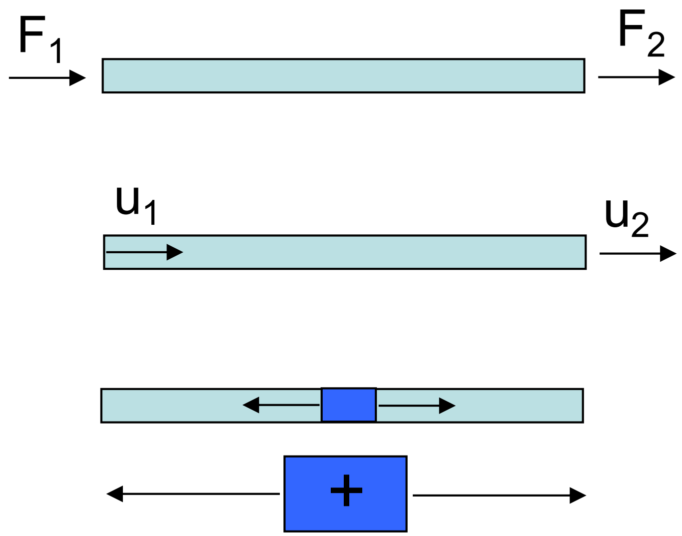
2D Bar Element

- **Definition :** A line element defined in one plane which resists axial forces only (tension or compression)

- **Local (element) axes:**

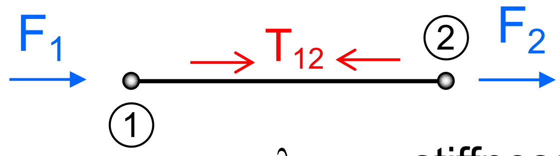


- **Sign Convention:**
  - External forces (Static sign convention)
  - Displacements
  - Internal forces (Deformation sign convention)



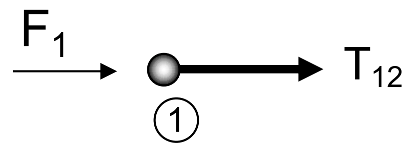
i.e. tensile positive

In-line 2D bar element



λ_{12} = stiffness of element 1-2
= force per unit displacement

- For **force-displacement equation** apply arguments :
  - Equilibrium: Sum of forces @ each node = 0
    - Node 1: $F_1 + T_{12} = 0$ (i)
    - Node 2: $F_2 - T_{12} = 0$ (ii)



  - Compatibility $e_{12} = u_2 - u_1$ (iii)
  - Constitution $T_{12} = \lambda_{12}\, e_{12}$ (iv)

Force-Disp. Equation

- Combining (i) – (iv) to obtain force-displacement equations
  - (i),(iv),(iii) $F_1 + \lambda_{12}(u_2 - u_1) = 0 \rightarrow F_1 = \lambda_{12} u_1 - \lambda_{12} u_2$
  - (ii),(iv),(iii) $F_2 - \lambda_{12}(u_2 - u_1) = 0 \rightarrow F_2 = -\lambda_{12} u_1 + \lambda_{12} u_2$

(v)

- In matrix form:

$$\begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix} = \lambda_{12} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

External nodal force vector

Nodal displacement vector vector

$$\{F\} = [k] \,.\, \{u\}$$

- This is the general form of the force-displacement equation for a 2D bar element.

Stiffness Matrix

- We can write a general 2 x 2 matrix for a 2D bar element as:

$$\lambda_{12}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Where λ_i = stiffness of bar = $\dfrac{A_i E_i}{l_i}$

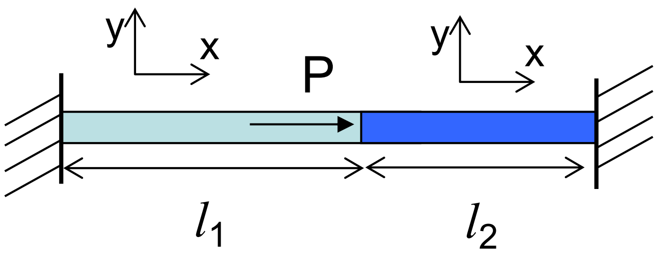
- This is the "Elemental Stiffness Matrix"

- We can now use this general matrix to build up frameworks of bar elements

Example 1

- Consider a coincident 2 bar element structure w.r.t. local element coordinates
- Bar stiffness

$$\lambda = \frac{P}{e}$$

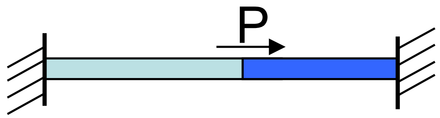


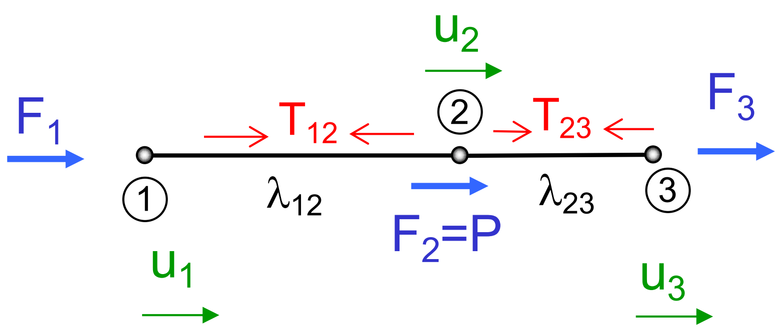
i.e. force per unit displacement

- Now $\sigma = \dfrac{P}{A}$, $\varepsilon = \dfrac{e}{l}$, $E = \dfrac{\sigma}{\varepsilon}$

- So $\lambda_i = \dfrac{A_i E_i}{l_i}$ for a given bar i

Finite Element Model





- Nodes 1, 2, 3

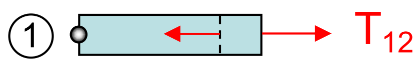
- Bar elements

  1 - 2, 2 - 3

Tip:- Draw all unknown forces and moments in a +ve sense according to external or internal sign convention

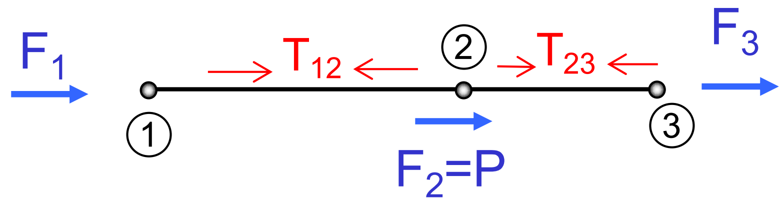
F_i = External forces, applied + reactions

T_i = Internal forces i.e.

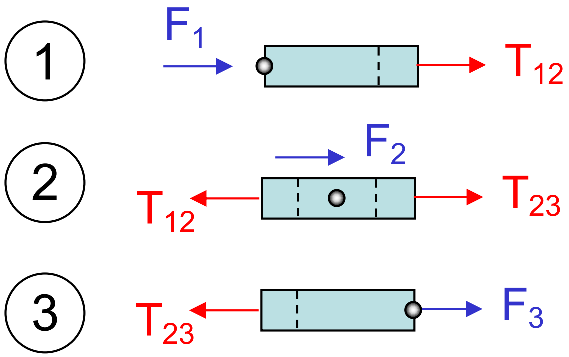


U_1 = Nodal displacements

Equilibrium



- Static equilibrium of forces
  - at each node $\sum \overset{+}{\rightarrow} = 0$



$$F_1 + T_{12} = 0 \qquad (1)$$

$$F_2 - T_{12} + T_{23} = 0 \qquad (2)$$

$$F_3 - T_{23} = 0 \qquad (3)$$

- Overall: $F_1 + F_2 + F_3 = 0$ (equilibrium of external forces)
  - linear combination of Eq^n (1), (2), (3)

- Only 3 equilibrium eq^{ns} but 4 unknowns
  F_1, F_3, T_{12}, T_{23}

- One redundancy – "Statically indeterminate"

Compatibility (Geometry of deformation)

Element 1 - 2 extension $e_{12} = u_2 - u_1$ (4)

Element 2 - 3 extension $e_{23} = u_3 - u_2$ (5)

Constitutive relation (Force – displacement)

Element 1 - 2 $T_{12} = \lambda_{12} \, . \, e_{12}$ (6)

Element 2 - 3 $T_{23} = \lambda_{23} \, . \, e_{23}$ (7)

where $$\lambda_i = \frac{A_i E_i}{l_i}$$

Summary of Equations and Unknowns

- Equations:

Force Equilibrium:	3 Eq^{ns}	(1), (2), (3)
Geometry of deform^{n}:	2 Eq^{ns}	(4), (5)
Force-displacement:	2 Eq^{ns}	(6), (7)
Total	7 Eq^{ns}	

- Unknowns:

External forces	$F_1 = ?$	
	$F_2 = P$	–> Applied
	$F_3 = ?$	
Internal forces	$T_{12} = ?$	
	$T_{23} = ?$	
Displacements	$u_1 = 0$	–> Fixed
	$u_2 = ?$	
	$u_3 = 0$	–> Fixed
Deformations	$e_{12} = ?$	
	$e_{23} = ?$	
Total	7 Unknowns	

Stiffness Matrix Formulation

- Create matrix equation set: $\{F\} = [k] . \{u\}$

External applied forces + reactions — Stiffness matrix — Nodal displacements

- By manipulating simultaneous eq^{ns} (1) – (7)

$$F_1 = \lambda_{12} u_1 - \lambda_{12} u_2 + 0 u_3 \quad (10)$$

$$F_2 = -\lambda_{12} u_1 + (\lambda_{12} + \lambda_{23}) u_2 - \lambda_{23} u_3 \quad (11)$$

$$F_3 = 0 u_1 - \lambda_{23} u_2 + \lambda_{23} u_3 \quad (12)$$

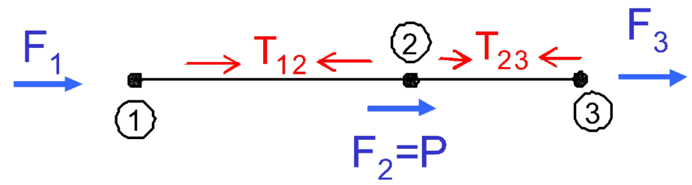
Check this yourselves

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} \lambda_{12} & -\lambda_{12} & 0 \\ -\lambda_{12} & \lambda_{12} + \lambda_{23} & -\lambda_{23} \\ 0 & -\lambda_{23} & \lambda_{23} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \quad (13)$$

Solving for Unknowns

- Include known boundary conditions
  i.e. $F_2=P$, $u_1=u_3=0$

$$\begin{Bmatrix} F_1 \\ P \\ F_3 \end{Bmatrix} = \begin{bmatrix} \lambda_{12} & -\lambda_{12} & 0 \\ -\lambda_{12} & \lambda_{12}+\lambda_{23} & -\lambda_{23} \\ 0 & -\lambda_{23} & \lambda_{23} \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ 0 \end{Bmatrix}$$



Row 2 $\rightarrow \quad P=(\lambda_{12}+\lambda_{23})u_2 \quad \rightarrow \quad u_2=\dfrac{P}{(\lambda_{12}+\lambda_{23})}$

Row 1 $\rightarrow \quad F_1=-\lambda_{12}u_2 \quad \rightarrow \quad F_1=\dfrac{-\lambda_{12}P}{(\lambda_{12}+\lambda_{23})}$

Row 3 $\rightarrow \quad F_3=-\lambda_{23}u_2 \quad \rightarrow \quad F_3=\dfrac{-\lambda_{23}P}{(\lambda_{12}+\lambda_{23})}$

i.e. all external forces and displacements now known

Internal forces

- Now use the element constitutive relations to find the internal forces:

(6) $T_{12} = \lambda_{12}\, e_{12}$

$$= \lambda_{12}\left(u_2 - \underbrace{\cancel{u_1}}_{=0}\right) = \lambda_{12}\, u_2 = \frac{\lambda_{12}\, P}{\lambda_{12}+\lambda_{23}}$$

+ve, i.e. tension – as expected

$$T_{23} = \lambda_{23}\, e_{23}$$

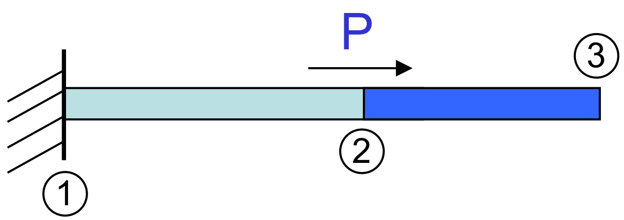
$$= \lambda_{23}\left(\underbrace{\cancel{u_3}}_{=0} - u_2\right) = -\lambda_{23}\, u_2 = -\frac{\lambda_{23}\, P}{\lambda_{12}+\lambda_{23}}$$

-ve, i.e. compression – as expected

Check balance: i.e. overall equilibrium and nodal equilibrium

"System" stiffness matrix

- For the in-line 2 bar structure e^{qn} (13) can now be used to solve any axial loading problem for this configuration
- E.g. Node 3 released



Here F_2 = P, F_3 = 0, u_1 = 0

- Inserting boundary conditions into (13) gives:-

$$\begin{Bmatrix} F_1 \\ P \\ 0 \end{Bmatrix} = \begin{bmatrix} \lambda_{12} & -\lambda_{12} & 0 \\ -\lambda_{12} & \lambda_{12}+\lambda_{23} & -\lambda_{23} \\ 0 & -\lambda_{23} & \lambda_{23} \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix}$$

i.e.

$$F_1 = -\lambda_{12} u_2 \qquad \text{(i)}$$
$$P = (\lambda_{12} + \lambda_{23}) u_2 - \lambda_{23} u_3 \qquad \text{(ii)}$$
$$0 = -\lambda_{23} u_2 + \lambda_{23} u_3 \qquad \text{(iii)}$$

- (iii) gives $u_3 = u_2$
- Substitute into (ii) gives:

$$P = (\lambda_{12} + \lambda_{23})u_2 - \lambda_{23} u_2$$
$$= \lambda_{12} u_2$$

i.e. $u_2 = \dfrac{P}{\lambda_{12}}$ (iv) - as expected by inspection

All displacements now known

- And so using (iv) in (i)

$$F_1 = -\lambda_{12} \frac{P}{\lambda_{12}}$$

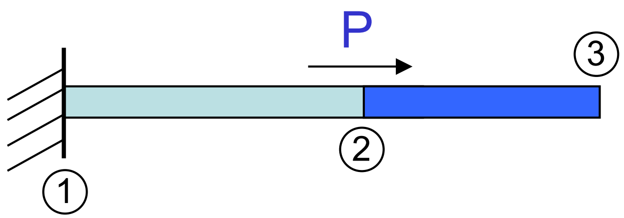
$F_1 = -P$ (v) - again as expected by inspection

All forces now known

- Finally for internal forces use constitutive relations (6)

$$T_{12} = \lambda_{12}(u_2 - u_1) = \lambda_{12}\left(\frac{P}{\lambda_{12}}\right) = P$$

$$T_{23} = \lambda_{23}(u_3 - u_2) = -\lambda_{23}\left(\frac{P}{\lambda_{12}} - \frac{P}{\lambda_{12}}\right) = 0$$



Stiffness Matrix Properties

- Recall - the determinant of a 3×3 matrix is given by

$$A = \begin{bmatrix} a & b & c \\ d & e & f \\ g & h & i \end{bmatrix}$$

$$|A| = aei - afh + bfg - bdi + cdh - ceg$$

- For our 2 bar system stiffness matrix determinant =

$$\begin{bmatrix} \lambda_{12} & -\lambda_{12} & 0 \\ -\lambda_{12} & \lambda_{12}+\lambda_{23} & -\lambda_{23} \\ 0 & -\lambda_{23} & \lambda_{23} \end{bmatrix}$$

$$\lambda_{12}.(\lambda_{12}+\lambda_{23}).\lambda_{23} - \lambda_{12}.(-\lambda_{23}).(-\lambda_{23}) - (-\lambda_{12}).(-\lambda_{12}).\lambda_{23}$$

$$= \lambda_{12}{}^2\lambda_{23} + \lambda_{12}\lambda_{23}{}^2 - \lambda_{12}\lambda_{23}{}^2 - \lambda_{12}{}^2\lambda_{23} = 0$$

- A matrix is invertible only if its determinant is non-zero.

$$[F] = [K]\{u\} \rightarrow \{u\} = [K]^{-1}[F]$$

- A general solution for displacements cannot be found unless some are known i.e. usually set to zero to prevent rigid body translation and displacement

Structures and Materials 3

Bar Element (continued…)

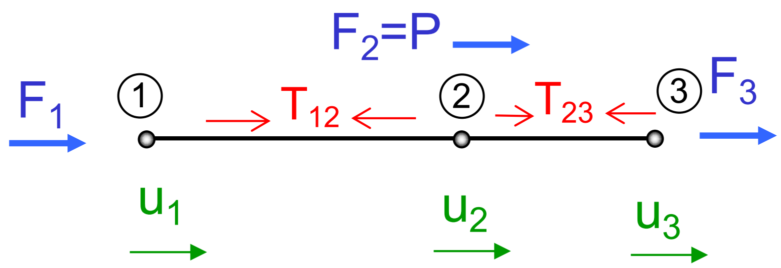
Dr Giuliano Allegri

Lecture 3

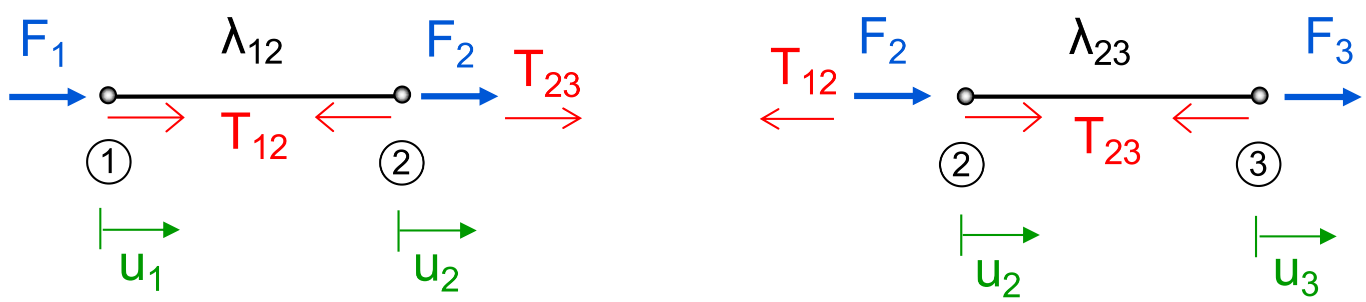
- Coincident 2 bar element structure
  - Assembled matrix formulation – *Method 2*
- Summary of method
- Inclined 2D bar element
  - Local/global axes transformation
  - Example 2: Non-coincident 3 bar element structure/global structural axes
- Special properties of stiffness matrixes

Assembled matrix formulation

- Previously we solved for our 2 bar structure

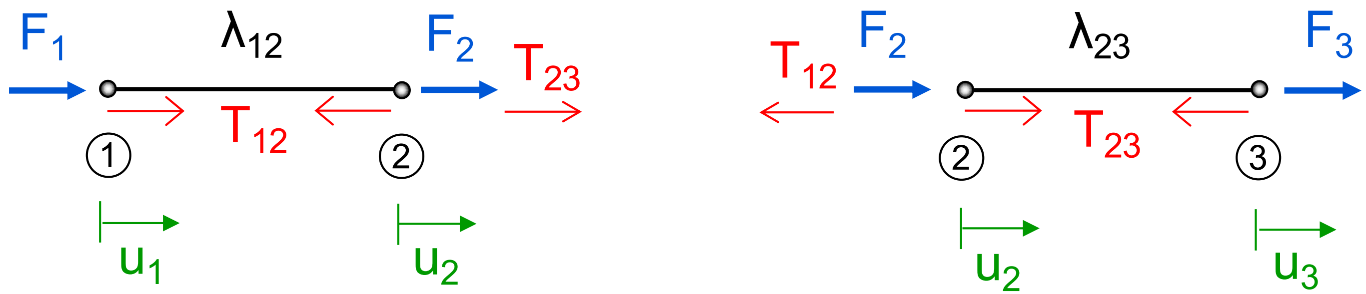


- We now formulate the structure stiffness matrix from assembly of individual element stiffness matrixes
  - Consider separate FBD's for each element:



$$\lambda_i = \frac{A_i E_i}{l_i}$$

Assembled matrix formulation



$$\lambda_i = \frac{A_i E_i}{l_i}$$

– Using same procedure as original formulation but for each element separately:

e.g. element ①-②:

- Equilibrium: $\sum \longrightarrow = 0$ at nodes
  – Node ①: $F_1 + T_{12} = 0$ (i)
  ②: $F_2 + T_{23} - T_{12} = 0$ (ii)
- Compatibility: $e_{12} = u_1 - u_2$ (iii)
- Constitution: $T_{12} = \lambda_{12} e_{12}$ (iv)

- Combining (i) – (iv) to obtain force – displacement equations:

(i), (iv), (iii) $\quad F_1 + \lambda_{12}(u_2 - u_1) = 0 \quad \rightarrow \quad F_1 = \lambda_{12}\, u_1 - \lambda_{12}\, u_2$

(ii), (iv), (iii) $\quad F_2 + T_{23} - \lambda_{12}(u_2 - u_1) = 0 \quad \rightarrow \quad F_2 + T_{23} = -\lambda_{12}\, u_1 + \lambda_{12}\, u_2$

- In matrix form:

$$\left\{\begin{matrix} F_1 \\ F_2 + T_{23} \end{matrix}\right\} = \lambda_{12}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\left\{\begin{matrix} u_1 \\ u_2 \end{matrix}\right\}$$

Similarly for element ② - ③

$$\left\{\begin{matrix} F_2 - T_{12} \\ F_3 \end{matrix}\right\} = \lambda_{23}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\left\{\begin{matrix} u_2 \\ u_3 \end{matrix}\right\}$$

(15)

– i.e. 2 x 2 stiffness matrix for each bar element

- Expanding:

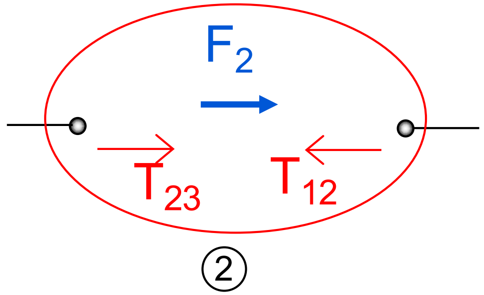
$$\begin{Bmatrix} F_1 \\ F_2 + T_{23} \\ 0 \end{Bmatrix} = \begin{bmatrix} \lambda_{12} & -\lambda_{12} & 0 \\ -\lambda_{12} & \lambda_{12} & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ 0 \end{Bmatrix} \quad \text{and} \quad \begin{Bmatrix} 0 \\ F_2 - T_{12} \\ F_3 \end{Bmatrix} = \begin{bmatrix} 0 & 0 & 0 \\ 0 & \lambda_{23} & -\lambda_{23} \\ 0 & -\lambda_{23} & \lambda_{23} \end{bmatrix} \begin{Bmatrix} 0 \\ u_2 \\ u_3 \end{Bmatrix} \qquad (16)$$

- Assembling: adding matrix equations (16) together (i.e. linking structure at node ②)

e.g. line 2: $F_2 + \underbrace{T_{23} + F_2 - T_{12}}_{} = -\lambda_{12}\, u_1 + \lambda_{12}\, u_2 + \lambda_{23}\, u_2 - \lambda_{23}\, u_3$

= 0 for equilibrium at node (2)

$$F_2 = -\lambda_{12}\, u_1 + (\lambda_{12} + \lambda_{23})\, u_2 - \lambda_{23}\, u_3$$



$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} = \begin{bmatrix} \lambda_{12} & -\lambda_{12} & 0 \\ -\lambda_{12} & (\lambda_{12} + \lambda_{23}) & -\lambda_{23} \\ 0 & -\lambda_{23} & \lambda_{23} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} \qquad (17) = (13)$$

i.e. matrix equation for complete structure

SUMMARY (1)

Procedure for FE model formulation and solution:

1. Model structure using finite elements
2. Obtain stiffness matrix [k] for each element
   a) Draw FBD for each element
      Write out equations:
      - Equilibrium
      - Compatibility
      - Constitutive

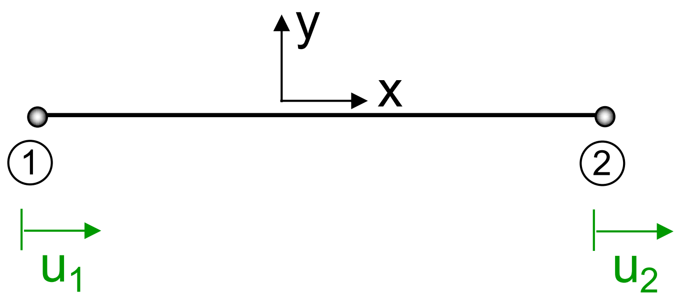
      for each element
   b) Manipulate into form {F} = [k] {u}
      - finding [k] in terms of λ
3. Expand separate element stiffness matrixes and assemble i.e. add for complete structure
   - This is equivalent to joining the structure at connecting nodes

SUMMARY (2)

4. Apply boundary conditions: known loads and disp's
5. Solve equations for unknown external loads and displacements
6. Find internal loads and calculate stresses

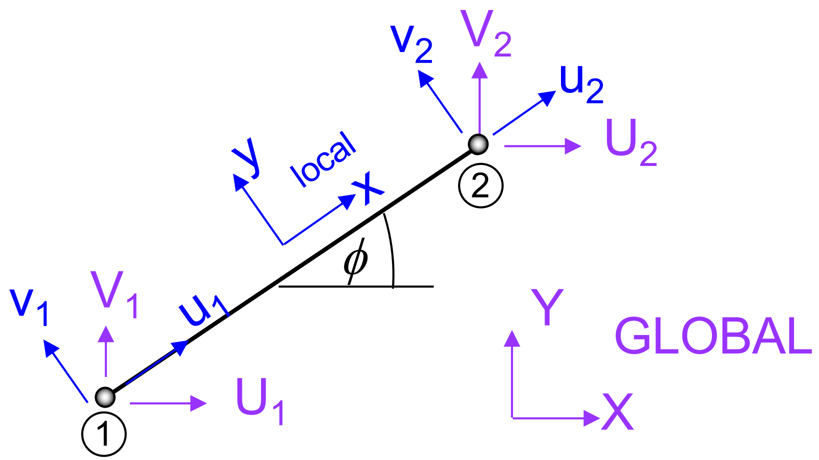
Inclined 2D bar elements

So far we have only considered bars wrt aligned "local axes"



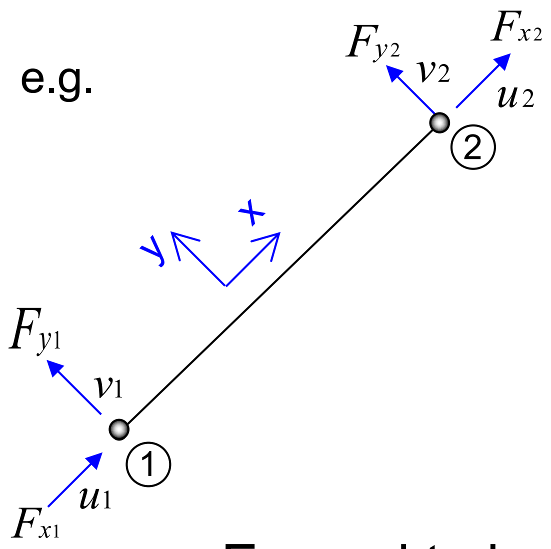
Only x direction relevant here

To enable framework of bars to be modelled we must now consider bars wrt <u>global axes</u> and transformation from local element axes



Single element wrt local co-ords: "x, y"

e.g.



$$\begin{Bmatrix} F_{x1} \\ F_{x2} \end{Bmatrix} = \lambda \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}$$

Expand to include y components: F_{y1}, F_{y2}, v_1, v_2

$\left.\begin{array}{l} \text{but note: } F_{y1} = F_{y2} = 0 \\ \text{and transverse stiffness} = 0 \end{array}\right\}$ by definition → i.e. only axial forces can be carried by a bar element. Bar stiffness is only defined axially

$$\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

So:

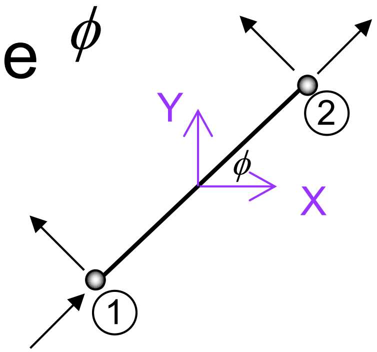
from 2 x 2

to 4 x 4 ➡

$$\begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \end{Bmatrix} = \lambda \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} \qquad (18)$$

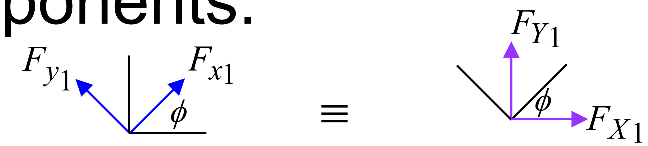
Single element wrt GLOBAL co-ords "X, Y"

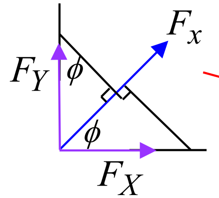
e.g. at angle ϕ

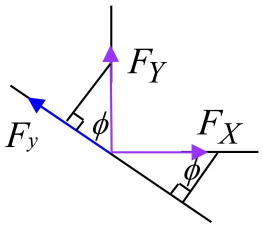


Writing local element x, y forces components in terms of global X, Y components:

e.g. at node ①







$$\left.\begin{aligned} F_x &\equiv F_X\cos\phi + F_Y\sin\phi \\ F_y &\equiv -F_X\sin\phi + F_Y\cos\phi \end{aligned}\right\} \quad T_\varphi = \begin{bmatrix} \cos & \sin \\ -\sin & \cos \end{bmatrix}$$

i.e. transformation relationship x, y → X, Y

$$\{F\}_{xy} = [T_\varphi]\{F\}_{XY}$$

Apply transformation at each node of bar element

e.g. in matrix form:

$$\begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \end{Bmatrix} = \begin{bmatrix} \cos\phi & \sin\phi & 0 & 0 \\ -\sin\phi & \cos\phi & 0 & 0 \\ 0 & 0 & \cos\phi & \sin\phi \\ 0 & 0 & -\sin\phi & \cos\phi \end{bmatrix} \begin{Bmatrix} F_{X1} \\ F_{Y1} \\ F_{X2} \\ F_{Y2} \end{Bmatrix} \quad (19)$$

i.e. $\{F\}_{xy_i} = [T]\{F\}_{XY_i}$ (20) where $[T]$ = "Transformation Matrix" for bar element

Note, displacements transform according to the same law:

$$\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix} = \begin{bmatrix} & & \\ & T & \\ & & \end{bmatrix} \begin{Bmatrix} U_1 \\ V_1 \\ U_2 \\ V_2 \end{Bmatrix} \quad (21)$$

i.e. $\{\delta\}_{xy} = [T]\{\Delta\}_{XY}$

Substituting (19), (21) into (18)

$$\{F\}_{xy_i} = [T]\{F\}_{XY_i} \ , \quad \{\delta\}_{xy_i} = [T]\{\Delta\}_{XY_i} \quad \text{into} \quad \{F\}_{xy_i} = \lambda[k]_{xy_i}\{\delta\}_{xy_i}$$

to give:

$$[T]\begin{Bmatrix} F_{X1} \\ F_{Y1} \\ F_{X2} \\ F_{Y2} \end{Bmatrix} = \lambda \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} [T] \begin{Bmatrix} U_1 \\ V_1 \\ U_2 \\ V_2 \end{Bmatrix} \qquad (22)$$

i.e. $$[T]\{F\}_{XY} = [k]_{xy}\,[T]\{\Delta\}_{XY}$$

$$\{F\}_{XY} = [T]^{-1}[k]_{xy}[T]\{\Delta\}_{XY}$$

$$\{F\}_{XY} = \underbrace{[T]'\,[k]_{xy}\,[T]}\{\Delta\}_{XY} \qquad (23)$$

Note [T] matrix is orthogonal
so $[T]^{-1} = [T]^{T}$ i.e. [T]'
i.e. inverse = transpose

i.e. $$\{F\}_{XY} = [K]_{XY}\{\Delta\}_{XY} \qquad (24)$$

Bar element stiffness matrix
wrt global co-ordinates

In full:

$$[K]_{XY} = [T]'[k]_{xy}[T] = \begin{bmatrix} c & -s & 0 & 0 \\ s & c & 0 & 0 \\ 0 & 0 & c & -s \\ 0 & 0 & s & c \end{bmatrix} \lambda \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} c & s & 0 & 0 \\ -s & c & 0 & 0 \\ 0 & 0 & c & s \\ 0 & 0 & -s & c \end{bmatrix} \quad (25)$$

$$= \lambda \begin{bmatrix} c^2 & sc & -c^2 & -sc \\ sc & s^2 & -sc & -s^2 \\ -c^2 & -sc & c^2 & sc \\ -sc & -s^2 & sc & s^2 \end{bmatrix} \quad (26)$$

Where $s = \sin\phi,\ c = \cos\phi$

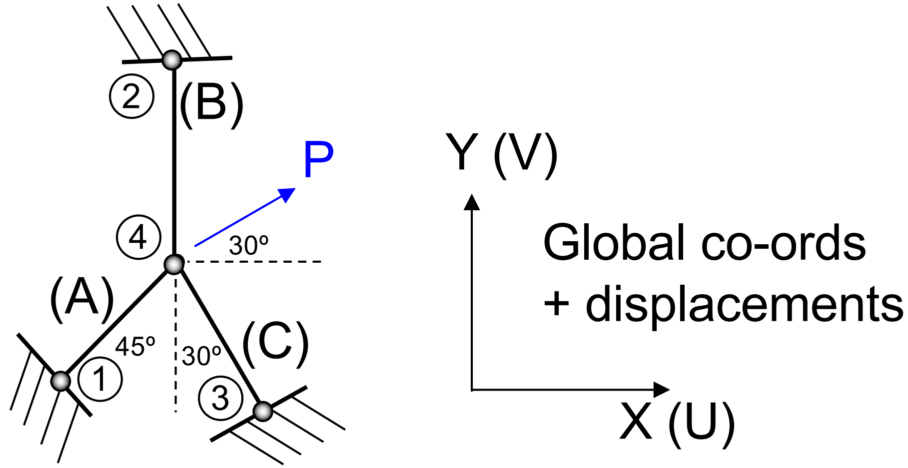
From (26) the global stiffness matrix can be obtained for any bar element by inserting the appropriate angle ϕ

Example 2

Non-coincident three bar element structure considered wrt global element co-ordinates

Bars: all: length l

stiffness λ



Requirement: to find bar forces, reactions and node (4) displacements

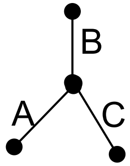
Note

– 3 bar forces etc
- But only 2 equations of static equilibrium at (4) i.e. $\sum\uparrow, \sum\rightarrow$

↳ Statically indeterminate problem

Using convenient notation for elements (A) (B) (C)

viz: global stiffness matrixes: $[a_{ij}],[b_{ij}],[c_{ij}]$



i, j = row, col no. of matrix

e.g. for element (A)

① - ④

$$\begin{Bmatrix} F_{X1} \\ F_{Y1} \\ F_{X4} \\ F_{Y4} \end{Bmatrix} = \begin{bmatrix} a_{11} & a_{12} & a_{13} & a_{14} \\ a_{21} & a_{22} & a_{23} & a_{24} \\ a_{31} & a_{32} & a_{33} & a_{34} \\ a_{41} & a_{42} & a_{43} & a_{44} \end{bmatrix}_{XY} \begin{Bmatrix} U_1 \\ V_1 \\ U_4 \\ V_4 \end{Bmatrix} \qquad (27)$$

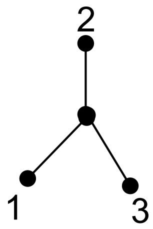
Assemble for complete structure:

$$\begin{Bmatrix} F_{X_1} \\ F_{Y_1} \\ F_{X_2} \\ F_{Y_2} \\ F_{X_3} \\ F_{Y_3} \\ F_{X_4} \\ F_{Y_4} \end{Bmatrix} = \begin{bmatrix} a_{11} & a_{12} & 0 & 0 & 0 & 0 & a_{13} & a_{14} \\ a_{21} & a_{22} & 0 & 0 & 0 & 0 & a_{23} & a_{24} \\ 0 & 0 & b_{11} & b_{12} & 0 & 0 & b_{13} & b_{14} \\ 0 & 0 & b_{21} & b_{22} & 0 & 0 & b_{23} & b_{24} \\ 0 & 0 & 0 & 0 & c_{11} & c_{12} & c_{13} & c_{14} \\ 0 & 0 & 0 & 0 & c_{21} & c_{22} & c_{23} & c_{24} \\ a_{31} & a_{32} & b_{31} & b_{32} & c_{31} & c_{32} & (a_{33}+b_{33}+c_{33}) & (a_{34}+b_{34}+c_{34}) \\ a_{41} & a_{42} & b_{41} & b_{42} & c_{41} & c_{42} & (a_{43}+b_{43}+c_{43}) & (a_{44}+b_{44}+c_{44}) \end{bmatrix} \begin{Bmatrix} U_1 \\ V_1 \\ U_2 \\ V_2 \\ U_3 \\ V_3 \\ U_4 \\ V_4 \end{Bmatrix} \qquad (28)$$

Note: node 4 common to each element here

- Apply boundary constraint conditions:

  i.e. $u_1 = v_1 = u_2 = v_2 = u_3 = v_3 = 0$



  ↳ Reduced stiffness matrix (i.e. many rows, columns may be omitted since zero)

$$\begin{Bmatrix} F_\alpha \\ F_\beta{}^{?} \end{Bmatrix} = \begin{bmatrix} K_{\alpha\alpha} & K_{\alpha\beta} \\ K_{\beta\alpha} & K_{\beta\beta} \end{bmatrix} \begin{Bmatrix} \Delta_\alpha{}^{?} \\ \Delta_\beta \end{Bmatrix} \quad (29)$$

Where F_α = known forces

Δ_α = displacements to be found?

F_β = external forces + reactions to be found

Δ_β = constrained displacements = 0

- Invert the reduced stiffness matrix to find displacements

  i.e. mathematically:

1. Find unknown disps

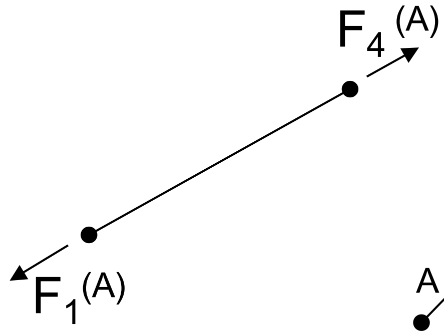
$$\{F_\alpha\} = [K_{\alpha\alpha}]\{\Delta_\alpha\} \longrightarrow \{\Delta_\alpha\} = [K_{\alpha\alpha}]^{-1}\{F_\alpha\} \quad (30)$$

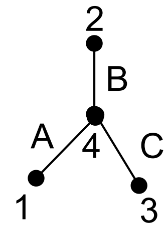
$$\{F_\beta\} = [K_{\beta\alpha}]\{\Delta_\alpha\} \longrightarrow \{F_\beta\} = [K_{\beta\alpha}][K_{\alpha\alpha}]^{-1}\{F_\alpha\}$$

2. Find unknown external forces

Finally

- Calculate internal force T_{14} from equilibrium sums at each node

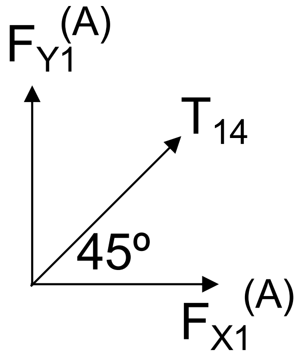




e.g. for element (A) using equation (27)

$$F_{X_1}{}^{(A)} = a_{11}U_1 + a_{12}V_1 + a_{13}U_4 + a_{14}V_4$$

e.g.



$$\sum\rightarrow \quad +T_{14}\cos 45 + F_{X1}{}^{(A)} = 0$$

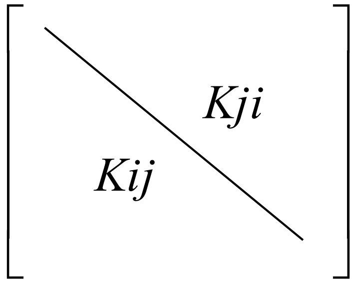
$$T_{14} = \frac{-F_{X1}{}^{(A)}}{\cos 45}$$

Note, $F_{x1}{}^{(A)}$ = -ve
Therefore T_{14} +ve
tension so looks ok

Special Properties of Stiffness Matrixes

(i) Symmetric about leading diagonal

$K_{ij} = K_{ji}$



i.e. "reciprocal behaviour" - deflection **d**$_i$ (at point **i**) due to a unit force **p**$_j$ (at point **j**) is equal to the deflection **d**$_j$ (at point **j**) due to a unit force **p**i (at point **i**)

(ii) Columns relating to forces sum to zero

i.e. sum of external forces must be zero for equilibrium

(iii) Stiffness matrix determinants are zero – matrixes are "singular"

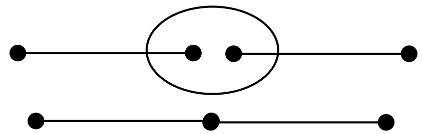
i.e. a solution is only possible provided that sufficient boundary restraints are specified to prevent the structure becoming a mechanism and to prevent rigid body movement

iv) Order determined by number of related nodal forces and displacements

e.g. single bar = 2 x 2

$$\begin{bmatrix} \# & \# \\ \# & \# \end{bmatrix}$$

two connected bars = 3 x 3



$$\begin{bmatrix} \# & \# & \# \\ \# & \# & \# \\ \# & \# & \# \end{bmatrix}$$

(v) Main diagonal terms are made up of summed stiffnesses of joining elements

$$\begin{bmatrix} \# & & \\ & \# & \\ & & \# \end{bmatrix}$$

(vi) Non zero terms tend to be concentrated near main diagonal

(vii) All terms on the leading diagonal are positive

i.e. positive nodal forces → positive nodal displacements

Structures and Materials 3

Beam Elements

Dr Giuliano Allegri

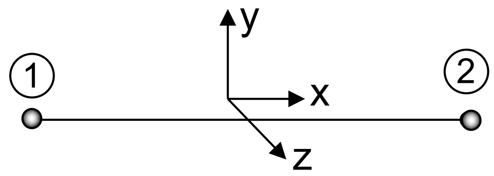
Lecture 4

- Beam Elements
  - Sign conventions
  - Stiffness matrix
  - Example
- Symmetry

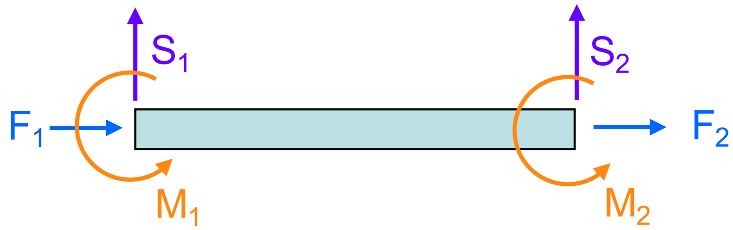
2D Beam Element

Definition: A line element defined in one plane which resists axial forces, **in-plane transverse forces + bending moments**

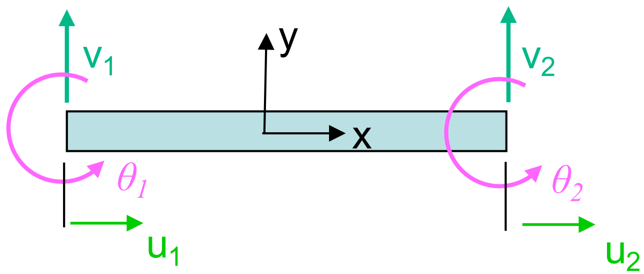
Local (element) axes



Sign Convention:
External forces + moments

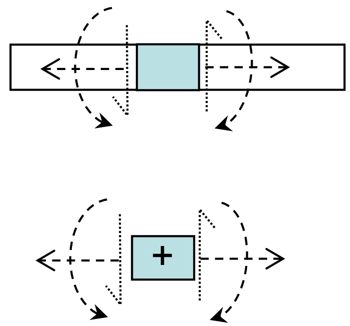


Displacements
(RH axes system)

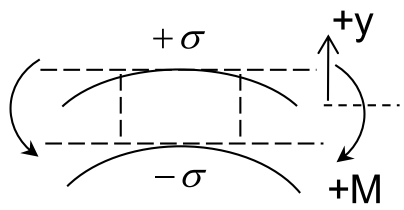


Sign Conventions Cont.

Internal forces and moments
(Deformation sign convention)

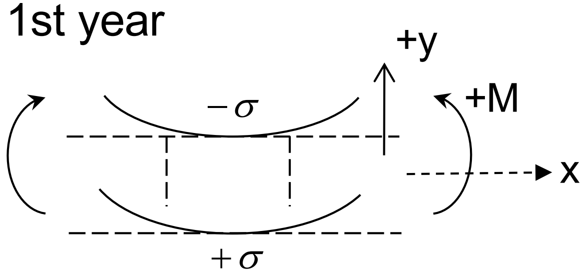


"hogging"



i.e. $\sigma = \dfrac{My}{I}$

Note different from 1st year notes:
i.e. 1st year



"sagging"

The choice of sign conventions is essentially arbitrary. But must be consistent.

Beam force-displacement / Moment-rotation matrix equation

$$\begin{array}{l} (F_1) \\ \\ (F_2) \\ \\ \end{array} \begin{Bmatrix} S_1 \\ M_1 \\ S_2 \\ M_2 \end{Bmatrix} = \begin{bmatrix} & & \\ & k_{ij} & \\ & & \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix} \begin{array}{l} (u_1) \\ \\ (u_2) \\ \\ \end{array}$$

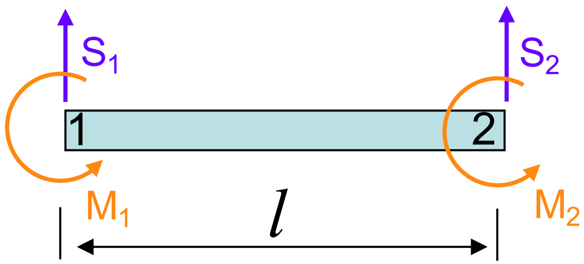
Note: for simplicity axial forces and displacements are omitted

(Axial forces and displacements F, u, can be allowed for by expanding the matrix equation set and treating as for bar examples).

Apply equilibrium, constitutive and compatibility arguments to find form of beam stiffness matrix $[k_{ij}]$

Equilibrium

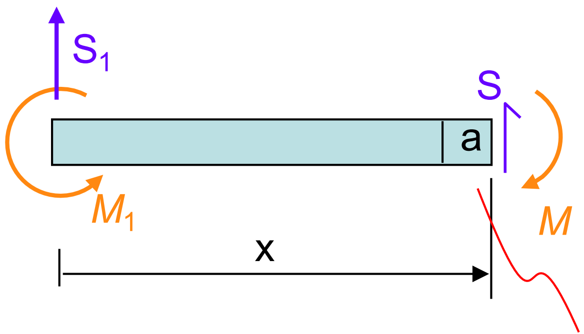
Force/Moment equations of statics



Moments: $\sum (1) +$ $\quad M_1 + M_2 + S_2 l = 0$ (32)

Forces: $\sum \uparrow +$ $\quad S_1 + S_2 = 0$ (33)

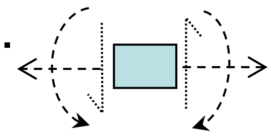
Consider partial FBD:



$$\sum (a) + \; : \; M_1 - S_1 x - M = 0$$

$$-M = S_1 x - M_1$$

i.e.



Constitutive Relation

Linear force-disp / moment-rotation relation

i.e. beam bending-curvature equation

$$M = -EI\frac{d^2v}{dx^2}$$

(neglecting shear deformation)

-ve sign here for "hogging" sign convention

M $\longrightarrow$ $$EI\frac{d^2v}{dx^2} = S_1 x - M_1 \tag{34.1}$$

θ $\longrightarrow$ $$EI\frac{dv}{dx} = S_1\frac{x^2}{2} - M_1 x + c_1 \tag{34.2}$$

v $\longrightarrow$ $$EIv = S_1\frac{x^3}{6} - M_1\frac{x^2}{2} + c_1 x + c_2 \tag{34.3}$$

Compatibility (Geometry of deformation)

- At node 1 $x = 0,\ v = v_1,\ \theta = \theta_1$

Position not displacement

$$(34.2) \longrightarrow EI\theta_1 = c_1 \qquad (36.1)$$

$$(34.3) \longrightarrow EIv_1 = c_2 \qquad (36.2)$$

- At node 2 $x = l,\ v = v_2,\ \theta = \theta_2$

$$(34.2) \longrightarrow EI\theta_2 = \frac{S_1 l^2}{2} - M_1 l + c_1 \qquad (36.3)$$

$$(34.3) \longrightarrow EIv_2 = \frac{S_1 l^3}{6} - \frac{M_1 l^2}{2} + c_1 l + c_2 \qquad (36.4)$$

- i.e. 6 equations: (32), (33), (36.1-4)
  - 6 "unknowns": $S_1,\ M_1,\ S_2,\ M_2,\ c_1,\ c_2$
  - To be found in terms of $v_1,\ \theta_1,\ v_2,\ \theta_2$

Assembling Stiffness Matrix

Manipulating equations and writing in matrix form: $\begin{Bmatrix} S_1 \\ M_1 \\ S_2 \\ M_2 \end{Bmatrix} = \begin{bmatrix} & & & \\ & k_{ij} & & \\ & & & \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix}$

e.g. obtaining S_1 expression for row 1:

$$\frac{l}{2}(36.3) - (36.4)$$ i.e. eliminating M_1

$$\frac{l}{2} EI\theta_2 - EIv_2 = S_1 l^3 \left(\frac{1}{4} - \frac{1}{6}\right) - 0 - \frac{l}{2} c_1 - c_2$$

From 36.1 and 36.2 i.e. $c_1 = EI\theta_1$, $c_2 = EIv_1$

$$\frac{S_1 l^3}{12} = EIv_1 + \frac{l}{2} EI\theta_1 - EIv_2 + \frac{l}{2} EI\theta_2$$

$$S_1 = \frac{6EI}{l^2}\left(\frac{2}{l} v_1 + \theta_1 - \frac{2}{l} v_2 + \theta_2\right)$$

i.e. row 1 of matrix equation set etc.

Beam Stiffness Matrix

$$\begin{Bmatrix} S_1 \\ M_1 \\ S_2 \\ M_2 \end{Bmatrix} = \frac{6EI}{l^2} \underbrace{\begin{bmatrix} 2/l & 1 & -2/l & 1 \\ 1 & 2l/3 & -1 & l/3 \\ -2/l & -1 & 2/l & -1 \\ 1 & l/3 & -1 & 2l/3 \end{bmatrix}} \begin{Bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{Bmatrix} \qquad (37)$$

Only for transverse force-disp / Moment-rotation

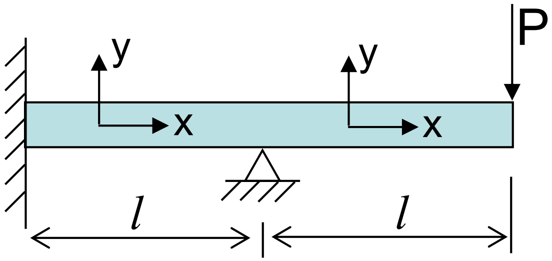
Built-up structures of beams may be analysed and solved in a similar way to previous bar structures

i.e. by

- Transforming
- Assembling
- Applying boundary conditions

Example 3

Coincident two-beam element structure



Both beams:

Same-uniform EI

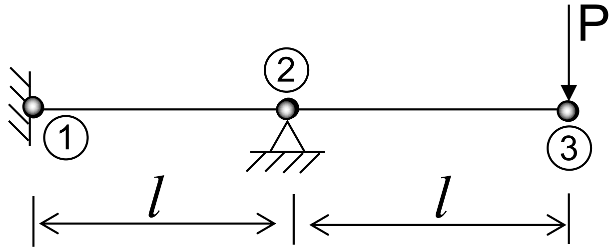
Given: beam load-displacement equation:

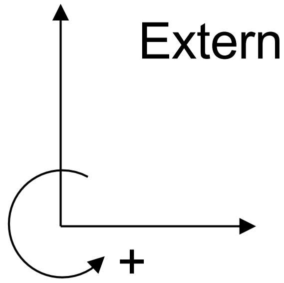
$$\begin{Bmatrix} S_1 l \\ M_1 \\ S_2 l \\ M_2 \end{Bmatrix} = \frac{6EI}{l^2} \begin{bmatrix} 2 & 1 & -2 & 1 \\ 1 & 2/3 & -1 & 1/3 \\ -2 & -1 & 2 & -1 \\ 1 & 1/3 & -1 & 2/3 \end{bmatrix} \begin{Bmatrix} v_1 \\ \theta_1 l \\ v_2 \\ \theta_2 l \end{Bmatrix} \quad (38)$$

i.e. (37) where l is eliminated from beam element stiffness matrix – by inclusion in {force, Mmt} and {Disp, Rotn} vectors

Requirement: to find tip deflection and draw SF & BM diagrams indicating principal values

FE Model





External sign conv.

Writing load displacement equation for each beam element:

$$\begin{Bmatrix} S_1 l \\ M_1 \\ S_2 l \\ M_2 \end{Bmatrix} = \frac{6EI}{l^2}\begin{bmatrix} 2 & 1 & -2 & 1 \\ 1 & 2/3 & -1 & 1/3 \\ -2 & -1 & 2 & -1 \\ 1 & 1/3 & -1 & 2/3 \end{bmatrix}\begin{Bmatrix} v_1 \\ \theta_1 l \\ v_2 \\ \theta_2 l \end{Bmatrix} \quad (38)$$

Beam ① - ②

$$\begin{Bmatrix} S_2 l \\ M_2 \\ S_3 l \\ M_3 \end{Bmatrix} = \frac{6EI}{l^2}\begin{bmatrix} 2 & 1 & -2 & 1 \\ 1 & 2/3 & -1 & 1/3 \\ -2 & -1 & 2 & -1 \\ 1 & 1/3 & -1 & 2/3 \end{bmatrix}\begin{Bmatrix} v_2 \\ \theta_2 l \\ v_3 \\ \theta_3 l \end{Bmatrix} \quad (39)$$

Beam ② - ③

Expanding and assembling stiffness matrixes for beam structure:

$$
\begin{array}{r}
\\ \\ \\ =0 \\ =-Pl \\ =0
\end{array}
\begin{Bmatrix} S_1 l \\ M_1 \\ S_2 l \\ \cancel{M_2} \\ S_3 l \\ \cancel{M_3} \end{Bmatrix}
= \frac{6EI}{l^2}
\begin{bmatrix}
2 & 1 & -2 & 1 & 0 & 0 \\
1 & 2/3 & -1 & 1/3 & 0 & 0 \\
-2 & -1 & 2+2 & -1+1 & -2 & 1 \\
1 & 1/3 & -1+1 & 2/3+2/3 & -1 & 1/3 \\
0 & 0 & -2 & -1 & 2 & -1 \\
0 & 0 & 1 & 1/3 & -1 & 2/3
\end{bmatrix}
\begin{Bmatrix} \cancel{v_1} \\ \cancel{\theta_1 l} \\ \cancel{v_2} \\ \theta_2 l \\ v_3 \\ \theta_3 l \end{Bmatrix}
\begin{array}{l}
=0 \\ =0 \\ =0 \\ \\ \\ \\
\end{array}
\qquad (40)
\qquad
\begin{array}{r}
(40.1) \\ (40.2) \\ (40.3) \\ (40.4) \\ (40.5) \\ (40.6)
\end{array}
$$

Applying boundary conditions:

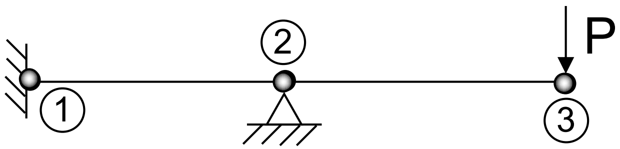
- Displacements & Rotations: @ ① $v_1 = 0, \quad \theta_1 = 0$ fully fixed

  @ ② $v_2 = 0$ pinned

- Forces & Moments: @ ② $M_2 = 0$ no applied or reacted mmt

  @ ③ $S_3 = -P$ applied load

  $M_3 = 0$ no applied or reacted mmt



Solving:

- Using equations from last three rows

$$0 = \frac{6EI}{l^2}\left(\frac{4}{3}\theta_2 l - v_3 + \frac{1}{3}\theta_3 l\right) \qquad (40.4)$$

$$-Pl = \frac{6EI}{l^2}\left(\theta_2 l + 2v_3 + \theta_3 l\right) \qquad (40.5)$$

$$0 = \frac{6EI}{l^2}\left(\frac{1}{3}\theta_2 l - v_3 + \frac{2}{3}\theta_3 l\right) \qquad (40.6)$$

- Eliminating v_3 from 40.4 and 40.5 gives:

$$-Pl = \frac{6EI}{l^2}\left(\frac{5}{3}\theta_2 l - \frac{1}{3}\theta_3 l\right) \qquad (41)$$

- Eliminating v_3 from 40.5 and 40.6 gives:

$$-Pl = \frac{6EI}{l^2}\left(-\frac{1}{3}\theta_2 l + \frac{1}{3}\theta_3 l\right) \qquad (42)$$

Solving for unknown displacements and rotations:

(41) + (42) $\longrightarrow$ $$\theta_2 = \frac{-Pl^2}{4EI}$$

(42) $\longrightarrow$ $$\theta_3 = \frac{-3Pl^2}{4EI}$$

(40.4) $\longrightarrow$ $$v_3 = \frac{-7}{12}\frac{Pl^3}{EI}$$

Solving for unknown forces (reactions): - External forces + mmts

(40.1) $\longrightarrow$ $$S_1 = -\frac{3}{2}P$$

(40.2) $\longrightarrow$ $$M_1 = -\frac{Pl}{2}$$

(40.3) $\longrightarrow$ $$S_2 = \frac{5}{2}P$$

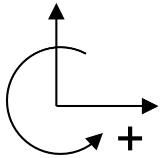
Also $M_2 = 0$

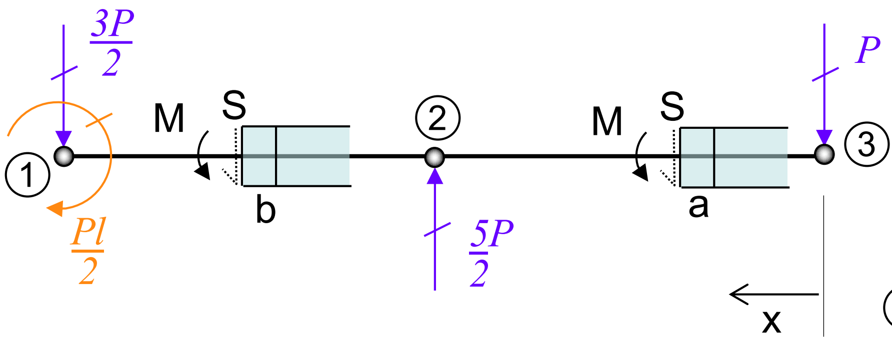
And $S_3 = -P \quad M_3 = 0$

Known boundary conditions

Shear force and Bending Moment Diagrams

Ext. FBD:





Using sections from RHS

②-③ $\Sigma\uparrow^{+}_{a}$

$$-S-P=0$$

$$S=-P$$

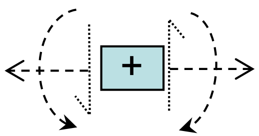
①-② $\Sigma\uparrow^{+}_{b}$

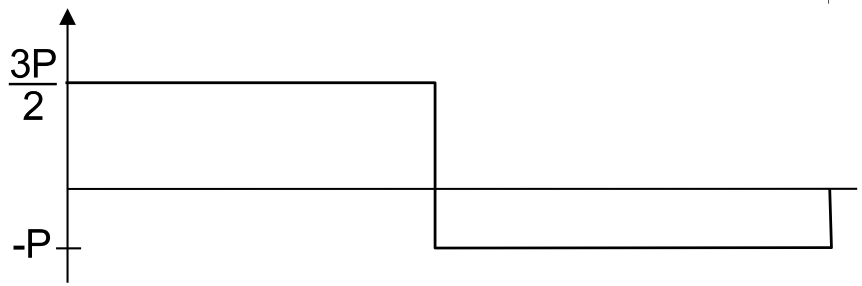
$$-S+\tfrac{5}{2}P-P=0$$

$$S=\tfrac{3}{2}P$$

Shear Force

Int. :





②-③ $\Sigma\circlearrowright^{+}_{a}$

$$M-Px=0 \qquad @③\ x=0 \qquad @②\ x=l$$

$$M=Px \qquad =0 \qquad =Pl$$

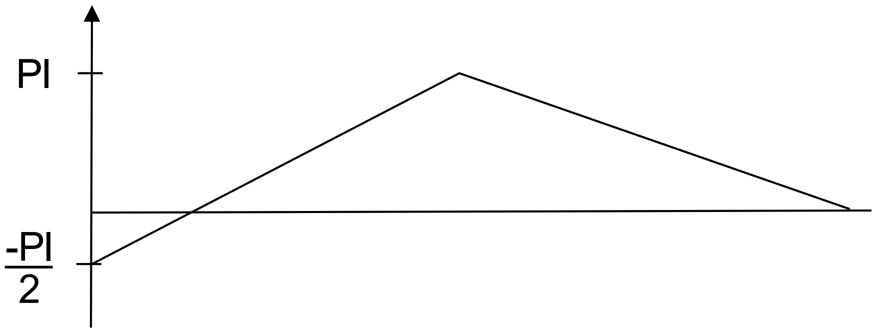
①-② $\Sigma\circlearrowright^{+}_{b}$

$$M+\tfrac{5}{2}P(x-l)-Px=0$$

$$M=Px-\tfrac{5}{2}P(x-l)$$

$$=Pl \quad @②\ x=l \qquad =-\frac{Pl}{2} \quad @①\ x=2l$$

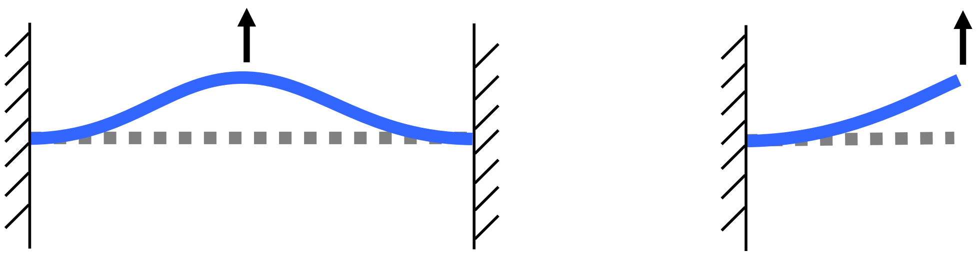
Bending Moment



- Note these are plotted for <u>Internal</u> forces + mmts

Symmetry

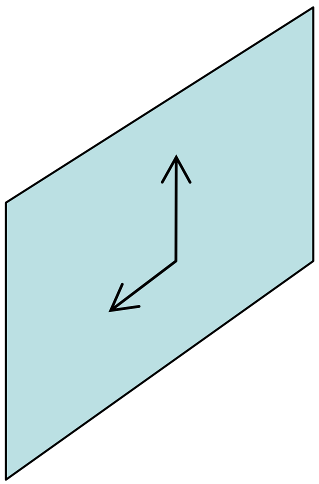
- Symmetry in a model can be used to reduce complexity and overall model size
- Symmetry needs to exist in structure and loading



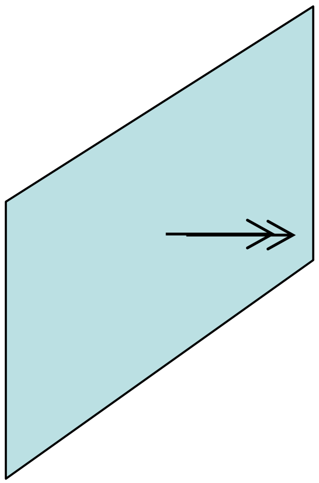
- Consider the degrees of freedom at the point of loading in each case

Symmetry

- For symmetry:-
  - Translations have no component normal to the plane of symmetry
  - Rotations have no component parallel to the plane of symmetry

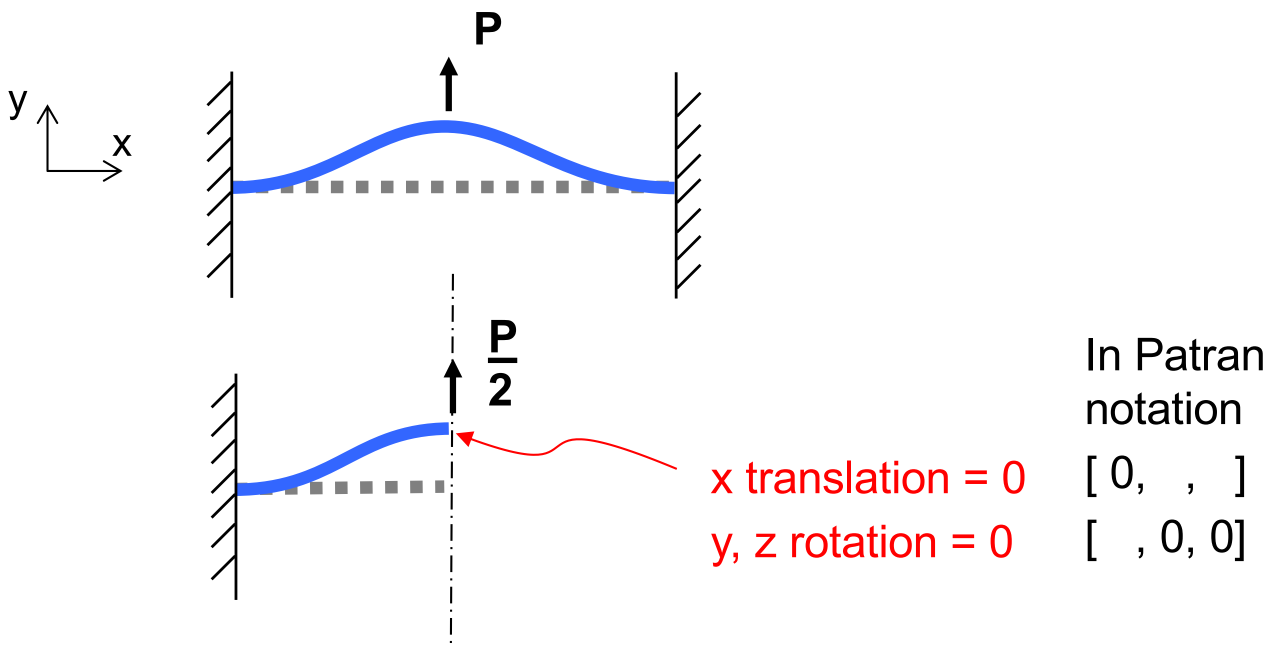


Permitted translations

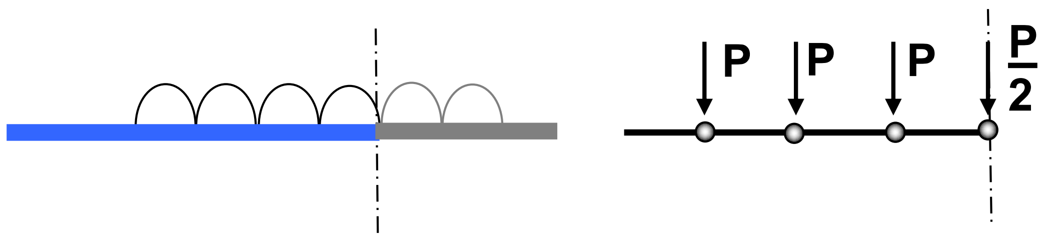


Permitted rotations

Symmetry



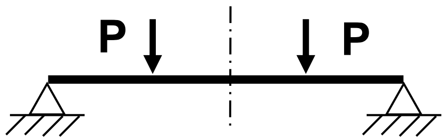
- Note: you need to take half of any loads that are applied at the centre line



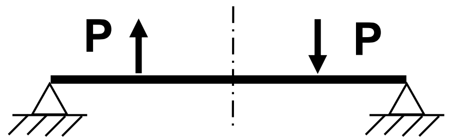
Antisymmetry

- Can also model antisymetric loading on a symmetric structure

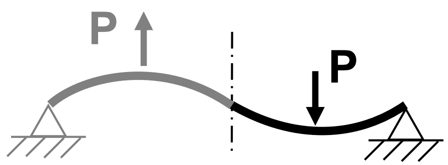
  i.e. structure is symmetric but loads are reversed
- Useful for the case of torsional loading



Symmetric

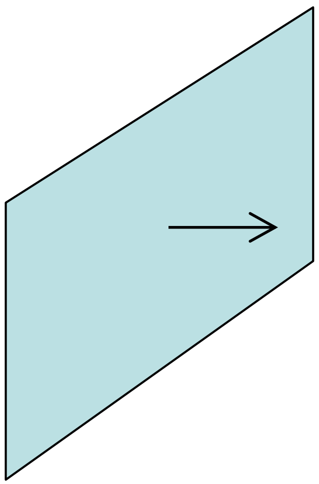


Antisymmetric

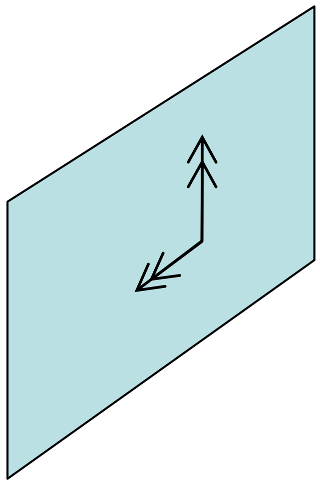


Antisymmetry

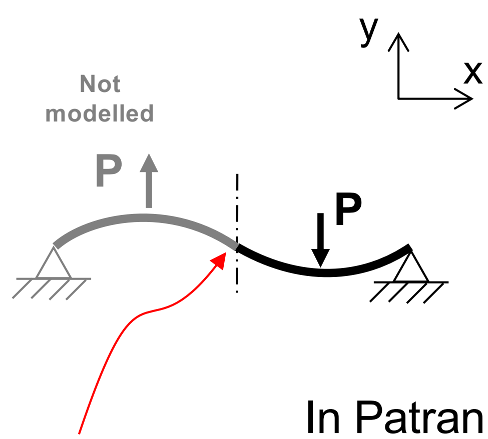
- For antisymmetry:-
  - Translations have no component parallel to the plane of symmetry
  - Rotations have no component normal to the plane of symmetry



Permitted translations



Permitted rotations

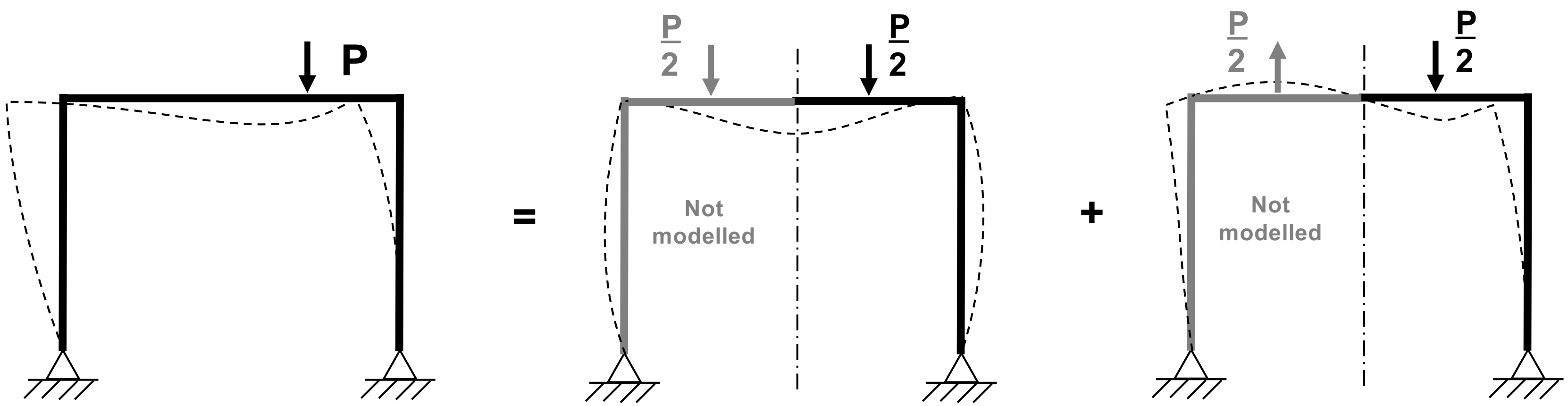


y, z translation = 0
x rotation = 0

In Patran notation
[, 0 , 0]
[0, ,]

Combined loading

- Can consider loading as the sum of symmetric and antisymmetric parts
- Separate load cases can be defined in the analysis and then final solution taken as the linear super-position of two load cases



Examples Sheets

- Available on Blackboard for Bars and Beams
  - Course Documents > StM2 Structures - Finite Element Analysis > FE Lecture notes
- Solutions provided

Structures and Materials 3

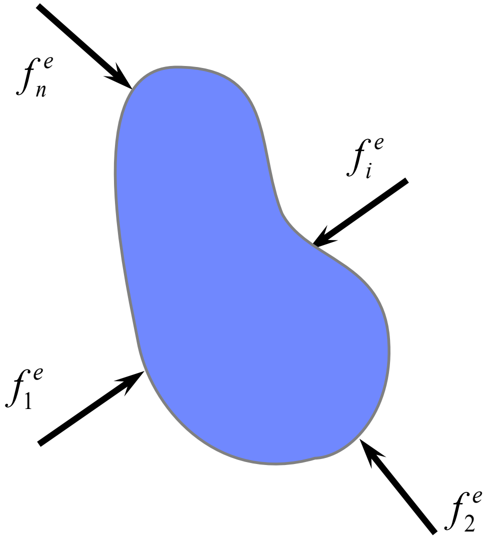
Triangular and Quadrilateral Elements

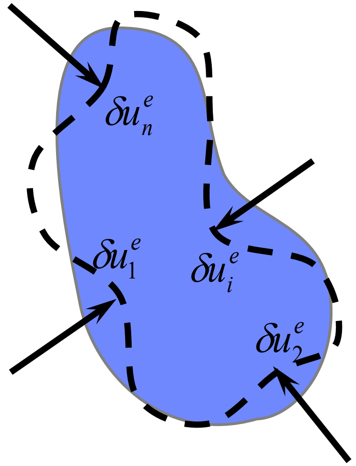
Dr Giuliano Allegri

Lecture 5

- Principal of Virtual Work
- Derivation of Triangular Element Stiffness Matrix
- Example
- Application to Rectangular Element

Principle of Virtual Work





System in equilibrium

$$\sum f^e \cdot \delta u^e = \int_V \sigma \cdot \delta\varepsilon \, dV$$

Virtual deformations

- A set of external forces f is in equilibrium causing internal stresses.
- Points at which the forces acting are given virtual displacements (i.e. fictitious, mathematical displacements).
- These virtual displacements will result in geometrically compatible deformations with virtual strains.
- The ***principle of virtual work*** states that for any system in equilibrium, the external virtual work must be equal to the internal virtual work. Hence a real system of forces and stresses is coupled with a virtual system of displacements and strains.

Derivation of Element Stiffness Matrix

We are looking to solve **{F} = [K] {U}**

Consider an element in equilibrium with displacements $\{u_n^e\}$, strains $\{\varepsilon\}$ and stresses $\{\sigma\}$

The stress-strain equation is:

$$\{\sigma\} = [D]\{\varepsilon\} \qquad 3.1$$

The strain-displacement relation is:

$$\{\varepsilon\} = [B]\{u_n^e\} \qquad 3.2$$

The evaluation of $[B]$ will be discussed later

$$\therefore \{\sigma\} = [D][B]\{u_n^e\} \qquad 3.3$$

Using Virtual Work

Now consider virtual nodal displacements $\{\delta u_n{}^e\}$

From 3.2 the resulting virtual strains are given by

$\{\delta\varepsilon\} = [B]\ \{\delta u_n{}^e\}$ 3.4

The internal work done by the virtual displacements $\{\delta u_n{}^e\}$ is

$$\int_V \{\delta\varepsilon\}^T \{\sigma\}\, dV$$

Using 3.3 and 3.4

$$= \int_V \{[B]\ \{\delta u_n^e\}\}^T\ [D][B]\{u_n^e\}\, dV \qquad 3.5$$

Tidying it up

$$= \int_V \{\delta u_n^e\}^T [B]^T [D][B]\{u_n^e\}\, dV \qquad 3.6$$

Derivation of Element Stiffness Matrix using Virtual Work

The quantities $\{u_n^e\}$ and $\{\delta u_n^e\}$ are at the nodes and do not vary with position.

They may therefore be moved out of the integral sign

$$Virtual\ internal\ work = \left\{\delta u_n^e\right\}^T \left\{u_n^e\right\} \int_V [B]^T[D][B]\, dV \qquad 3.7$$

Now the external work done by the forces at the nodes $\{f^e\}$, in imposing displacements $\{\delta u_n^e\}$

$$External\ work = \left\{\delta u_n^e\right\}^T \left\{f^e\right\} \qquad 3.8$$

Derivation of Element Stiffness Matrix using Virtual Work

For equilibrium internal and external work must be equal. Equating 3.7 and 3.8 gives

$$\cancel{\left\{ \delta u_n^e \right\}}^T \left\{ f^e \right\} = \cancel{\left\{ \delta u_n^e \right\}}^T \int_V [B]^T [D][B] \, dV \left\{ u_n^e \right\} \qquad 3.9$$

Now $\left\{ f^e \right\} = \left[K^e \right] \left\{ u_n^e \right\}$ 3.10

Where $[K^e]$ is the element stiffness matrix

so $\left[K^e \right] = \int_V [B]^T [D][B] \, dV$ 3.11

To form the element stiffness matrices we need the stress-strain matrix $[D]$

$\{\sigma\} = [D]\{\varepsilon\}$ 3.1

Stress-Strain Matrix for Plane Stress

Assume plane stress i.e. $\sigma_z = 0$, $\varepsilon_z \neq 0$

Hooke's Law:

$$\left.\begin{aligned} \varepsilon_x &= \sigma_x /E - \nu\sigma_y /E \\ \varepsilon_y &= \sigma_y /E - \nu\sigma_x /E \\ \gamma_{xy} &= \tau_{xy}/G \\ \text{and} \quad G &= E/2(1+\nu) \end{aligned}\right\} \text{Derived in 2nd year notes}$$

These equations can be solved to give

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = E/\left(1-\nu^2\right) \begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} \qquad 3.12$$

$\{\sigma\}$ $\qquad$ $[D]$ $\qquad$ $\{\varepsilon\}$

Stiffness Matrix for Triangular Membrane Element

Assume a linear variation of displacement over the element:

$$\left.\begin{aligned} u &= \alpha_1 + \alpha_2 x + \alpha_3 y \\ v &= \alpha_4 + \alpha_5 x + \alpha_6 y \end{aligned}\right\} \quad 3.13$$

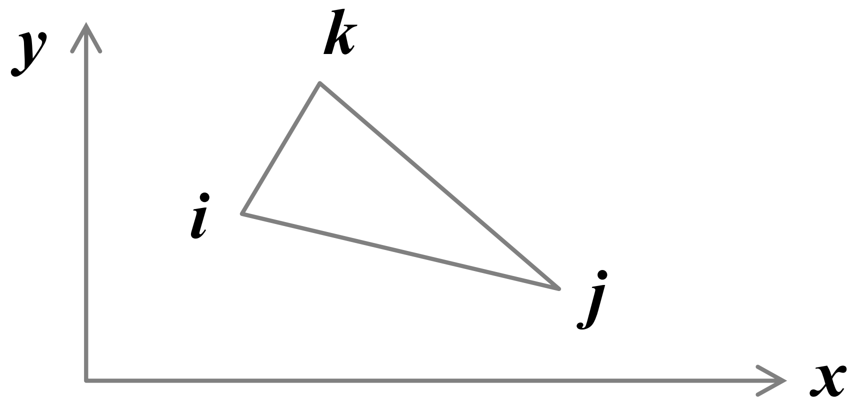
α_1 - α_6 are constants for a given element

and can be calculated from the displacements at the nodes

$$u_i = \alpha_1 + \alpha_2 x_i + \alpha_3 y_i$$

$$v_i = \alpha_4 + \alpha_5 x_i + \alpha_6 y_i$$

$$u_j = etc\ldots$$



Stiffness Matrix for Triangular Membrane Element

or in matrix form:-

$$\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_k \\ v_k \end{Bmatrix} = \begin{bmatrix} 1 & x_i & y_i & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x_i & y_i \\ 1 & x_j & y_j & 0 & 0 & 0 \\ & & & & & \\ & & etc. & & & \\ & & & & & \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix} \qquad 3.14$$

i.e. $\{u_n{}^e\} = [A]\{\alpha\}$ 3.15

$\{\alpha\} = [A]^{-1}\ \{u_n{}^e\}$ 3.16

We now want to evaluate the matrix $[B]$ which relates element strains to nodal displacements

$\{\varepsilon\} = [B]\{u_n{}^e\}$ 3.2

Stiffness Matrix for Triangular Membrane Element

Differentiating 3.13

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y$$
$$v = \alpha_4 + \alpha_5 x + \alpha_6 y$$

$$\frac{\partial u}{\partial x} = \alpha_2 = \varepsilon_x$$

$$\frac{\partial v}{\partial y} = \alpha_6 = \varepsilon_y$$

$$\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \alpha_3 + \alpha_5 = \gamma_{xy}$$

In matrix form

$$\begin{Bmatrix} \varepsilon_x \\ \varepsilon_y \\ \gamma_{xy} \end{Bmatrix} = \begin{bmatrix} 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \\ 0 & 0 & 1 & 0 & 1 & 0 \end{bmatrix} \begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix} \qquad 3.17$$

i.e. $\{\varepsilon\} = [C]\{\alpha\}$ 3.18

Stiffness Matrix for Triangular Membrane Element

Substituting $\{\alpha\}$ from 3.16 gives

$$\{\varepsilon\} = [C][A]^{-1}\{u_n^e\} \qquad 3.19$$

Comparing this with 3.2 :- $\{\varepsilon\} = [B]\{u_n^e\}$ we can write

$$[B] = [C]\,[A]^{-1} \qquad 3.20$$

$[K^e]$ can now be evaluated from the virtual work method

$$\left[K^e\right] = \int_V [B]^T[D][B]\, dV \qquad 3.11$$

Substitute 3.20

$$\left[K^e\right] = \int_V [A]^{-T}[C]^T[D][C]\,[A]^{-1}\, dV \qquad 3.21$$

Stiffness Matrix for Triangular Membrane Element

All the matrices are constant over the element hence

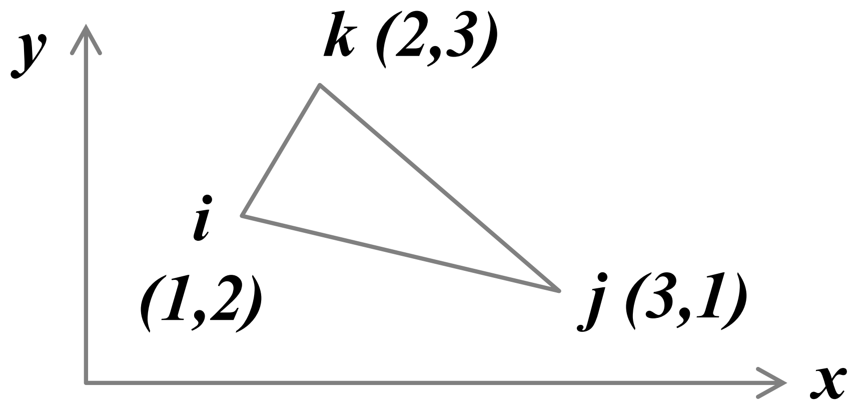
$$\left[K^e\right] = [A]^{-T}[C]^T[D][C][A]^{-1} a^e t \qquad 3.22$$

Assuming constant thickness, t

a^e is the area of the element, determined from Cramer's rule

$$a^e = \tfrac{1}{2}\begin{vmatrix} 1 & x_i & y_i \\ 1 & x_j & y_j \\ 1 & x_k & y_k \end{vmatrix} \qquad 3.23$$

Example: Stiffness Matrix for a Triangular Element



Assume linear variation of displacement

$$
\left.
\begin{aligned}
u &= \alpha_1 + \alpha_2 x + \alpha_3 y \\
v &= \alpha_4 + \alpha_5 x + \alpha_6 y
\end{aligned}
\right\} \quad 3.13
$$

Substituting nodal coordinates into 3.14 (3.13 in matrix form) :-

$$
\begin{Bmatrix} u_i \\ v_i \\ u_j \\ v_j \\ u_k \\ v_k \end{Bmatrix}
=
\begin{bmatrix}
1 & 1 & 2 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 1 & 2 \\
1 & 3 & 1 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 3 & 1 \\
1 & 2 & 3 & 0 & 0 & 0 \\
0 & 0 & 0 & 1 & 2 & 3
\end{bmatrix}
\begin{Bmatrix} \alpha_1 \\ \alpha_2 \\ \alpha_3 \\ \alpha_4 \\ \alpha_5 \\ \alpha_6 \end{Bmatrix}
$$

$$\{u_n^e\} = [A] \{\alpha\}$$

Example: Stiffness Matrix for a Triangular Element

From 3.20 $[B] = [C]\,[A]^{-1}$

$$[A]^{-1} = \frac{1}{3}\begin{bmatrix} 7 & 0 & 1 & 0 & -5 & 0 \\ -2 & 0 & 1 & 0 & 1 & 0 \\ -1 & 0 & -1 & 0 & 2 & 0 \\ 0 & 7 & 0 & 1 & 0 & -5 \\ 0 & -2 & 0 & 1 & 0 & 1 \\ 0 & -1 & 0 & -1 & 0 & 2 \end{bmatrix}$$

$[C]$ has been calculated in 3.17 hence

$$[B] = \frac{1}{3}\begin{bmatrix} -2 & 0 & 1 & 0 & 1 & 0 \\ 0 & -1 & 0 & -1 & 0 & 2 \\ -1 & -2 & -1 & 1 & 2 & 1 \end{bmatrix}$$

Example: Stiffness Matrix for a Triangular Element

For plane stress

$$[D] = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix} \qquad 3.12$$

For aluminium alloy, $E = 7\ X\ 10^4\ N/mm^2,\ \nu = 0.3$

$$[D] = \frac{7\times10^4}{0.91}\begin{bmatrix} 1 & 0.3 & 0 \\ 0.3 & 1 & 0 \\ 0 & 0 & 0.35 \end{bmatrix} = 10^4\begin{bmatrix} 7.69 & 2.31 & 0 \\ 2.31 & 7.69 & 0 \\ 0 & 0 & 2.69 \end{bmatrix}$$

Example: Stiffness Matrix for a Triangular Element

Area of triangular element, $a^e = \frac{1}{2}\begin{vmatrix} 1 & 1 & 2 \\ 1 & 3 & 1 \\ 1 & 2 & 3 \end{vmatrix}$ 3.22

Let t=2/300 $\therefore\ a^e.t = 0.01$

From 3.11 $\left[K^e\right] = \int_V [B]^T[D][B]\,dV = 0.01[B]^T[D][B]$

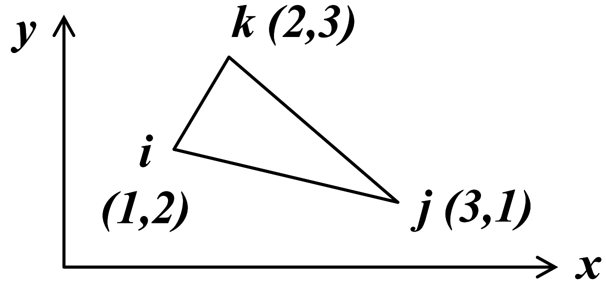
$$[D][B] = 10^4\begin{bmatrix} -5.13 & -0.77 & 2.56 & -0.77 & 2.56 & 1.54 \\ -1.54 & -2.56 & 0.77 & -2.56 & 0.77 & 5.13 \\ -0.90 & -1.79 & -0.90 & 0.90 & 1.79 & 0.90 \end{bmatrix}$$

Hence, $\left[K^e\right] = \begin{bmatrix} 372 & 111 & -141 & 21 & -231 & -133 \\ & 205 & 34 & 26 & -145 & -231 \\ & & 115 & -56 & 26 & 21 \\ & & & 115 & 34 & -141 \\ & sym. & & & 205 & 111 \\ & & & & & 423 \end{bmatrix}$

Calculation of Stresses

If nodes i and j are grounded (i.e. $u_i = v_i = u_j = v_j = 0$) and node k is perturbed by 0.01mm in the x direction and 0.03mm in the y direction (i.e. $u_k = 0.01$ and $v_k = 0.03$) find the element stresses

$$u_i = \alpha_1 + \alpha_2 x_i + \alpha_3 y_i$$
$$u_j = \alpha_1 + \alpha_2 x_j + \alpha_3 y_j$$
$$u_k = \alpha_1 + \alpha_2 x_k + \alpha_3 y_k$$



$$0 = \alpha_1 + \alpha_2 + 2\alpha_3$$
$$0 = \alpha_1 + 3\alpha_2 + \alpha_3$$
$$0.01 = \alpha_1 + 2\alpha_2 + 3\alpha_3$$

$$\alpha_1 = \tfrac{-5}{300}, \alpha_2 = \tfrac{1}{300}, \alpha_3 = \tfrac{2}{300}$$

i.e. $$u = \frac{-5}{300} + \frac{x}{300} + \frac{2y}{300}$$

Calculation of Stresses

Similarly substituting for v

$$0 = \alpha_4 + \alpha_5 + 2\alpha_6$$

$$0 = \alpha_4 + 3\alpha_5 + \alpha_6$$

$$0.03 = \alpha_4 + 2\alpha_5 + 3\alpha_6$$

$$\alpha_4 = \tfrac{-5}{100}, \alpha_5 = \tfrac{1}{100}, \alpha_6 = \tfrac{2}{100}$$

i.e. $$v = \frac{-5}{100} + \frac{x}{100} + \frac{2y}{100}$$

$$\varepsilon_x = \tfrac{\partial u}{\partial x} = \frac{1}{300}$$

$$\varepsilon_y = \tfrac{\partial v}{\partial y} = \frac{2}{100}$$

$$\gamma_{xy} = \tfrac{\partial u}{\partial y} + \tfrac{\partial v}{\partial x} = \frac{2}{300} + \frac{1}{100} = \frac{5}{300}$$

$$\{\varepsilon\} = \begin{Bmatrix} \frac{1}{300} \\ \frac{1}{50} \\ \frac{1}{60} \end{Bmatrix}$$

$$\{\sigma\} = [D]\{\varepsilon\} = 10^4 \begin{bmatrix} 7.69 & 2.31 & 0 \\ 2.31 & 7.69 & 0 \\ 0 & 0 & 2.69 \end{bmatrix} \begin{Bmatrix} \frac{1}{300} \\ \frac{1}{50} \\ \frac{1}{60} \end{Bmatrix} \qquad \begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} 718 \\ 1615 \\ 448 \end{Bmatrix}$$

Calculation of Stresses

Alternatively $\{\sigma\}$ can be calculated directly from displacements

$$\{\sigma\} = [D][B]\{u_n^e\} \qquad 3.3$$

$$\{\sigma\} = 10^4 \begin{bmatrix} -5.13 & -0.77 & 2.56 & -0.77 & 2.56 & 1.54 \\ -1.54 & -2.56 & 0.77 & -2.56 & 0.77 & 5.13 \\ -0.90 & -1.79 & -0.90 & 0.90 & 1.79 & 0.90 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0.01 \\ 0.03 \end{Bmatrix}$$

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} 718 \\ 1616 \\ 449 \end{Bmatrix}$$

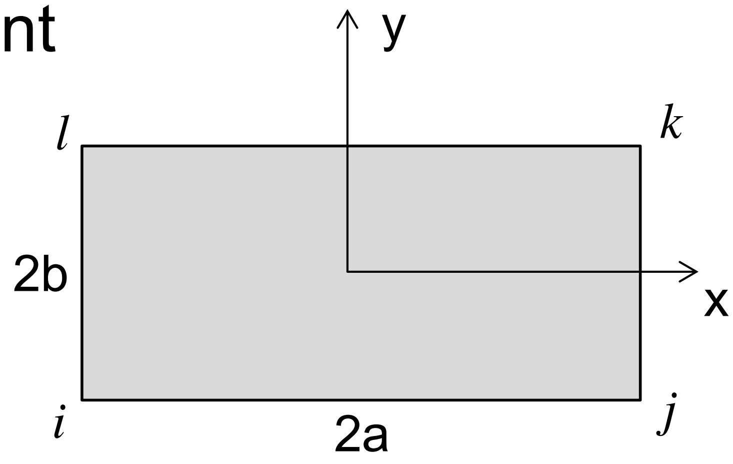
i.e. as before except for rounding errors

Rectangular Membrane Element

Assume linear variation of displacement

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 x y$$

$$v = \alpha_5 + \alpha_6 x + \alpha_7 y + \alpha_8 x y \qquad 3.24$$



As for triangle, substitute values of u and v at the nodes

$$\{u_n^e\} = [A]\{\alpha\} \qquad 3.15$$

i.e.

$$\{U_n^e\} = \begin{bmatrix} 1 & -a & -b & ab & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & -a & -b & ab \\ 1 & a & -b & -ab & 0 & 0 & 0 & 0 \\ & etc... & & & & & & \end{bmatrix} \{\alpha\} \qquad 3.25$$

Rectangular Membrane Element

So as before:

$\{\alpha\} = [A]^{-1}\ \{u_n^{e}\}$ 3.16

Differentiating 3.24 to get strains:

$$\frac{\partial u}{\partial x} = \alpha_2 + \alpha_4\, y = \varepsilon_x \qquad \frac{\partial v}{\partial y} = \alpha_7 + \alpha_8\, x = \varepsilon_y$$

$$\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \alpha_3 + \alpha_4\, x + \alpha_6 + \alpha_8\, y = \gamma_{xy}$$

i.e.

$$\{\varepsilon\} = \begin{bmatrix} 0 & 1 & 0 & y & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x \\ 0 & 0 & 1 & x & 0 & 1 & 0 & y \end{bmatrix} \{\alpha\} \qquad 3.26$$

Or as before

$\{\varepsilon\} = [C]\{\alpha\}$ 3.18

Rectangular Membrane Element

Substituting for $\{\alpha\}$ in 3.18 gives

$$\{\varepsilon\} = [C][A]^{-1}\{U^e_n\} \qquad 3.19$$

$$\therefore \quad [B] = [C][A]^{-1} \qquad 3.20$$

i.e. as for the triangle, but with [A] and [C] as given by 3.25 and 3.26

$$\left[K^e\right] = \int_V [B]^T[D][B]\, dV \qquad 3.11$$

[D] is as before; 3.12 for plane stress

[B] however is no longer constant since [C] contains x and y

With constant thickness t, 3.11 can be written as

$$\left[K^e\right] = t \int_{-b}^{b} \int_{-a}^{a} [B]^T [D][B]\, dx\, dy \qquad 3.27$$

Structures and Materials 3

Convergence and Meshing Quality

Dr Giuliano Allegri

Lecture 6

- Convergence Criteria
- Mesh Quality
- BC’s and symmetry
- Example

Mesh Convergence Criteria

Beam elements without shear flexibility match bending theory exactly for the case of constant shear force and linear bending moment, i.e. cantilever loading

Increasing the number of elements will have no effect on results.

In the general case results improve with increasing number of elements because the assumed displacement field is only an approximation.

The results should converge to the exact solution as the mesh is refined if :

1. Rigid body motion of the element does not induce strains.
2. The element can represent any state of constant strain.
3. The displacement functions are such that the strains at the interfaces between elements are finite, but displacements are continuous across elements.

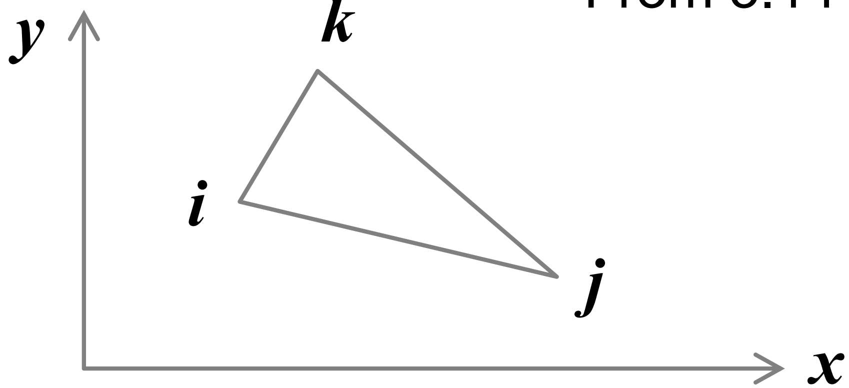
Convergence Criteria

For the triangular membrane element it is apparent that 2 and 3 are met.

Recall the assumed linear displacement functions (3.13) :-

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y \qquad\qquad v = \alpha_4 + \alpha_5 x + \alpha_6 y$$

To demonstrate that 1 is met, consider the particular case where $\alpha_2 = \alpha_6 = 0$ and $\alpha_5 = -\alpha_3$



From 3.14

$$\left\{u_n^e\right\} = \begin{Bmatrix} \alpha_1 + \alpha_3 y_i \\ \alpha_4 - \alpha_3 x_i \\ \alpha_1 + \alpha_3 y_j \\ \alpha_4 - \alpha_3 x_j \\ \alpha_1 + \alpha_3 y_k \\ \alpha_4 - \alpha_3 x_k \end{Bmatrix} \qquad (4.1)$$

Hence: $u_i = u_j = u_k$
$v_i = v_j = v_k$

Convergence Criteria

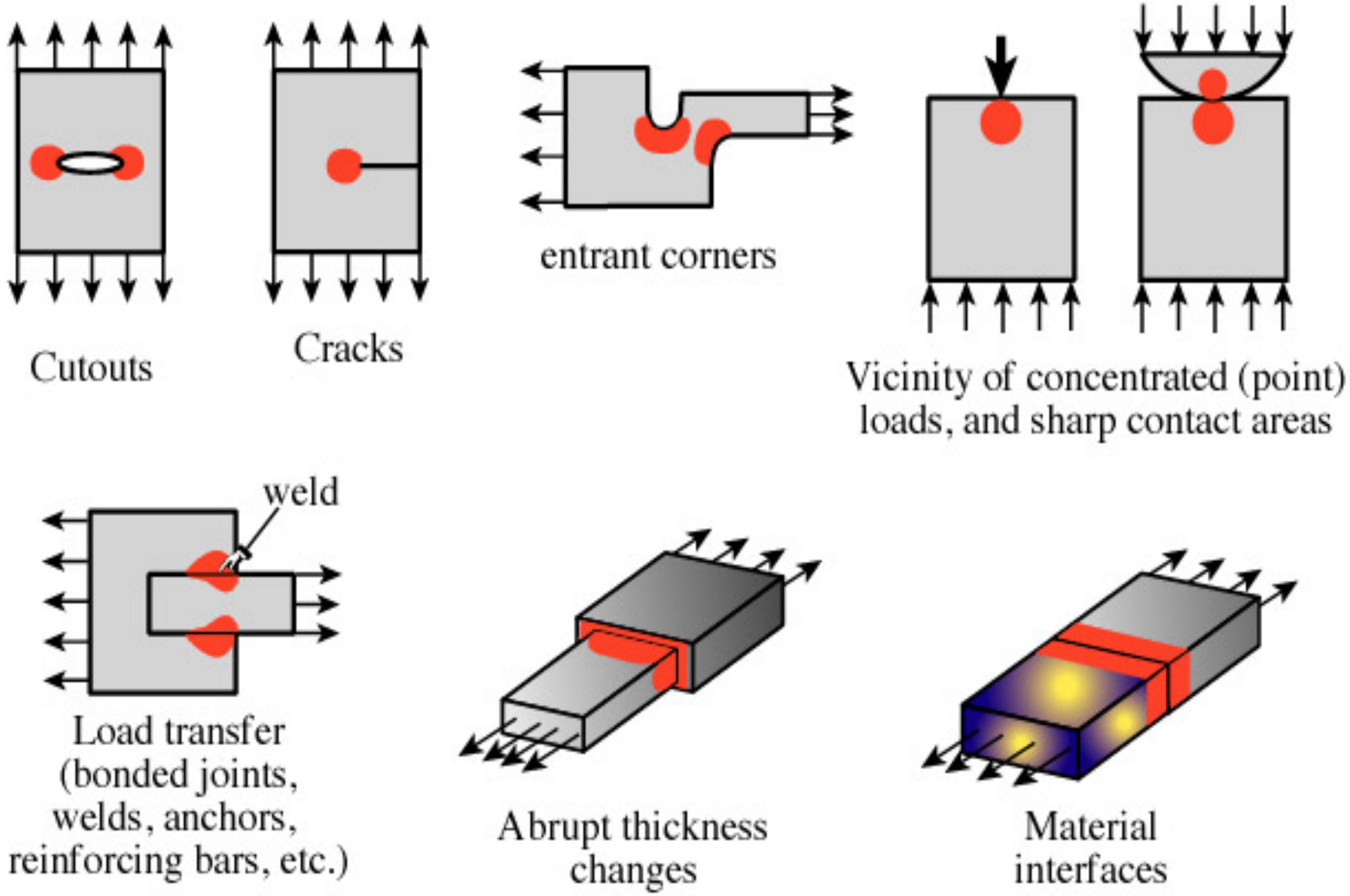
Differentiating displacement functions (3.13) gives us strain

$$\frac{\partial u}{\partial x} = \alpha_2 = \varepsilon_x \qquad \frac{\partial v}{\partial y} = \alpha_6 = \varepsilon_y \qquad \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} = \alpha_3 + \alpha_5 = \gamma_{xy}$$

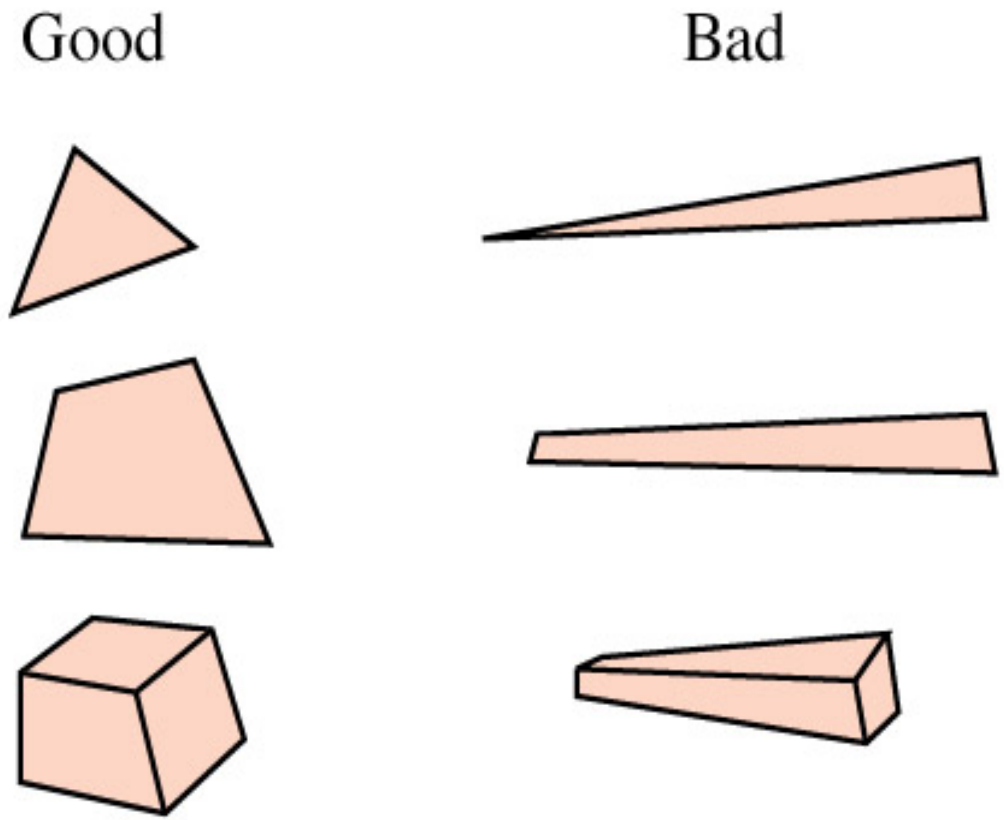
But $\alpha_2 = \alpha_6 = 0$ and $\alpha_5 = -\alpha_3$

Hence: $\varepsilon_x = 0$; $\varepsilon_y = 0$; $\gamma_{xy} = 0$

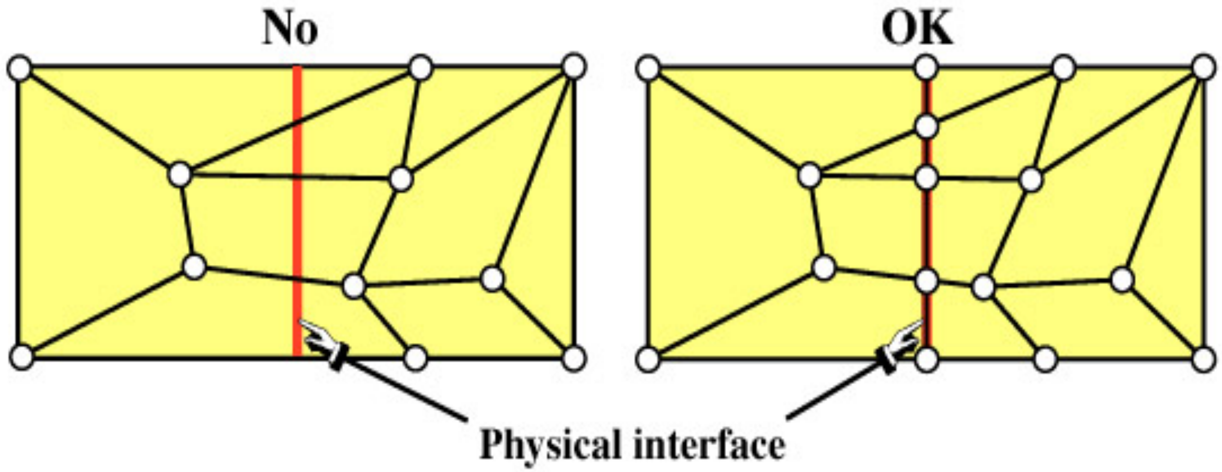
Where Finer Meshes Should be Used



Avoid elements with bad aspect ratio



Elements must not cross interfaces



Element Geometry Preferences

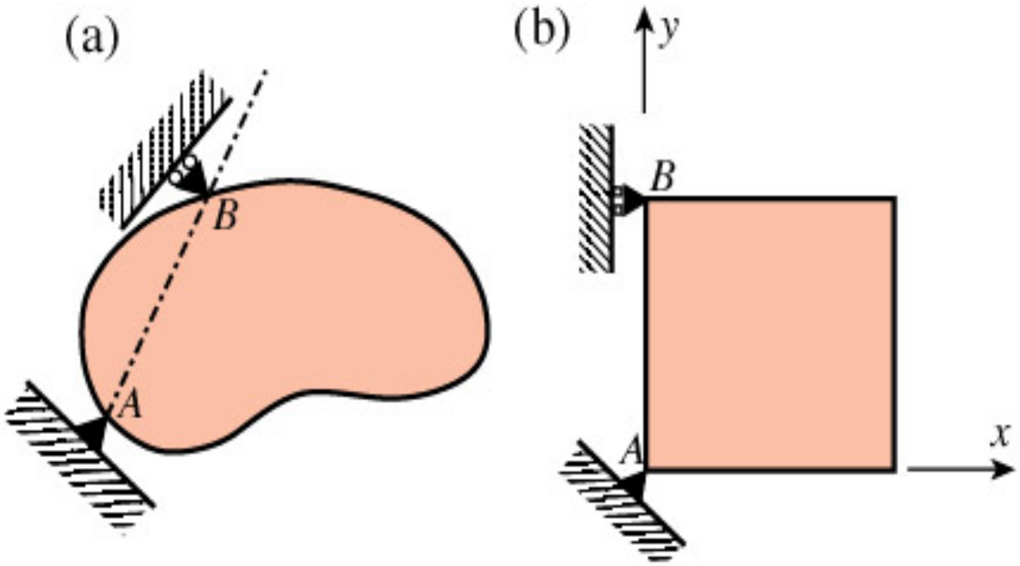
Other things being equal, prefer

in 2D: Quadrilaterals over Triangles

in 3D: Bricks over Wedges
Wedges over Tetrahedra

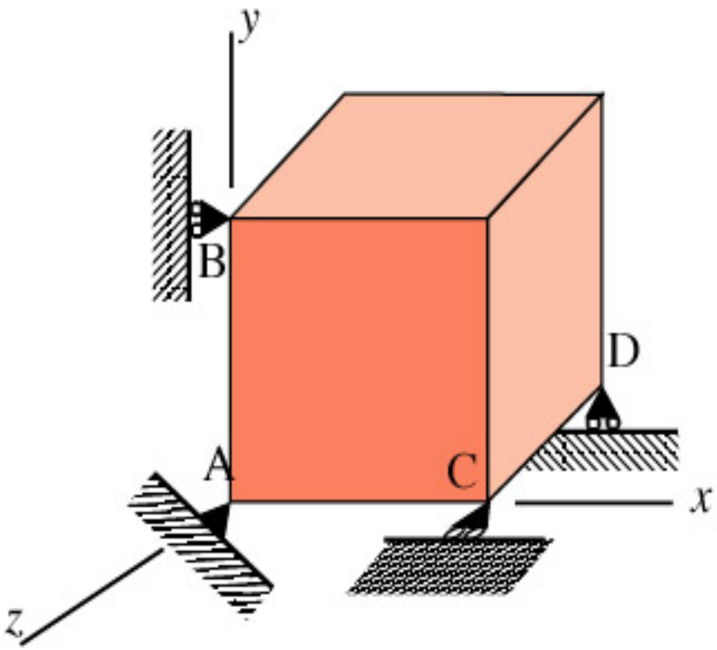
BC's must suppress Rigid Body Motion

2D



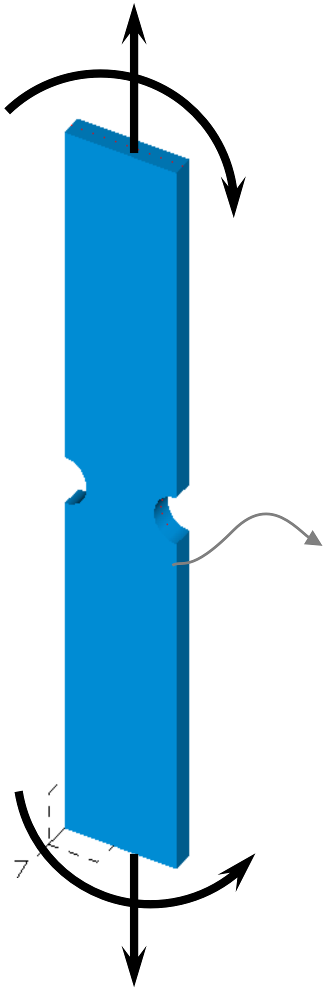
BC's must suppress Rigid Body Motion

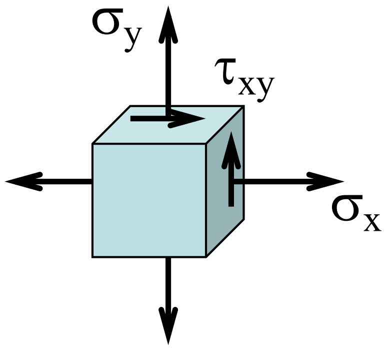
3D

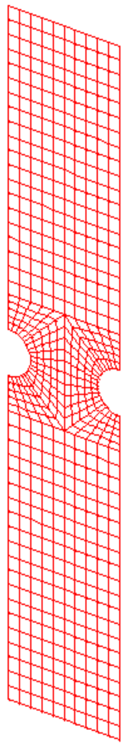


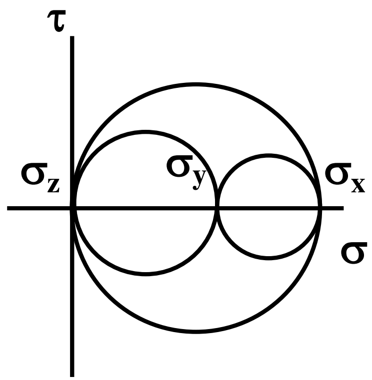
Plane Stress vs plane strain

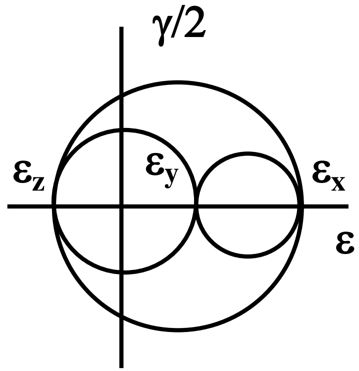
2D Plane Stress Elements







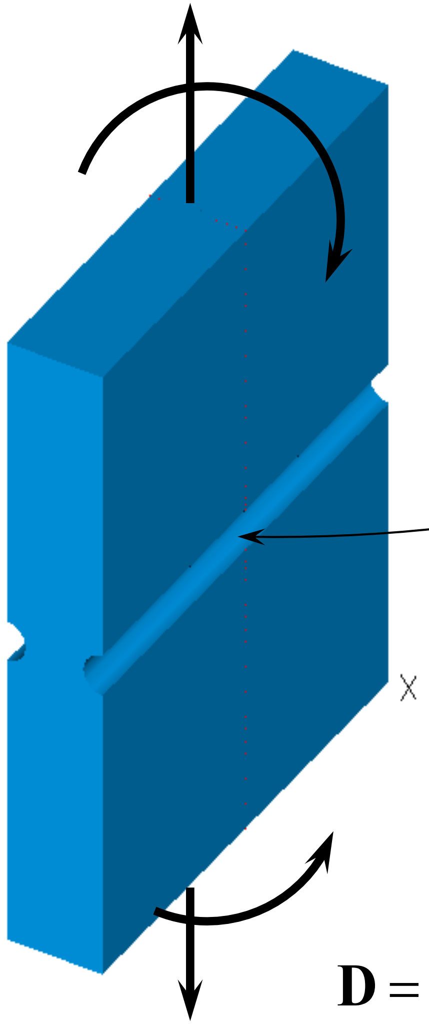


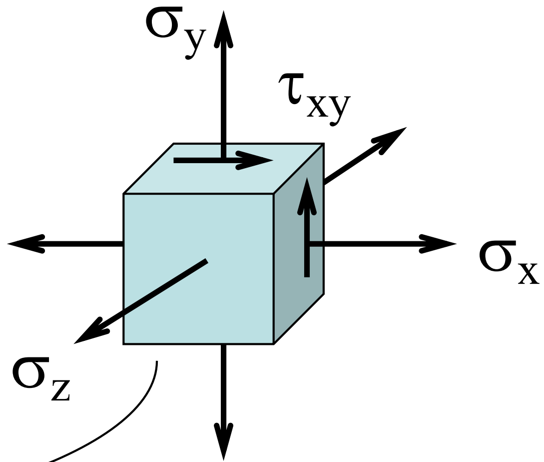


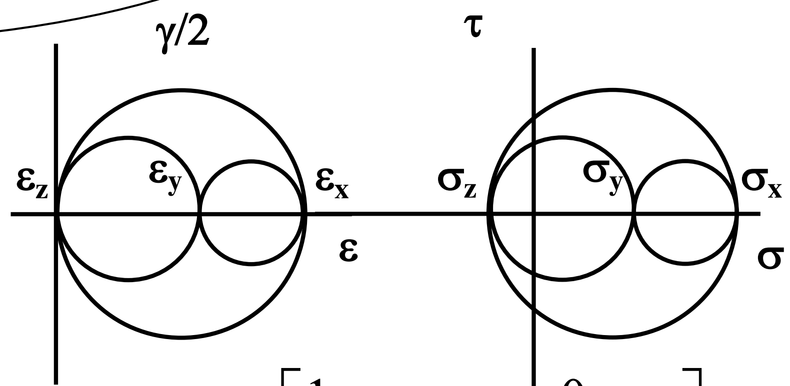
$$\{\sigma\} = [D]\{\varepsilon\}$$

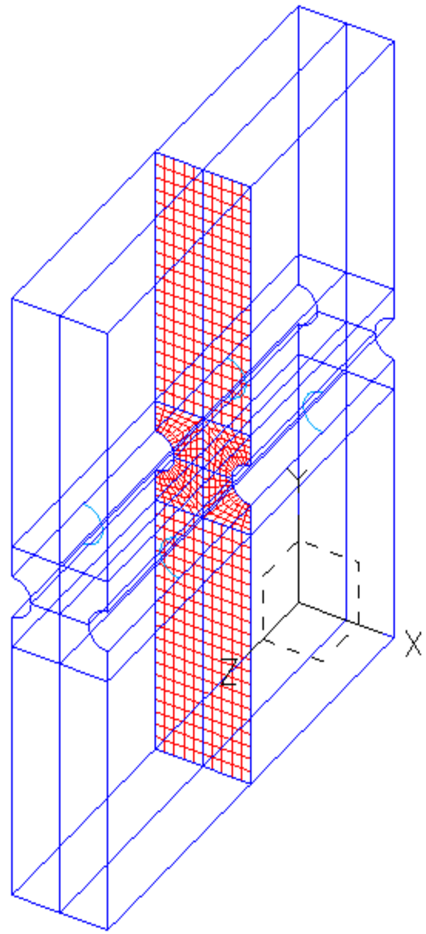
$$\mathbf{D} = \frac{E}{1-\nu^2}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-\nu)/2 \end{bmatrix}$$

2D Plane Strain Elements





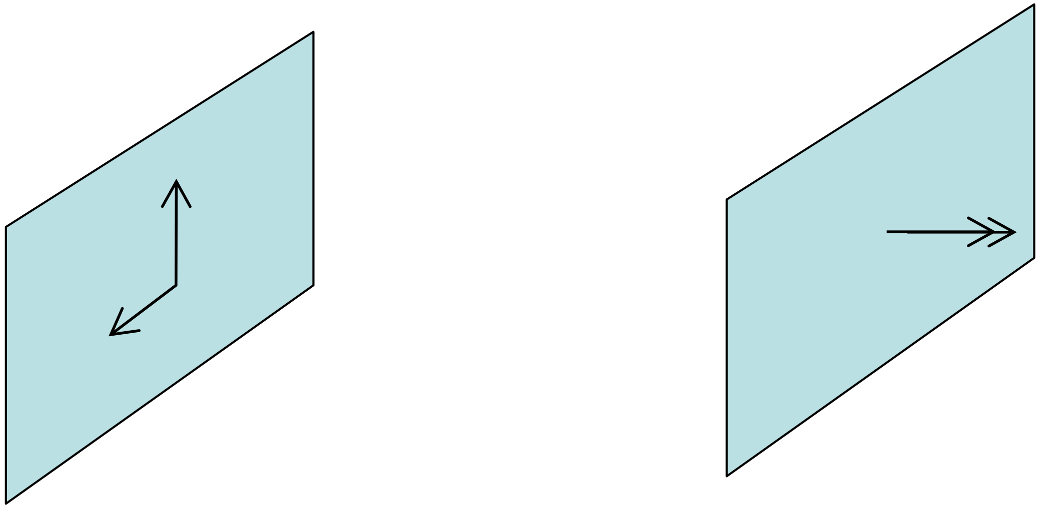




$$\mathbf{D} = \frac{E}{(1+\nu)(1-2\nu)}\begin{bmatrix} 1 & \nu & 0 \\ \nu & 1 & 0 \\ 0 & 0 & (1-2\nu)/2 \end{bmatrix}$$

Symmetry

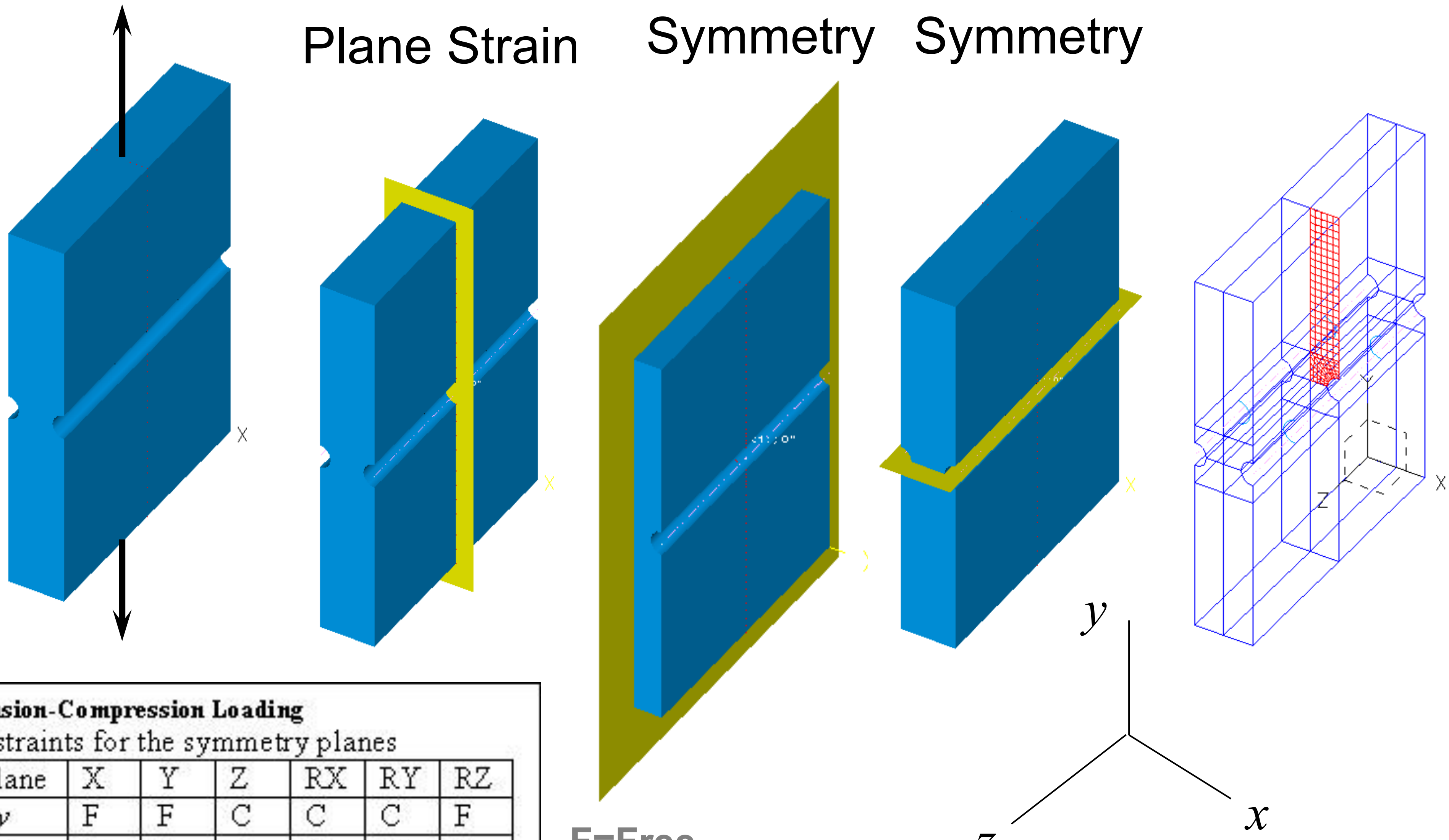
- For symmetry:-
  - Translations have no component normal to the plane of symmetry
  - Rotations have no component parallel to the plane of symmetry



Permitted translations

Permitted rotations

Taking Advantage of Symmetry



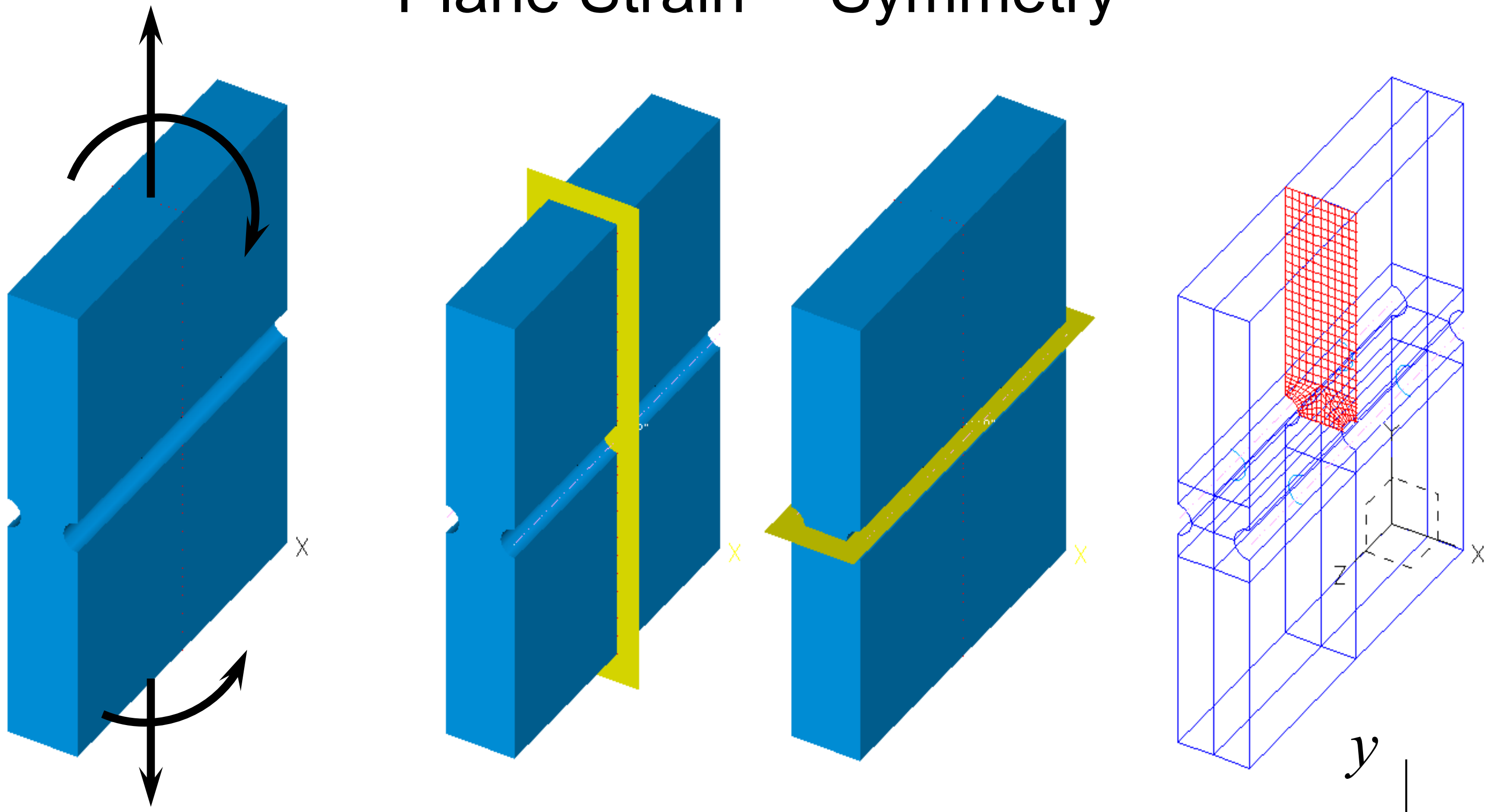
Tension-Compression Loading
Restraints for the symmetry planes

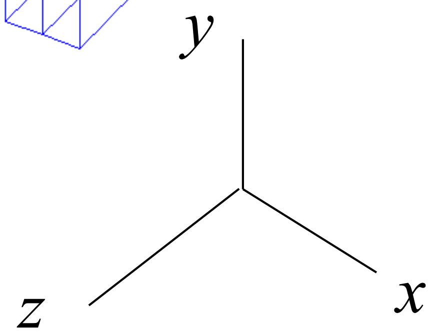
plane	X	Y	Z	RX	RY	RZ
xy	F	F	C	C	C	F
yz	C	F	F	F	C	C
xz	F	C	F	C	F	C

F=Free
C=Constrained

Taking Advantage of Symmetry

Plane Strain   Symmetry



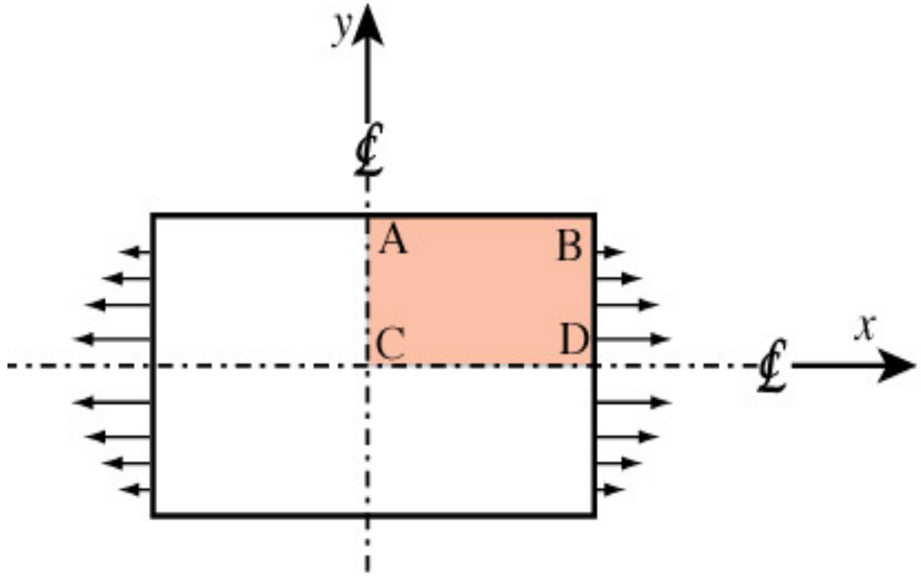


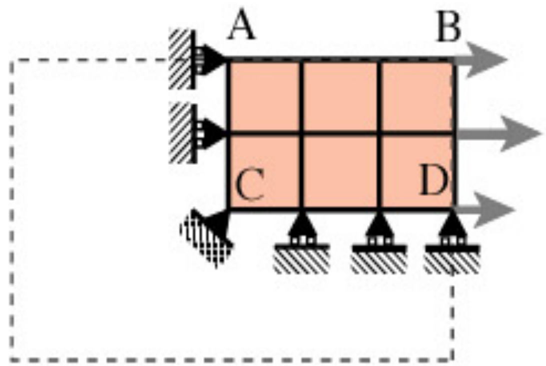
Four-Point Bending

Restraints for the symmetry planes

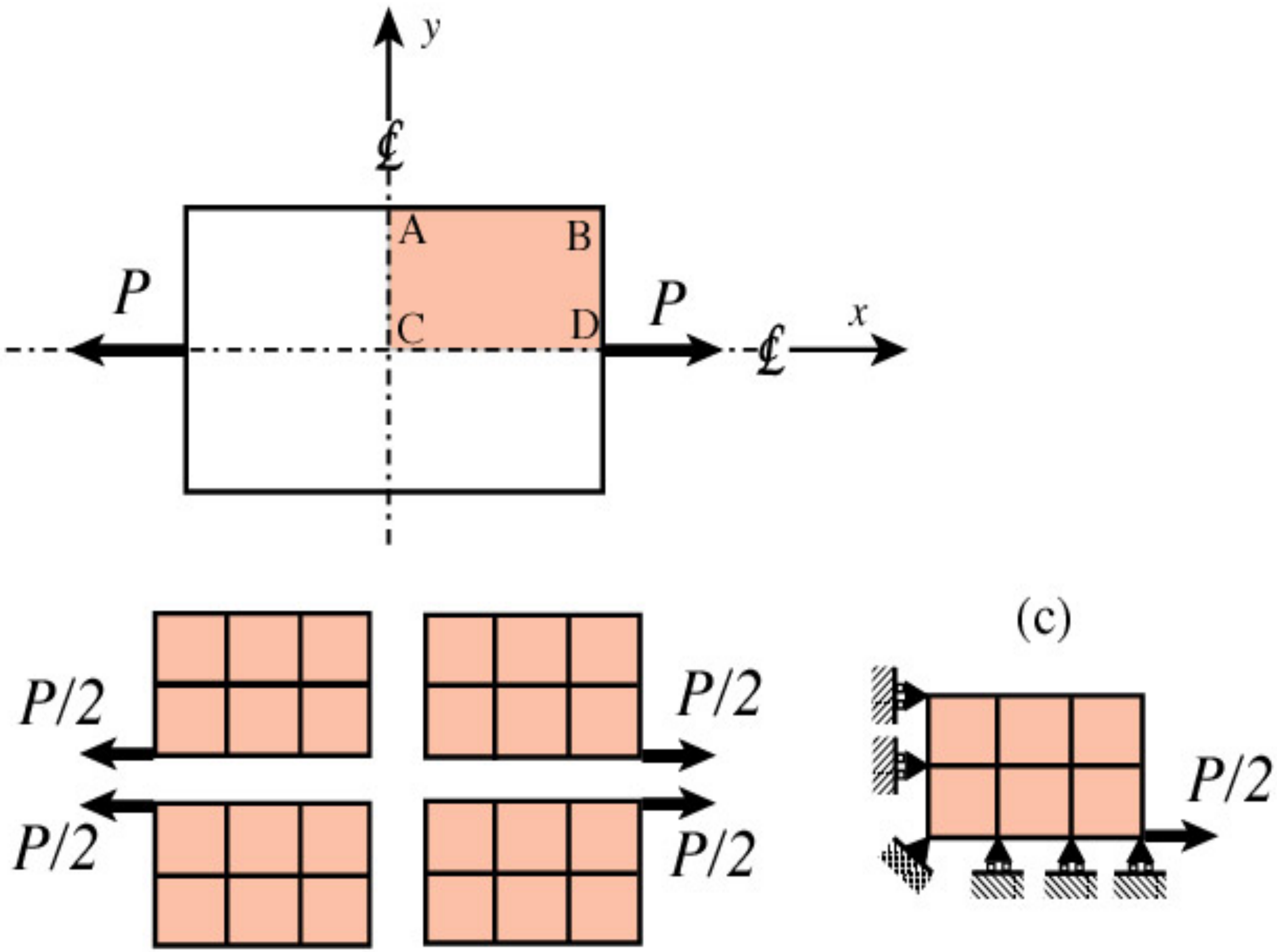
plane	X	Y	Z	RX	RY	RZ
xy	F	F	C	C	C	F
xz	F	C	F	C	F	C

Example of application of symmetry BC's



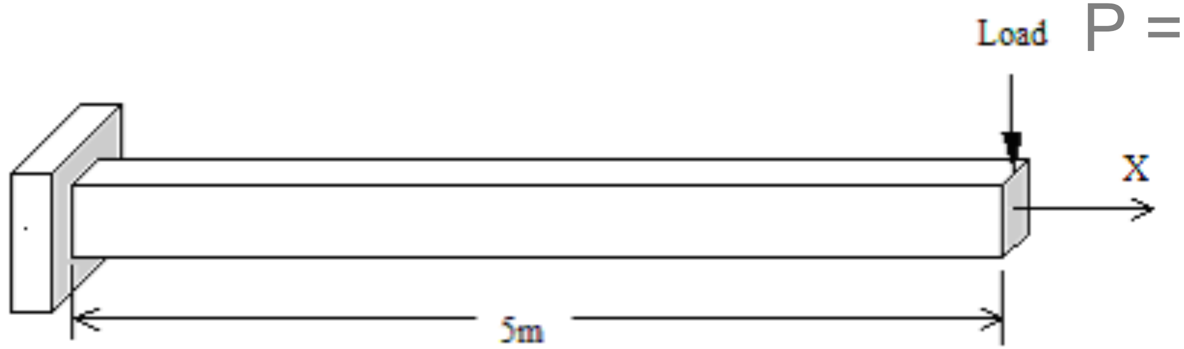


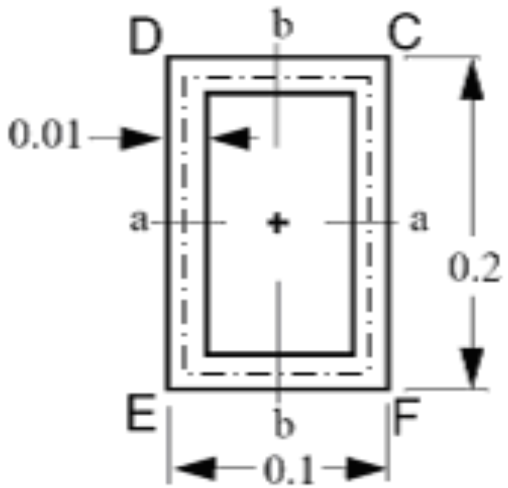
Breaking up point loads at symmetry BC's



Lab example

P = 100N





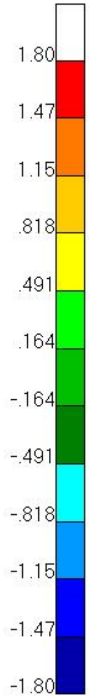
Beam theory gives:-

$$I = \frac{BD^3}{12} - \frac{bd^3}{12} = 2.78 \times 10^7 mm^4$$

$$\delta = \frac{PL^3}{3EI} = 2.11mm$$

$$\sigma_x = \frac{My}{I} = 1.8MPa$$

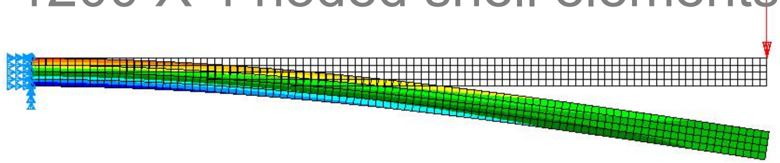
FE Results for different meshes



Single beam element

Max σ_x= 1.80MPa, δ = 2.12mm

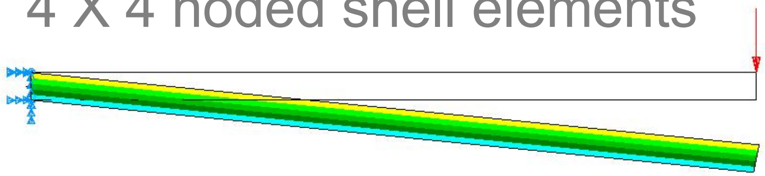
1200 X 4 noded shell elements



Max σ_x= 1.72MPa, δ = 2.12mm

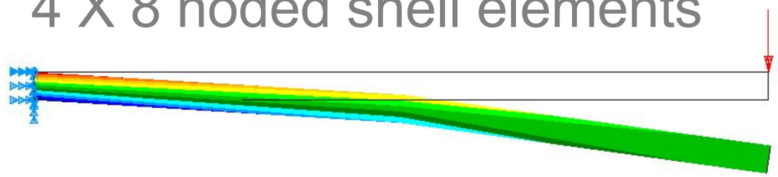
= 1.84MPa (shell top surface)

4 X 4 noded shell elements



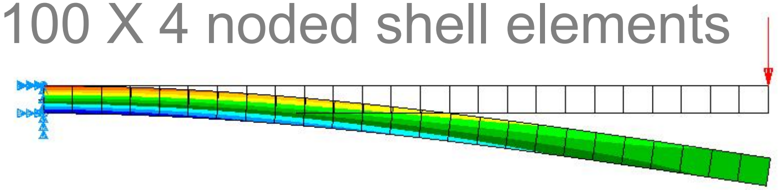
Max σ_x= 0.83MPa, δ = 1.55mm

4 X 8 noded shell elements



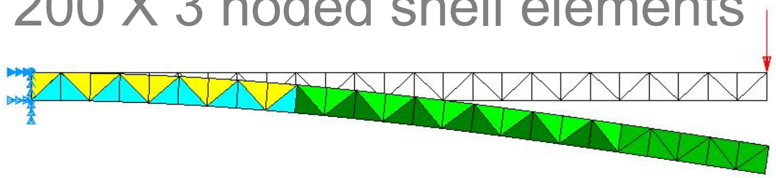
Max σ_x= 1.69MPa, δ = 2.05mm

100 X 4 noded shell elements



Max σ_x= 1.63MPa, δ = 2.12mm

200 X 3 noded shell elements



Max σ_x= 0.79MPa, δ = 1.03mm

Structures and Materials 3

Finite Element Method
Practical Application

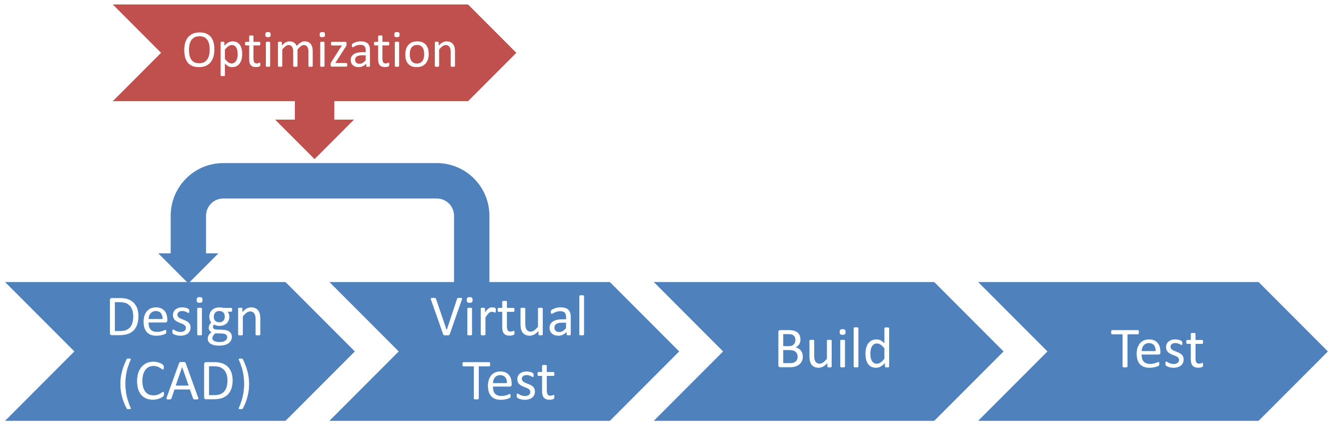
Dr Giuliano Allegri

Solve an Engineering Problem

Methods to Solve Any Engineering Problem

Analytical Method	Numerical Method	Experimental Method
• Classical approach • 100% accurate results • Closed form solutions • Applicable only for simple problems and boundary conditions (cantilever/simply supported beams)	• Mathematical representation of problem • Approximate, assumptions and simplifications • Applicable even in physical prototype not available (initial design phase) • Results should be verified against experimental data or hand calculations.	• Measurements obtained with a reasonable degree of accuracy and precision. • Time consuming and expensive • Physical porotype required for testing. • Many prototypes should be tested to obtain a statistical measure of results.
• Analytical Methods	• **Finite Element Method** – Structural Analysis • **Computational Fluid Dynamics** (CFD) etc.	• Strain gauges • Photo-elasticity • Digital Image Correlation • Vibration measurements etc.

CAE Design Driven Process

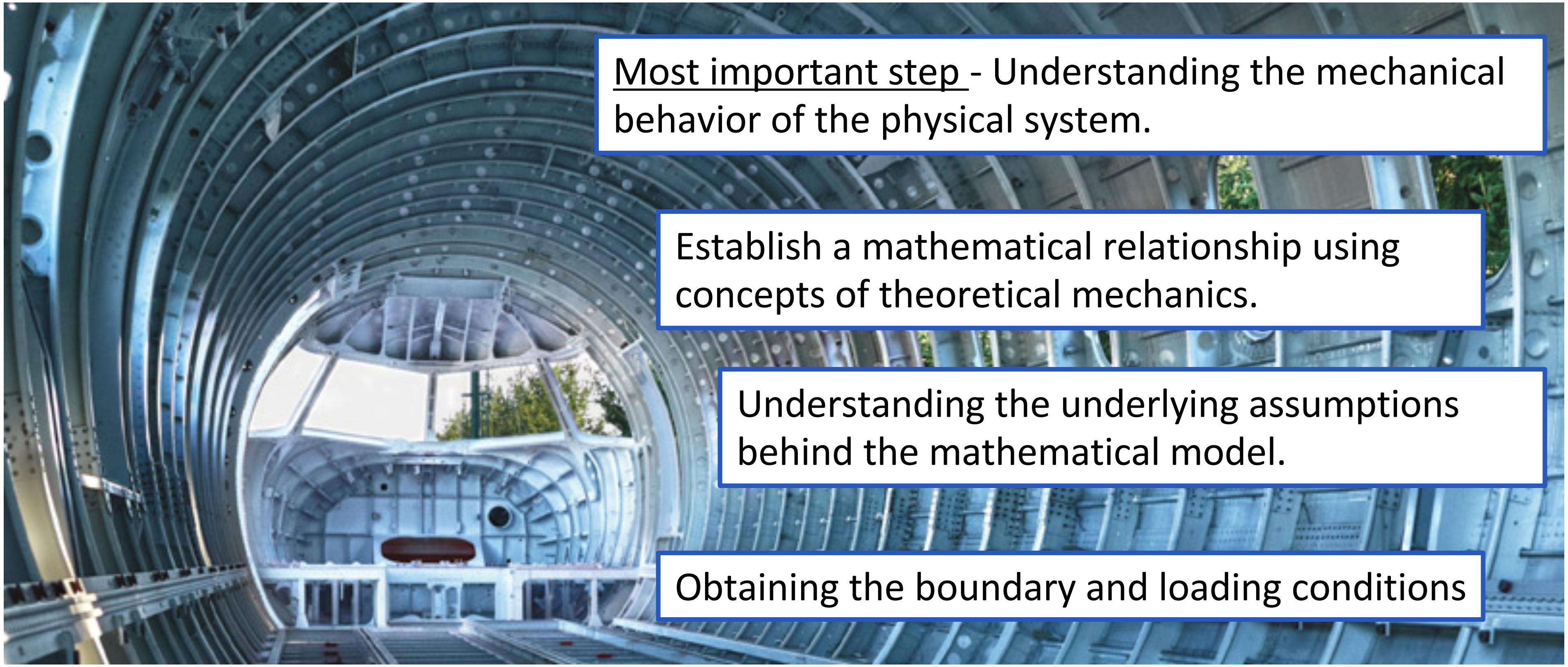


- Visualisation ↑
- Design cycle time ↓
- No. of prototypes ↓
- Testing ↓
- Optimum design

- Novel and innovative designs
- Components with superior material efficiency = lighter designs.
- Shorter development cycles and a reduction in the number of prototypes by minimizing “Trial and Error” attempts.

Simulation can save time, reduces costs, and essentially strengthens the competitiveness of companies, thus strengthening their market position.

Understanding of the Physical System



Most important step - Understanding the mechanical behavior of the physical system.

Establish a mathematical relationship using concepts of theoretical mechanics.

Understanding the underlying assumptions behind the mathematical model.

Obtaining the boundary and loading conditions

First Step : Think about the problem before you start…

Solutions:

- Analytical ("closed form solutions") – simple BC
- Finite Element Method - Approximation.

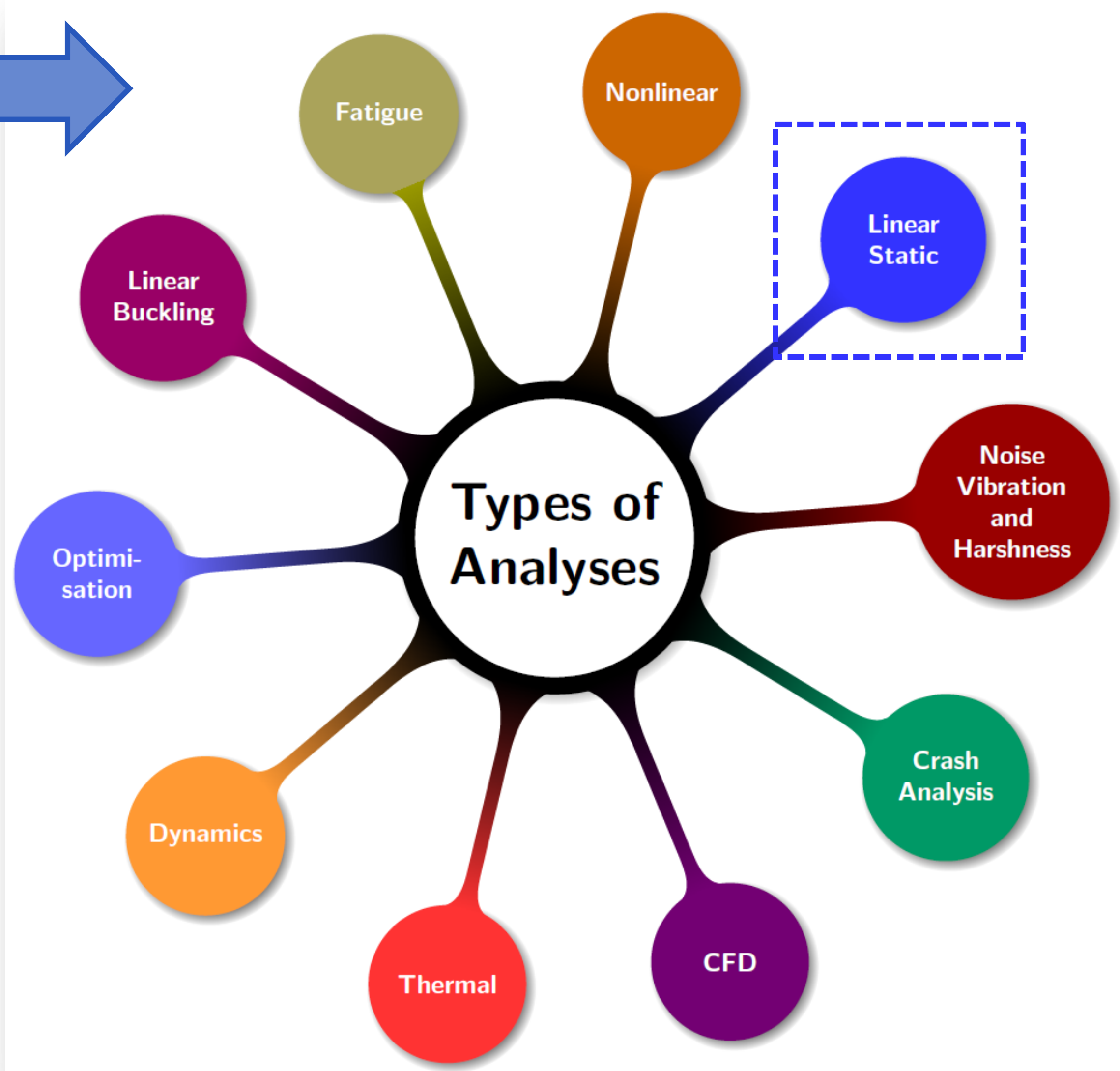
The Finite Element Method

Recall Lecture 1

Physical System	1. Complex physical entity
Idealisation	2. Physical entity is reduced to a simplified representation that captures the structural entities.
Discretisation	3. A body is modelled by dividing it in to an equivalent system of smaller bodies or units (finite elements) interconnected at points common to adjacent elements (nodes)
Discrete Solution	4a. Obtain a set of algebraic equations and solve for unknown nodal values (displacement) 4b. Stresses and strains are back calculated from displacements using constitutive relations
Interpretation of Solution	5. Results are examined for critical stresses etc.

Analysis Types

Most CAE (Computer Aided Engineering) includes the following types of analyses:

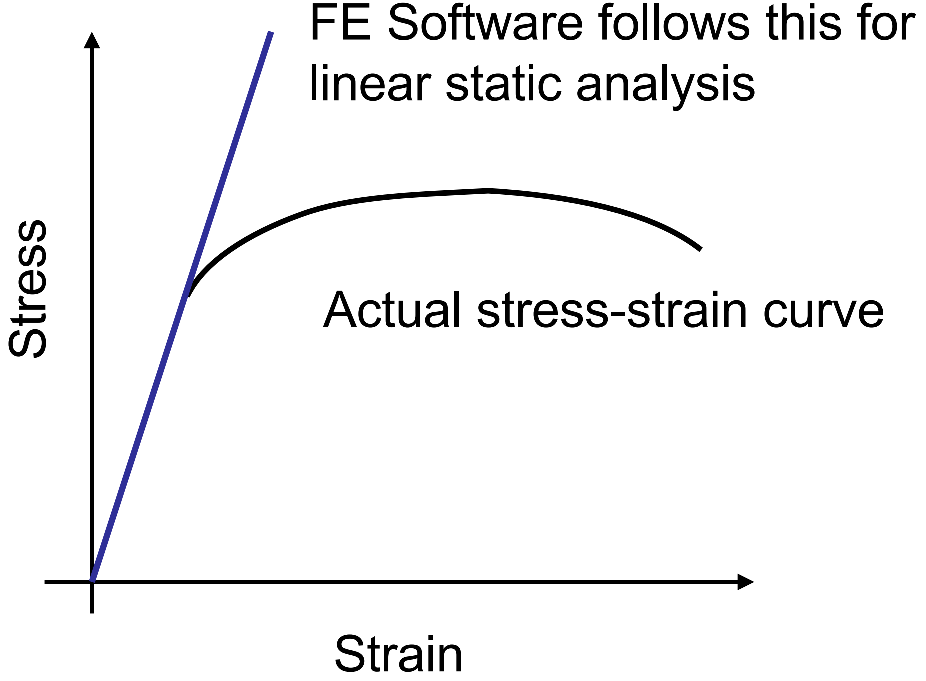


<u>Practical Applications:</u>

- A linear static analysis is the most commonly used analysis.
- Most aerospace, automobile, offshore and civil engineering industries perform linear static analyses.

Analysis Types – Linear Static

Linear

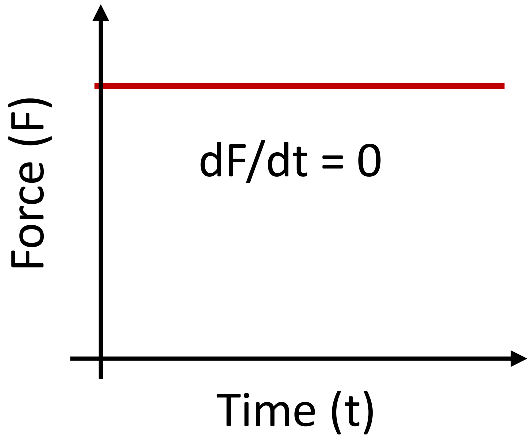


The complete model summation of the external forces and moments is equal to the reaction forces and moments.

Static

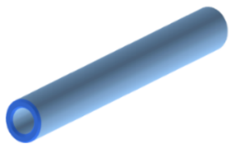
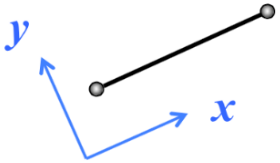
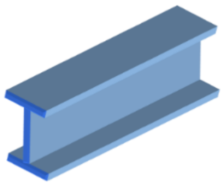
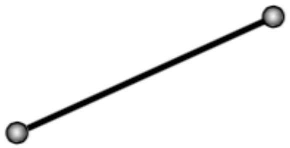
Two conditions must be met for static analysis:

1. The force is static i.e. there is no variation with respect to time



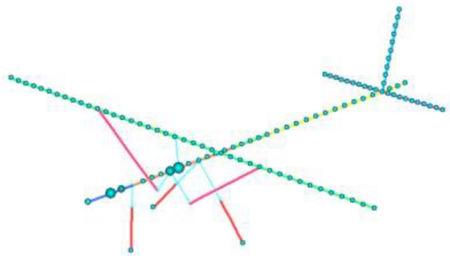
2. Equilibrium condition:
   - Σforces (F_x, F_y, F_z) = 0.
   - ΣMoments (M_x, M_y, M_z) = 0.

1D Elements

Physical Structural Component	Finite Element Discretisation	Active degrees of freedom
 Bar/Truss		Axial u_x, u_y, u_z
 Beam		Axial, bending, (torsion) u_x, u_y, u_z $\theta_x, \theta_y, \theta_z$

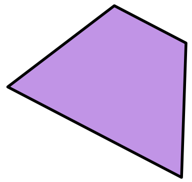
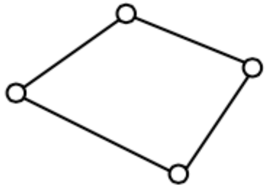
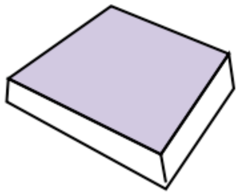
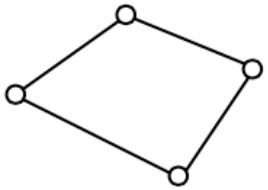
Truss-like structures - Harbourside cranes, Bristol, UK

Simple Loads model



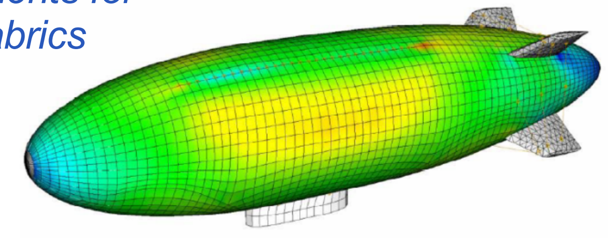
- Often referred to as line elements.
- Used to represent members, which are long compared to the measurement of the cross-section **(L/r > 20)**
- Useful when bending is the root cause of failure.
- Fundamental assumption: Changes in material properties along the cross-section are negligible.

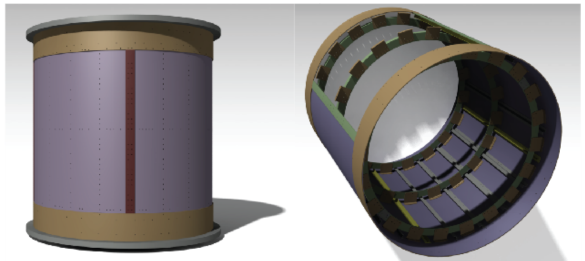
2D Elements

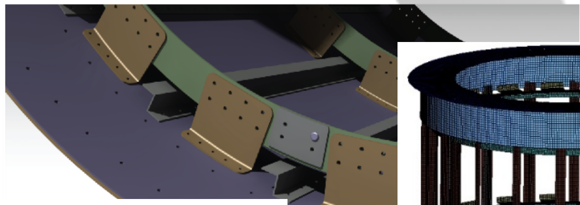
Physical Structural Component	Finite Element Discretisation	Active degrees of freedom
 Membrane		u_x, u_y , u_z Offer in-plane stiffness e.g. fabric
 Shell		u_x , u_y , u_z , θ_x ,θ_y ,θ_z In-plane + bending e.g. metal plate

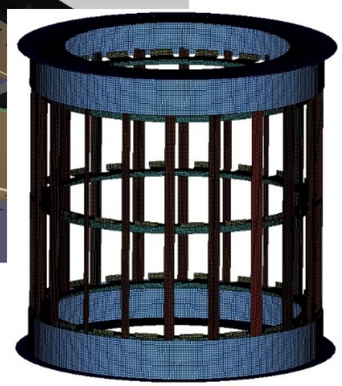
- Used when thin, sheet structures are under bending deformation.
- Can consider 2D stress conditions and bending and shear deformations.
- Fundamental assumption: Changes in material properties along the thickness of the structure are negligible.

Membrane elements for fabrics





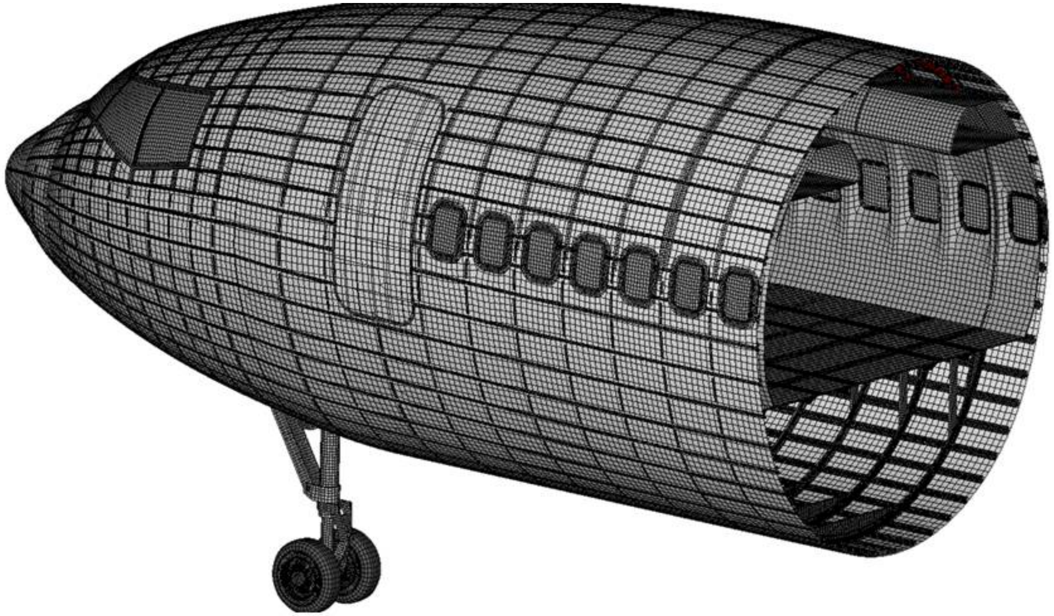


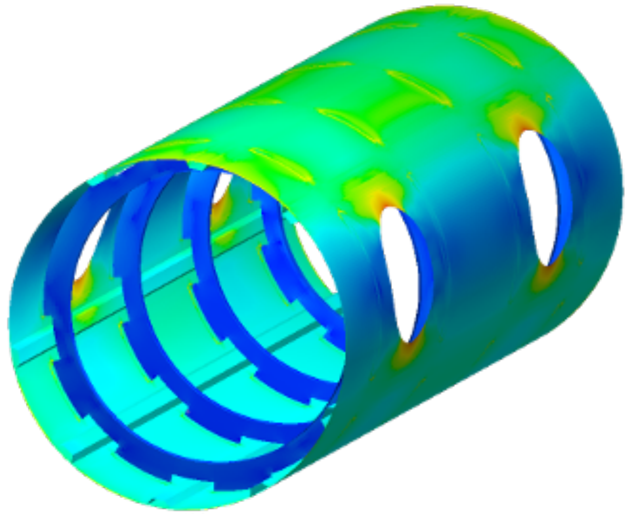


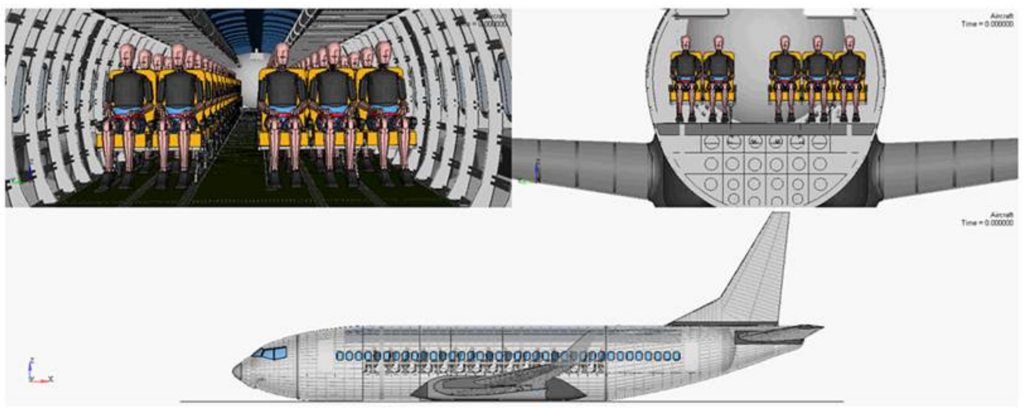
Barrel skin meshed with 2D shell elements

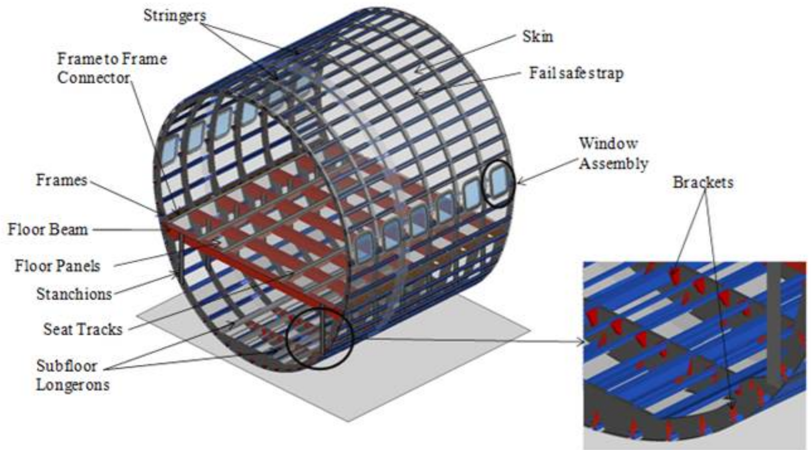
Aerospace Applications using 2D elements

Metallic Narrow Body Transport Aircraft FE Model









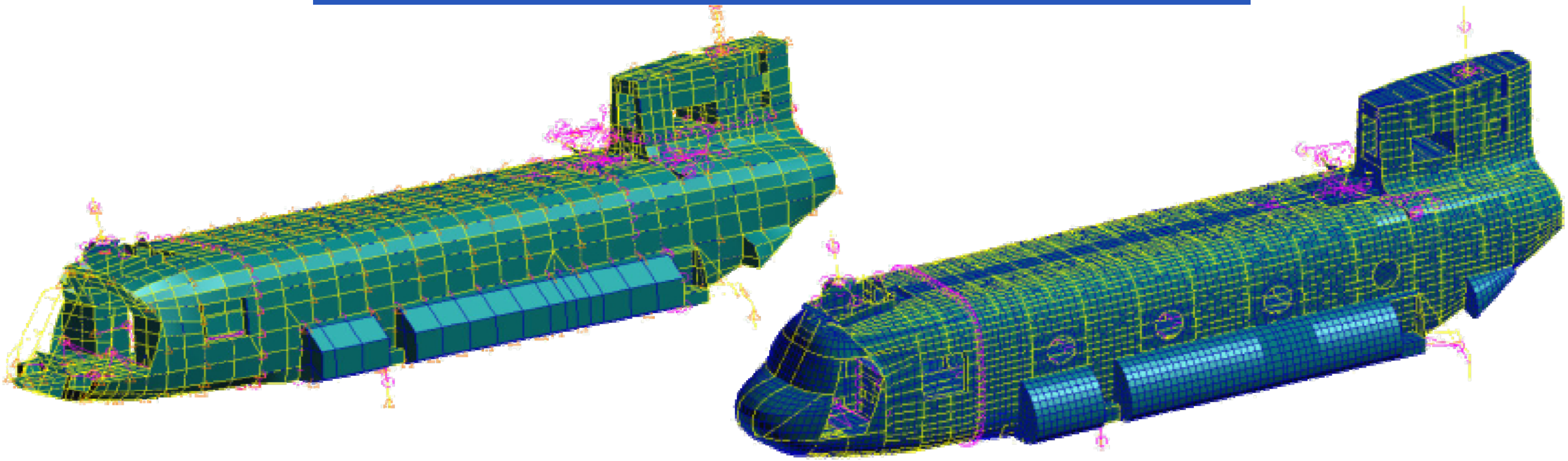
http://asidiconference.org/wp-content/uploads/2012/09/airframe.jpg

Aerospace Applications using 2D elements

Low fidelity – course mesh (early design stages)

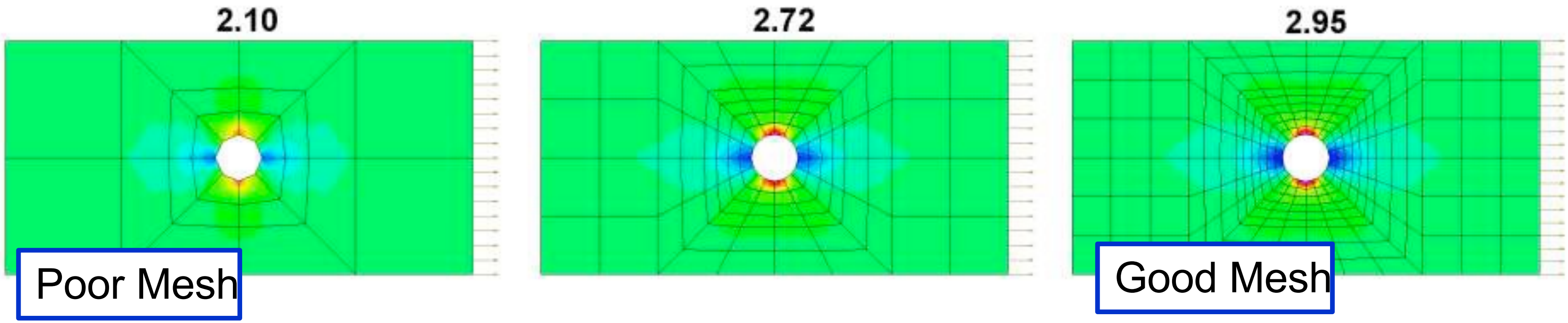
High fidelity – refined mesh (final design and detailed stress check)

What is the appropriate level of discretisation?



Why is mesh quality (discretization) important?

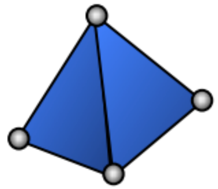
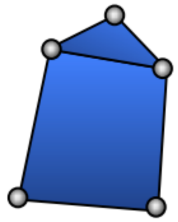
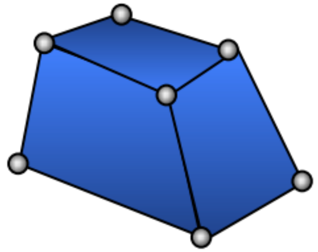
- To obtain an accurate representation of geometry.
- To avoid convergence issues due to poor mesh design and shape.
- To ensure uniform transfer of load among adjacent nodes.



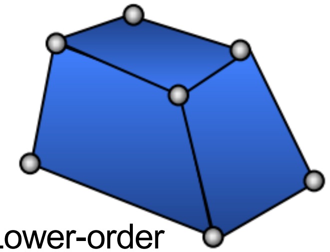
Stress concentration in a thin sheet of aluminium with a circular hole and uniform far-field stress:

- Three levels of mesh refinement produce three different results

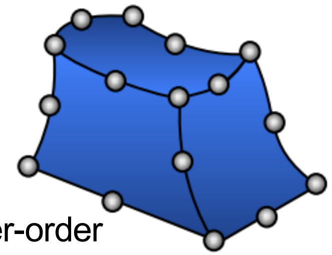
3D Elements

Type	Tetrahedron	Pentahedron	Hexahedron
Shape			
No. of nodes	**4** (lower-order elements) **10** (higher-order elements)	**6** (lower-order elements) **15** (higher-order elements)	**8** (lower-order elements) **20** (higher-order elements)
DOF per node	**3** Translational DOF (Tx, Ty, Tz) **No rotational DOF**		

Higher order elements have mid-side nodes and can represent more complex shapes/stresses.



Lower-order elements

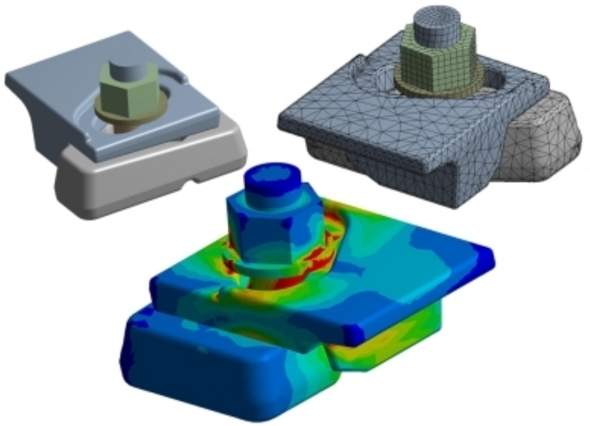


Higher-order elements

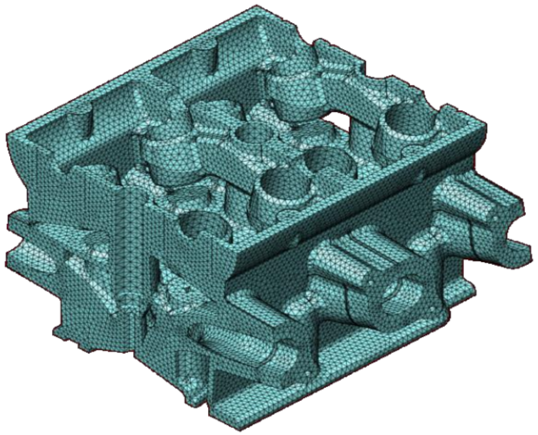
- Also referred to as **solid or brick elements**.
- Used in analyses generated from 3D CAD models
- **Hexahedral elements produce good results -** Primarily required where accurate displacements/stresses are required.
- **Tetrahedral elements** are easily generated by auto-meshers for interiors, where stiffness and mass calculations are more meaningful.

3D Elements - Applications

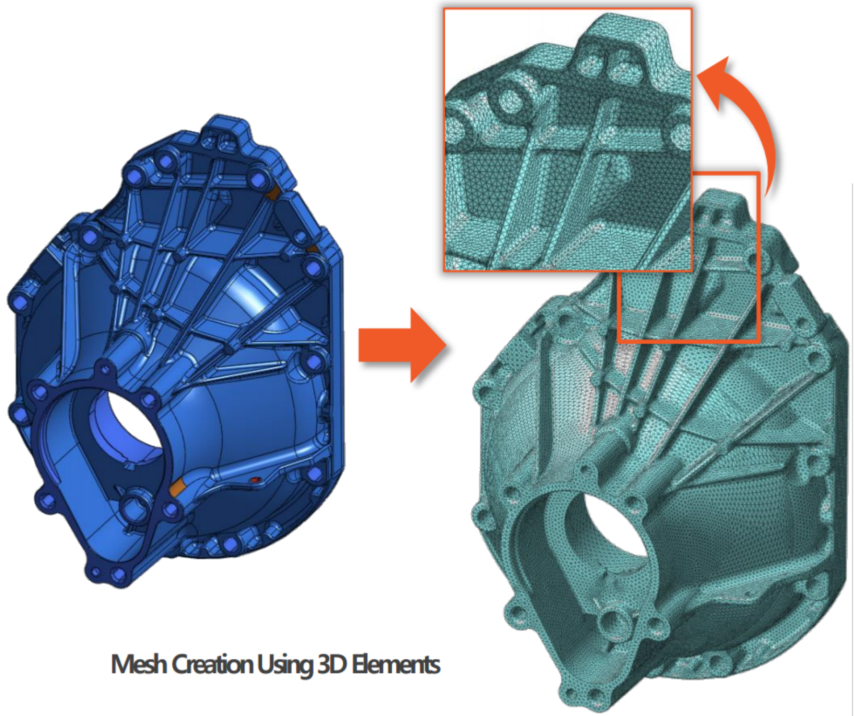
Easier to mesh a complex CAD models using 3D elements.



Bolted pre-tension analysis

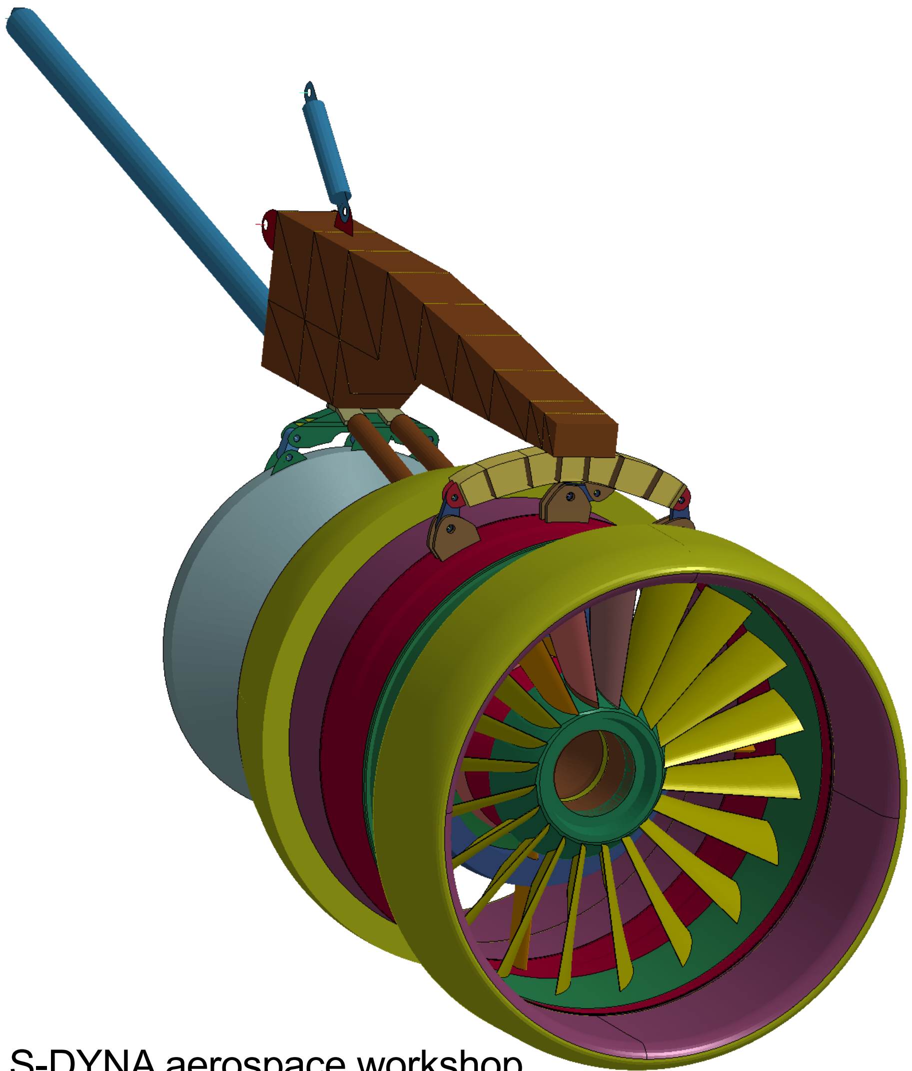


Engine Block meshed using tetrahedral elements

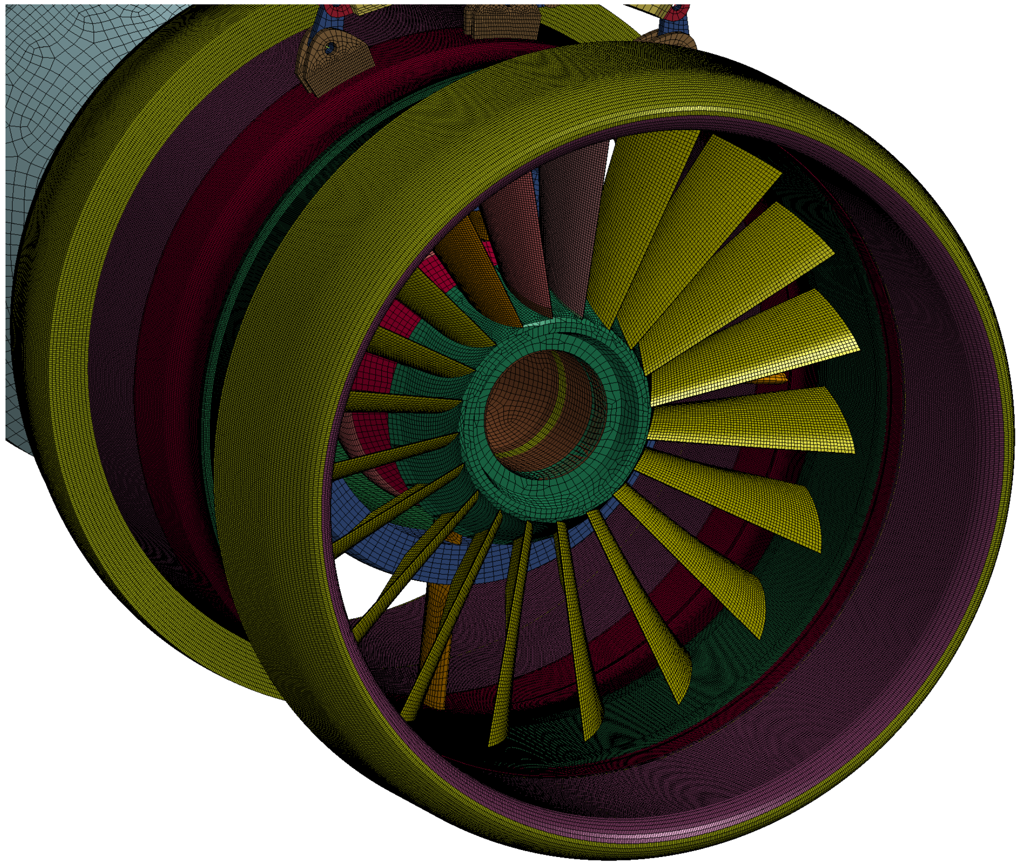


Mesh Creation Using 3D Elements

Combination of Analysis and Element Types



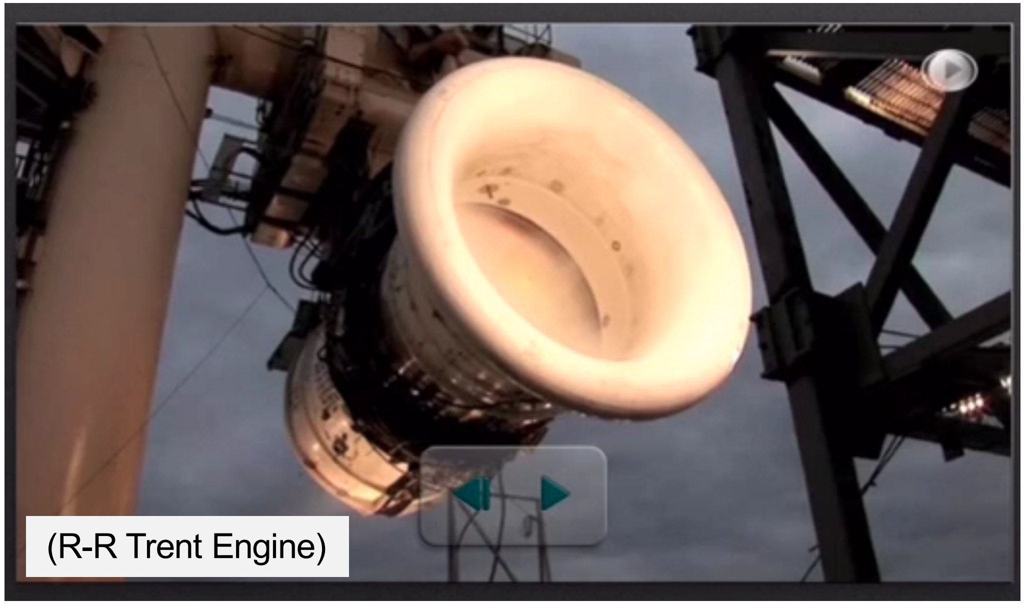
LS-DYNA aerospace workshop



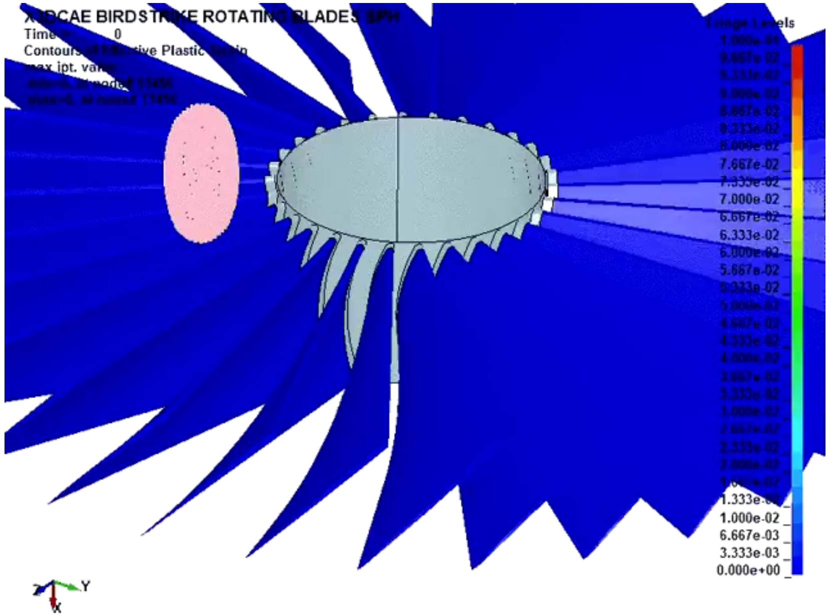
Structures can be modelled with combination of element types e.g. shell elements for fan blade, 3D elements for dovetail disc, 1D beam elements for mounting fixture.

Bird Strike Simulation

Crash Analysis



(R-R Trent Engine)



<u>Nonlinearity</u>

- Geometric – Large Deformation
- Material – Damage, beyond elastic limit
- Contact – Interaction

<u>Dynamic</u>

- Force is no longer static
- Force varies with respect to time
  - F = ma
- Output results are a function of time/frequency
- Must define inertia of system

MSC Nastran Explicit,
https://www.youtube.com/watch?v=0zmBb3JER5g

FINITE ELEMENT ANALYSIS

COMMON MISTAKES AND ERRORS

TAKE NOTE!!!

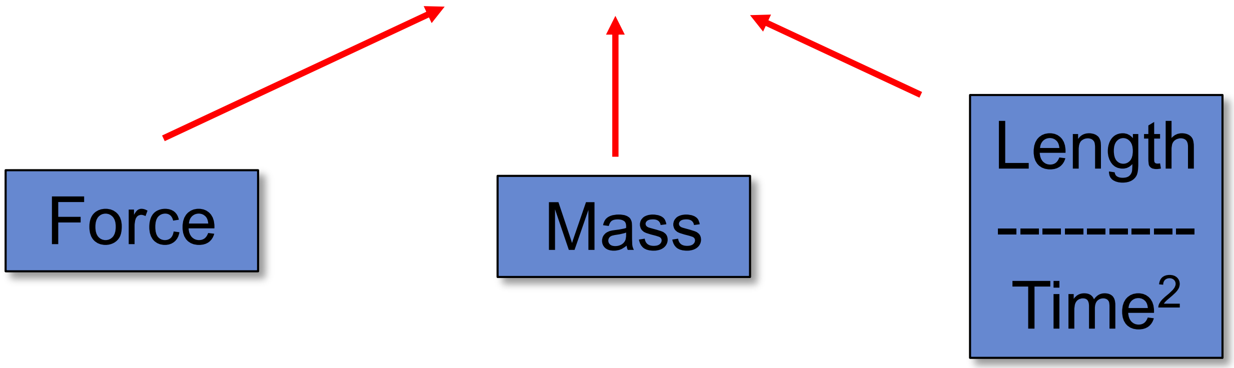
Common Mistakes and Errors

- Geometry Simplification
- Poorly defined boundary and loading conditions
- Mesh quality and design
- Visualisation – Check the magnitude and direction of contour plots
- Inconsistencies in your unit system

Consistent Units

- Newton's Second Law of Motion contains the units of force, mass, length, and time

$$F = M \cdot a$$



- We can choose any three of the four units as our base units. The fourth unit is then derived from these three base units

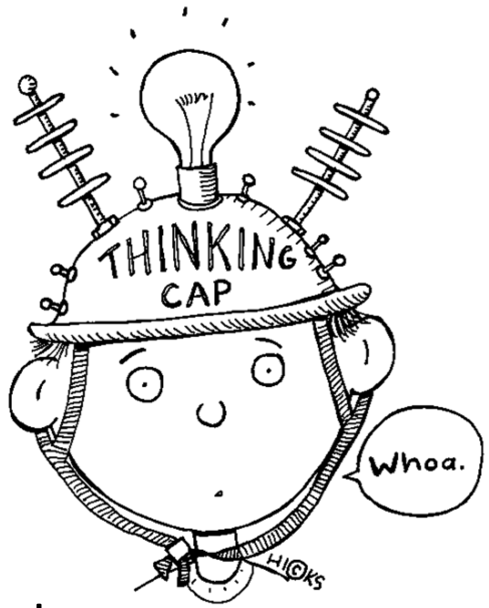
Examples of consistent systems of units

Following table contains some of the most commonly used consistent system of units

System of Units	Input							Output		
	Length	Force	Elastic Modulus	Mass	Mass Density	WTMASS Parameter	1 G	Disp	Force	Stress
1	m	N	Pa	kg	kg/m^3	1.0	9.807 m/sec^2	m	N	Pa
2	mm	N	MPa	t or Mg	t/mm^3 or Mg/mm^3	1.0	9807 mm/sec^2	mm	N	MPa
3	ft	lb_f	psf	slug	slug/ft^3	1.0	32.17 ft/sec^2	ft	lb_f	psf
4	in	lb_f	psi	$lb_f \cdot sec^2/in$	$lb_f \cdot sec^2/in^4$	1.0	386.1 in/sec^2	in	lb_f	psi
5	in	lb_f	psi	lb_f	lb_f/in^3	2.59×10^{-3}	386.1 in/sec^2	in	lb_f	psi

Importance of **Engineering Judgment**

- What kind of behavior is essential to analyze to investigate this problem (linear, nonlinear, static, dynamic, steady, transient etc.)?
- What type of elements should be used?
- Does the finite element model effectively represent the physics of the system?
- Does it comply with the theoretical assumptions?



- Is this the best available method in terms of time & cost-effectiveness?
- If not, are there any practical, economical and effective alternatives?
- How much percentage error can we account for, by adopting the more practical alternative?

Final Comments

- Use the **simplest** finite element model that will do the job.
- ***Don't*** use complicated or special purpose elements, unless you are absolutely sure of what you are doing.
- Use the **coarsest** mesh you think will capture the dominant behavior of the physical system, particularly in design applications.

Keep it simple!!!