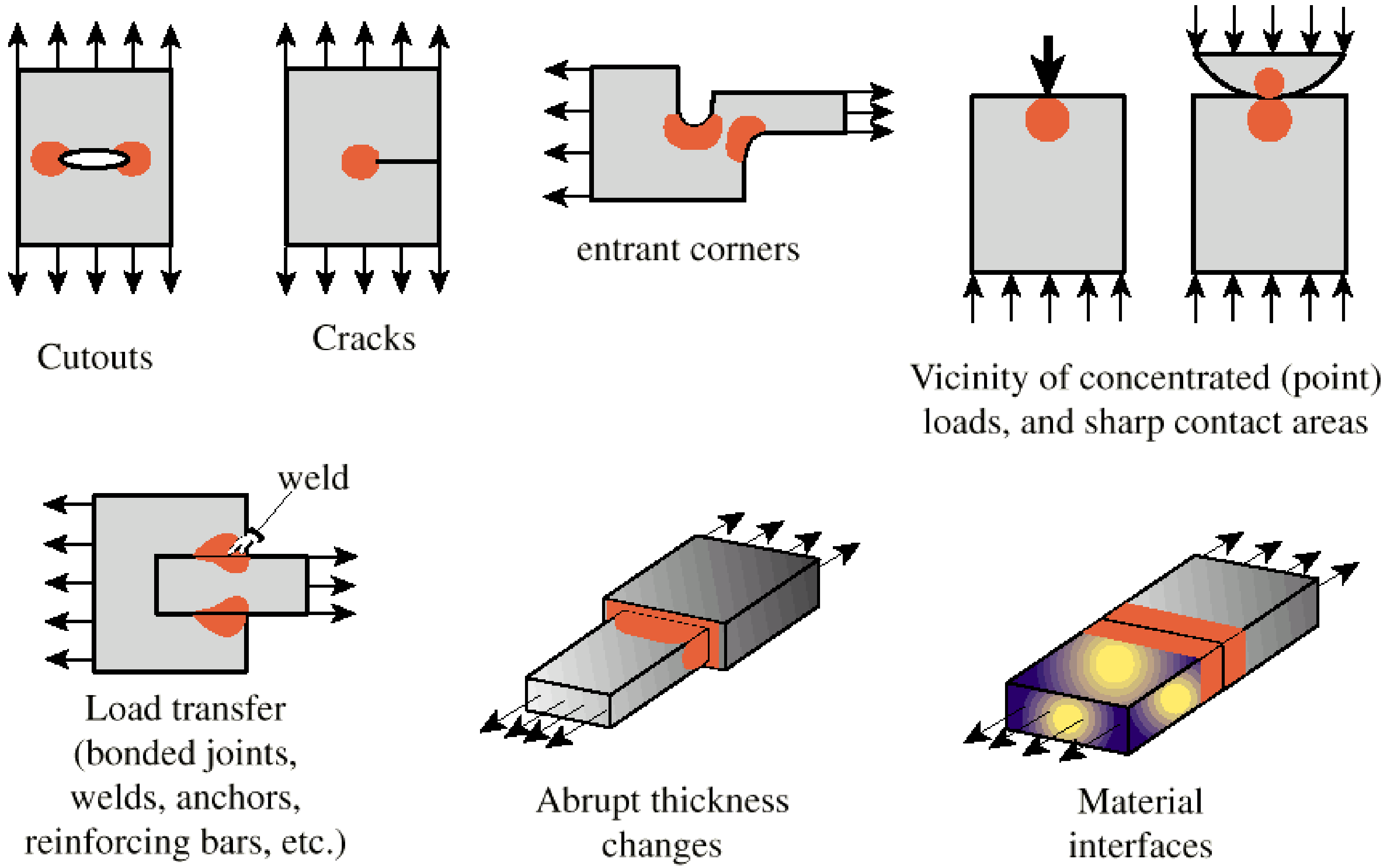
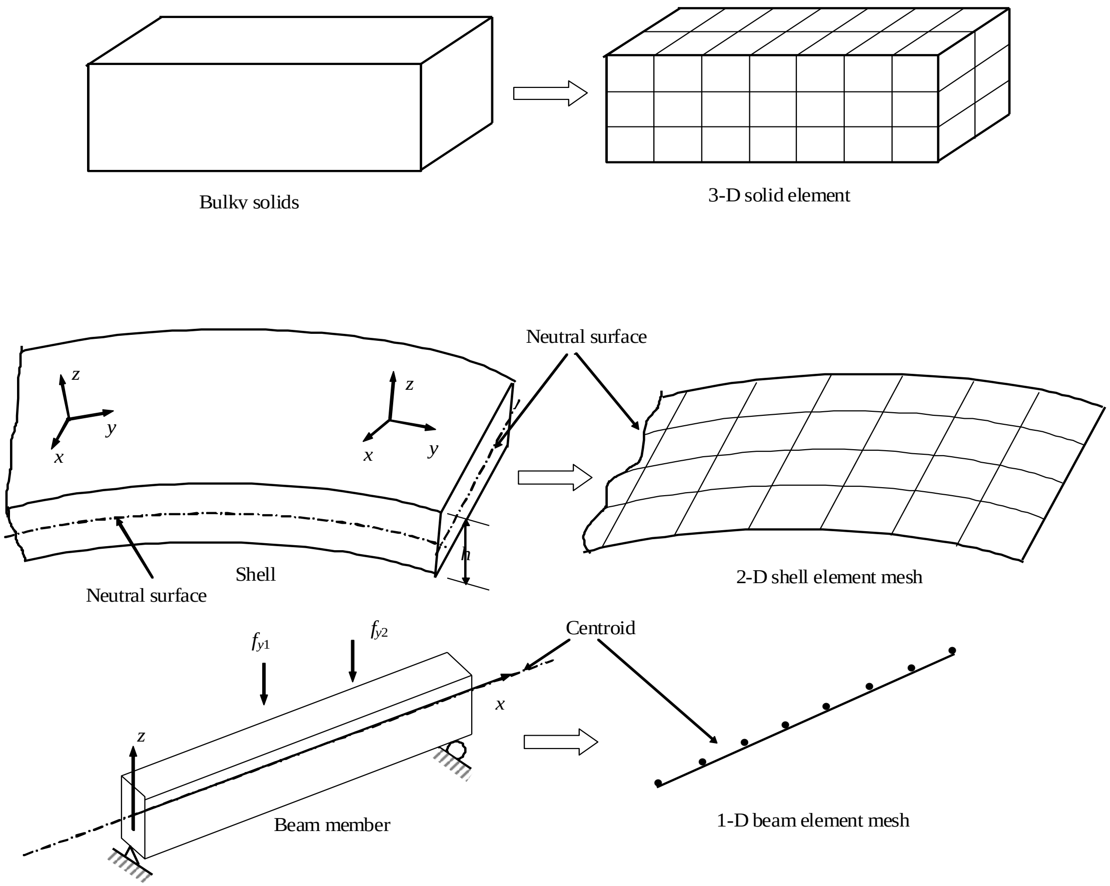
Finite Element Modeling Techniques (2)

Where Finer Meshes Should be Used



GEOMETRY MODELLING

- Reduction of a complex geometry to a manageable one.
- 3D? 2D? 1D? Combination?

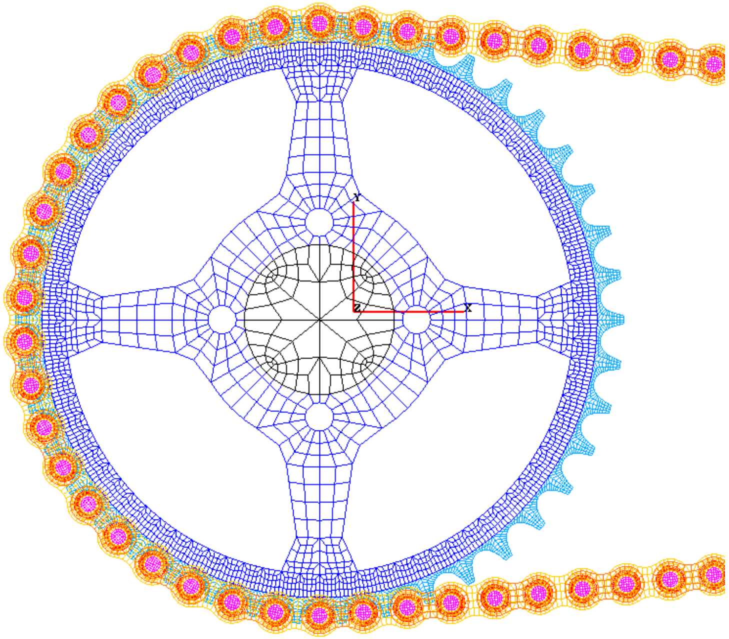


(Using 2D or 1D makes meshing much easier)

- Detailed modelling of areas where critical results are expected.
- Use of CAD software to aid modelling.
- Can be imported into FE software for meshing.

Mesh density

- To minimize the number of DOFs, have fine mesh at important areas.
- In FE packages, mesh density can be controlled by mesh seeds.



(Image courtesy of Institute of High Performance Computing and Sunstar Logistics(s) Pte Ltd (s))

Element distortion

- Use of distorted elements in irregular and complex geometry is common but there are some limits to the distortion.
- The distortions are measured against the basic shape of the element

  Square $\Rightarrow$ Quadrilateral elements

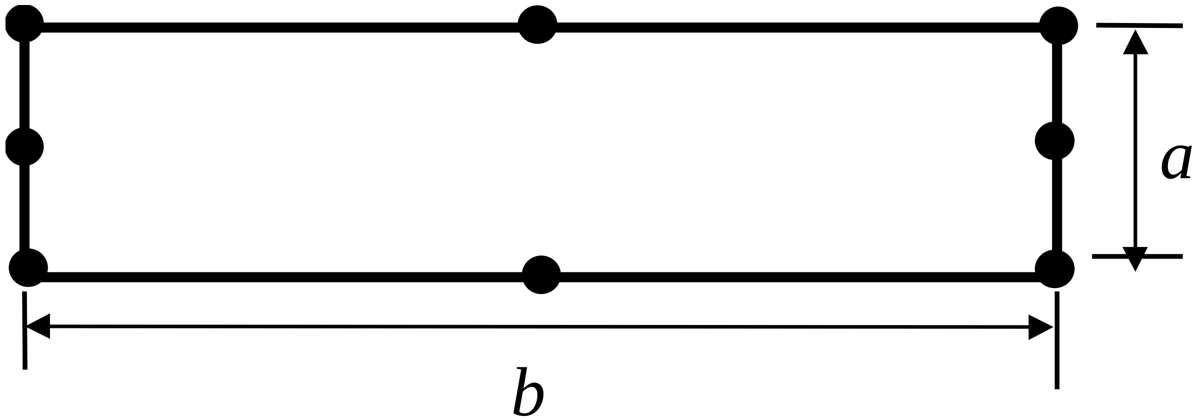
  Isosceles triangle $\Rightarrow$ Triangle elements

  Cube $\Rightarrow$ Hexahedron elements

  Isosceles tetrahedron $\Rightarrow$ Tetrahedron elements

Element distortion

- *Aspect ratio distortion*

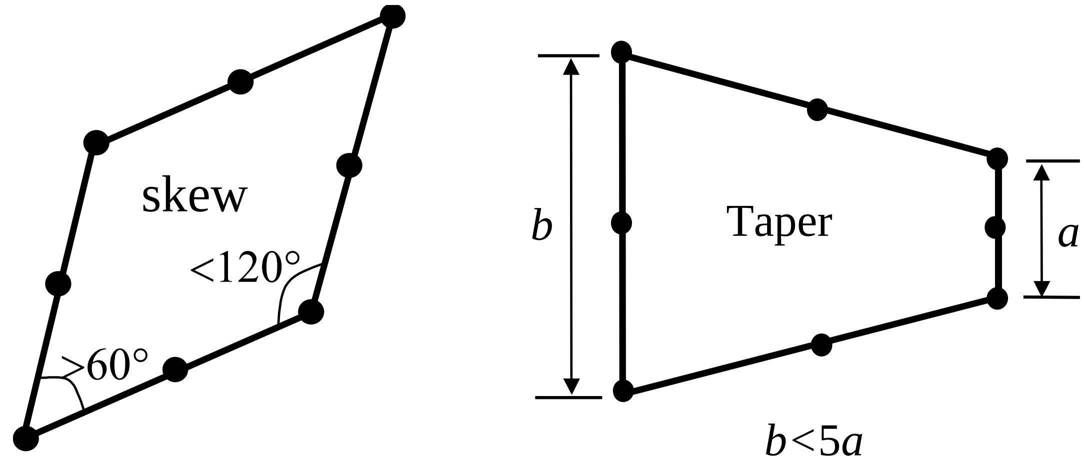


Rule of thumb:

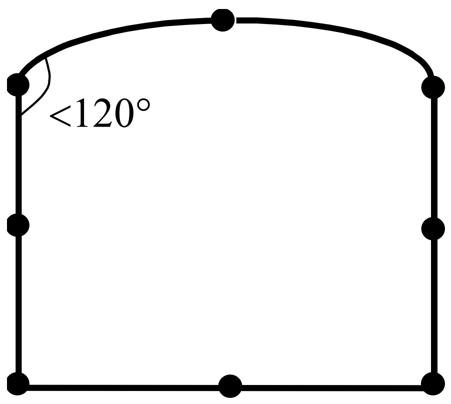
$$\frac{b}{a} \leq \begin{cases} 3 & \text{Stress analysis} \\ 10 & \text{Displacement analysis} \end{cases}$$

Element distortion

- *Angular distortion*

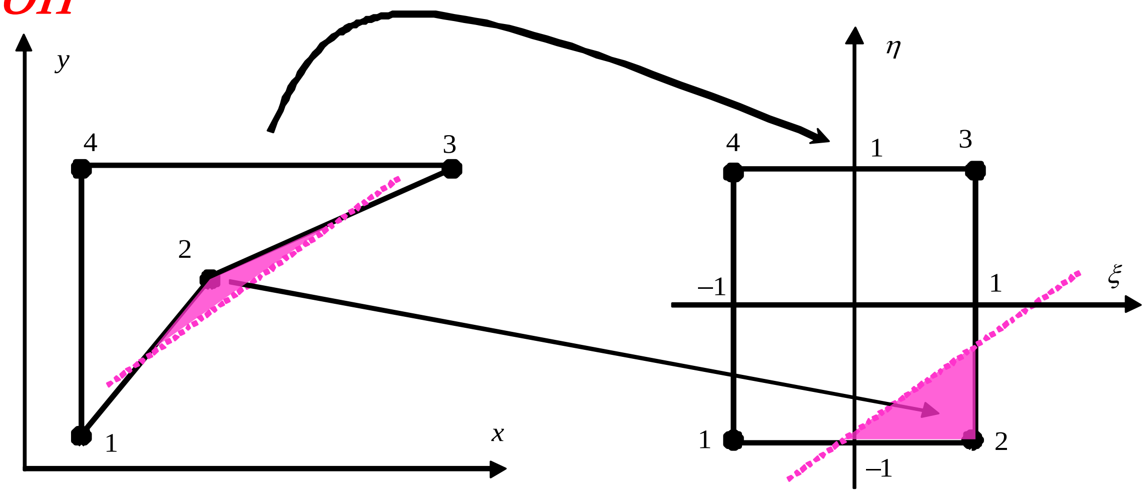


- *Curvature distortion*

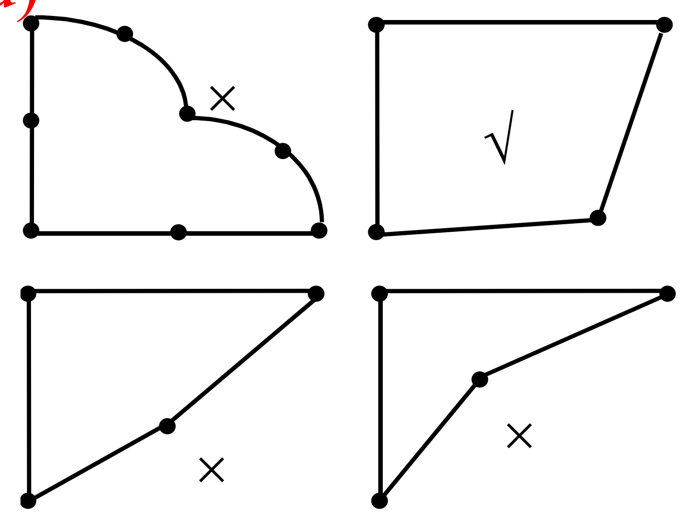


- *Volumetric distortion*

Area outside distorted element maps into an internal area – negative volume integration

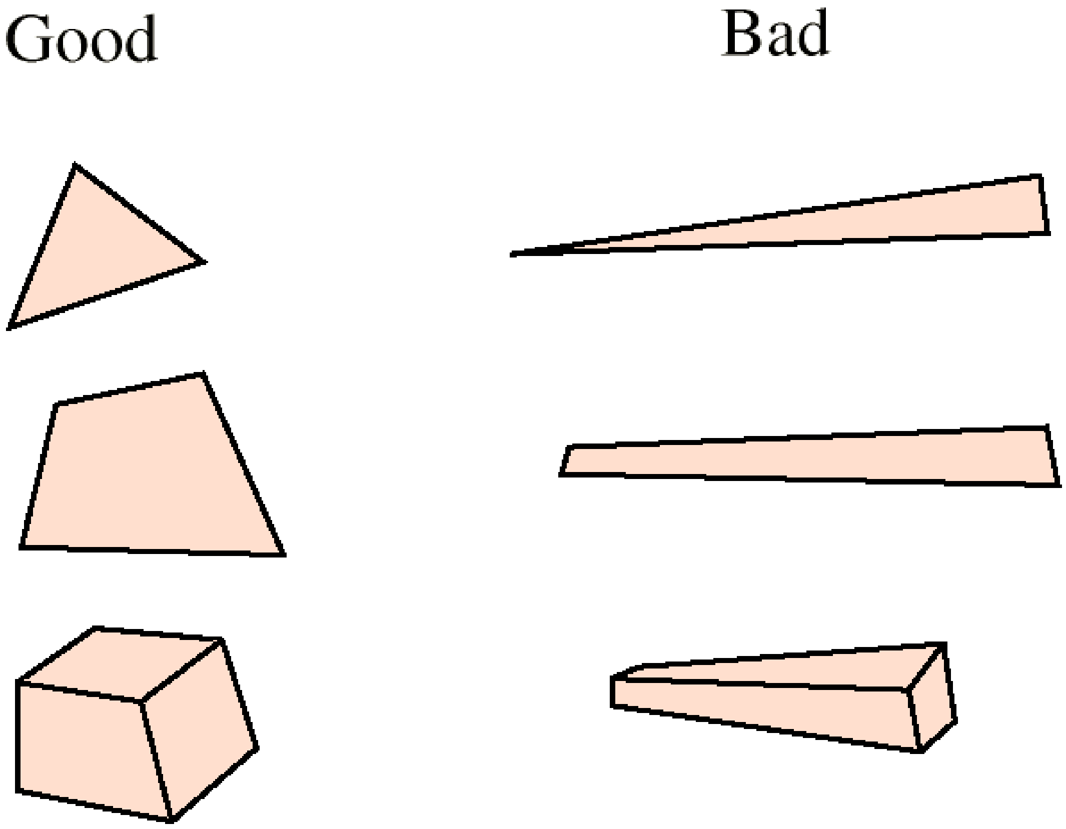


- *Volumetric distortion* (Cont'd)



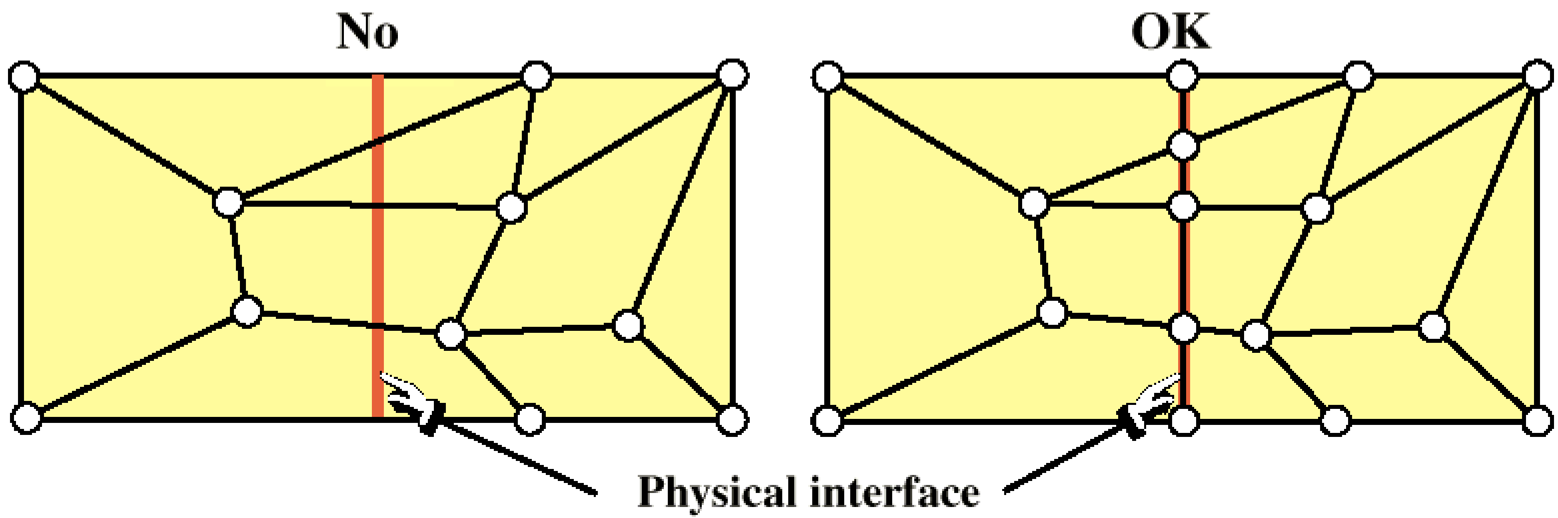
Element distortion

Avoid 2D/3D Elements of Bad Aspect Ratio



Elements Must Not Cross Interfaces

A physical interface, resulting from example from a change in material, should also be an interelement boundary.



Element Geometry Preferences

Other things being equal, prefer

in 2D: Quadrilaterals over Triangles

in 3D: Bricks over Wedges, Wedges over Tetrahedral

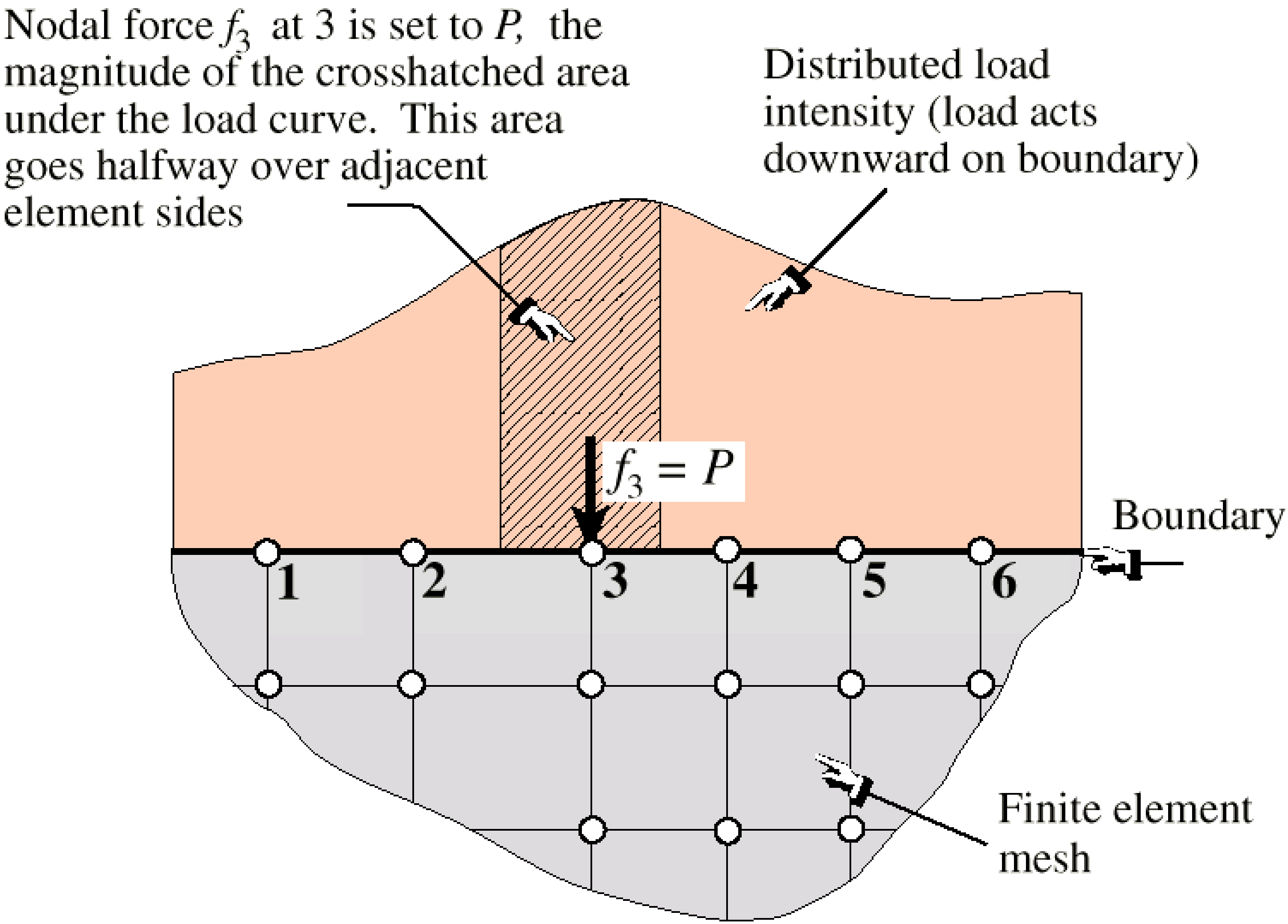
Direct Lumping of Distributed Loads

In practical structural problems, distributed loads are more common than concentrated (point) loads. In fact, one of the objectives of a good design is to avoid or alleviate stress concentrations produced by concentrated forces.

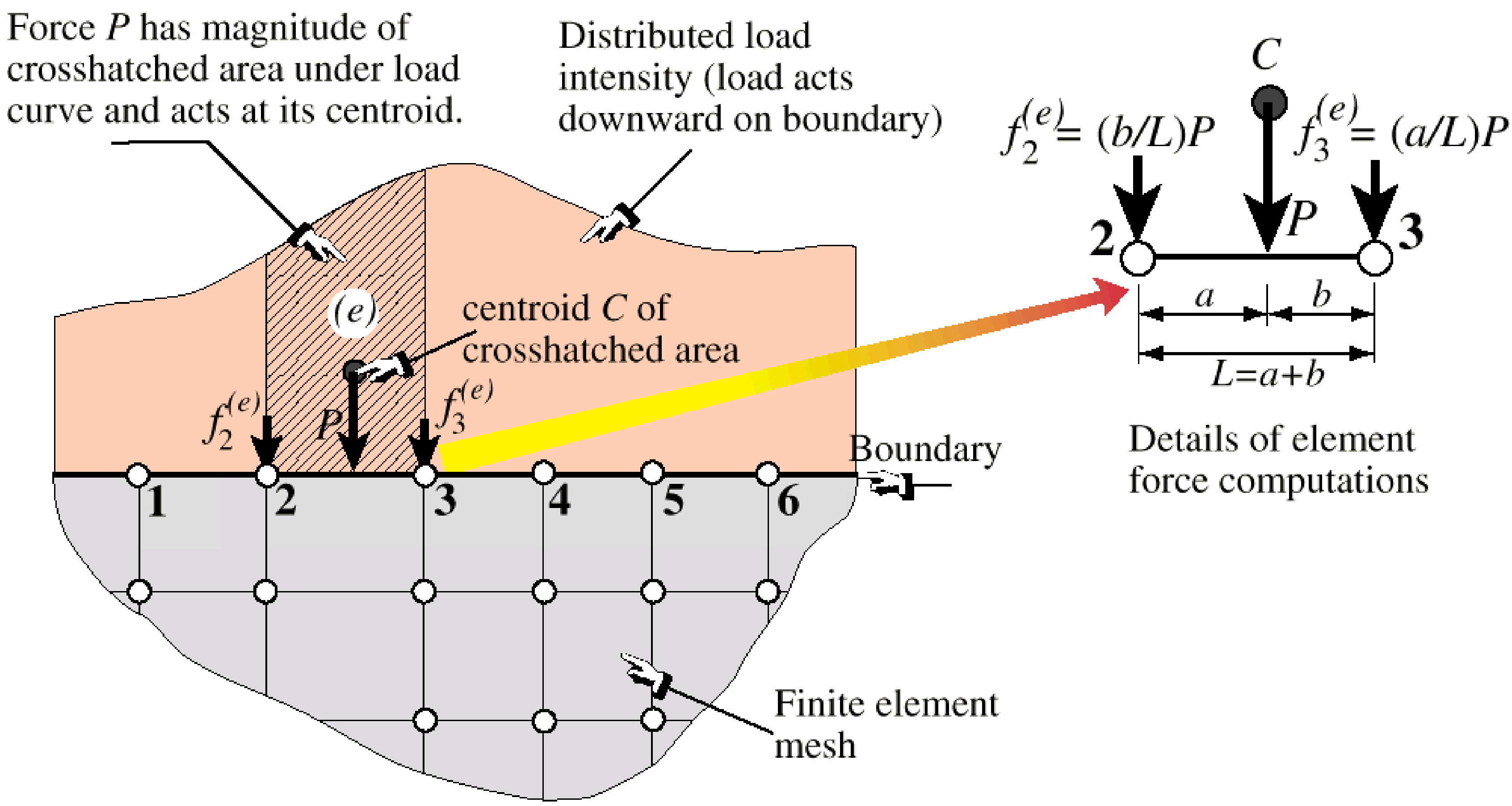
Whatever their nature or source, distributed loads *must be converted to consistent nodal forces* for FEM analysis. These forces eventually end up in the right-hand side of the master stiffness equations.

The meaning of "consistent" can be made precise through variational arguments, by requiring that the distributed loads and the nodal forces produce the same external work. However, a simpler approach called *direct load lumping*, or simply *load lumping*, is often used by structural engineers in lieu of the more mathematically impeccable but complicated variational approach. Two variants of this technique are described below for distributed surface loads.

Node by Node (NbN) Distributed Load Lumping



Element by Element (EbE) Distributed Load Lumping



Boundary Conditions (BCs)

The most difficult topic for FEM program users

Two types $\begin{cases} \text{Essential} \\ \\ \text{Natural} \end{cases}$

1. If a BC involves one or more DOF in a *direct* way, it is *essential* and goes to the Left Hand Side (LHS) of **Ku** = **f**

2. Otherwise it is *natural* and goes to the Right Hand Side (RHS) of **Ku** =**f**

Boundary Conditions in Structural Problems

In mechanical problems, essential boundary conditions are those that involve *displacements* (but not strain-type displacement derivatives). The support conditions for the truss problem furnish a particularly simple example. But there are more general boundary conditions that occur in practice. A structural engineer must be familiar with displacement B.C. of the following types.

Ground or support constraints. Directly restraint the structure against rigid body motions.

Symmetry conditions. To impose symmetry or antisymmetry restraints at certain points, lines or planes of structural symmetry.

Ignorable freedoms. To suppress displacements that are irrelevant to the problem. (In classical dynamics these are called *ignorable coordinates*.) Even experienced users of finite element programs are sometimes baffled by this kind. An example are rotational degrees of freedom normal to shell surfaces.

Connection constraints. To provide connectivity to adjoining structures or substructures, or to specify relations between degrees of freedom. Many conditions of this type can be subsumed under the label *multipoint constraints* or *multifreedom constraints*, which can be notoriously difficult to handle from a numerical standpoint.

Boundary Conditions in Structural Problems

In structural problems, the distinguishes between essential and natural BC is:

if it directly involves the nodal freedoms, such as displacements or rotations, it is essential. Otherwise it is natural.

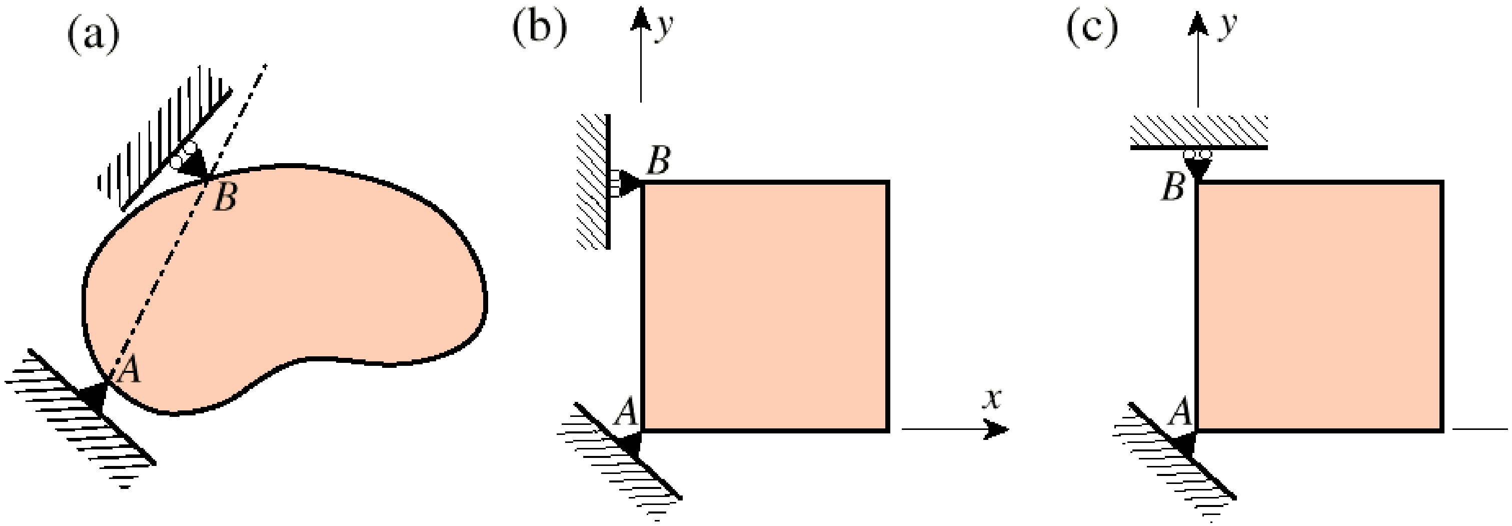
Conditions involving applied loads are natural.

Essential BCs take precedence over natural BCs. The simplest essential boundary conditions are support and symmetry conditions. These appear in many practical problems. More exotic types, such as multifreedom constraints, require more advanced mathematical tools.

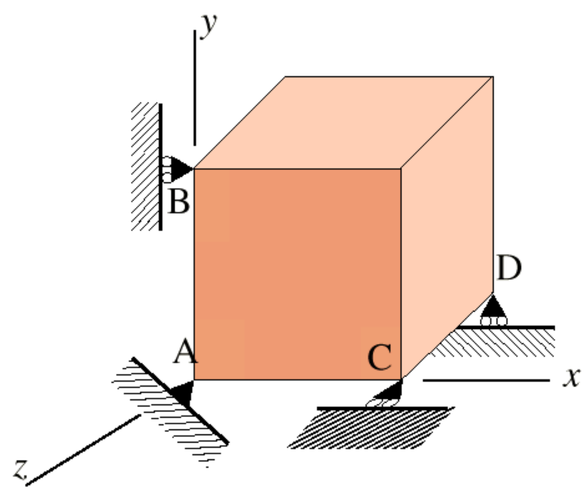
Boundary Conditions in Structural Problems

Minimum Support Conditions to Suppress Rigid Body Motions in 2D

2D

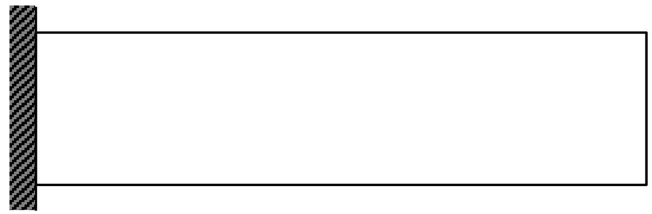


3D

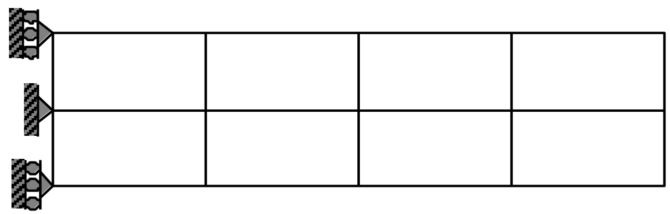


Boundary Conditions in Structural Problems

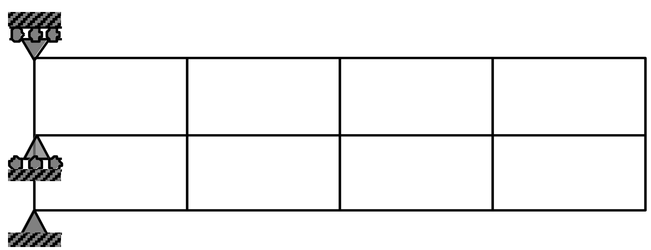
Example: Modelling of Supports



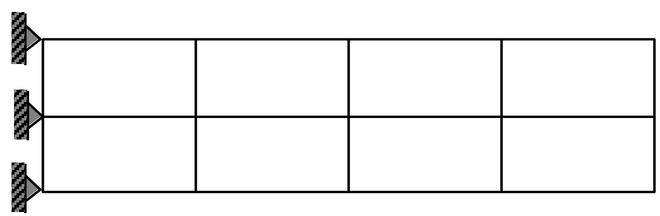
Beam with "built-in end"



a) Full constraint only in the horizontal direction



b) Support provides full constraint only on the lower surface

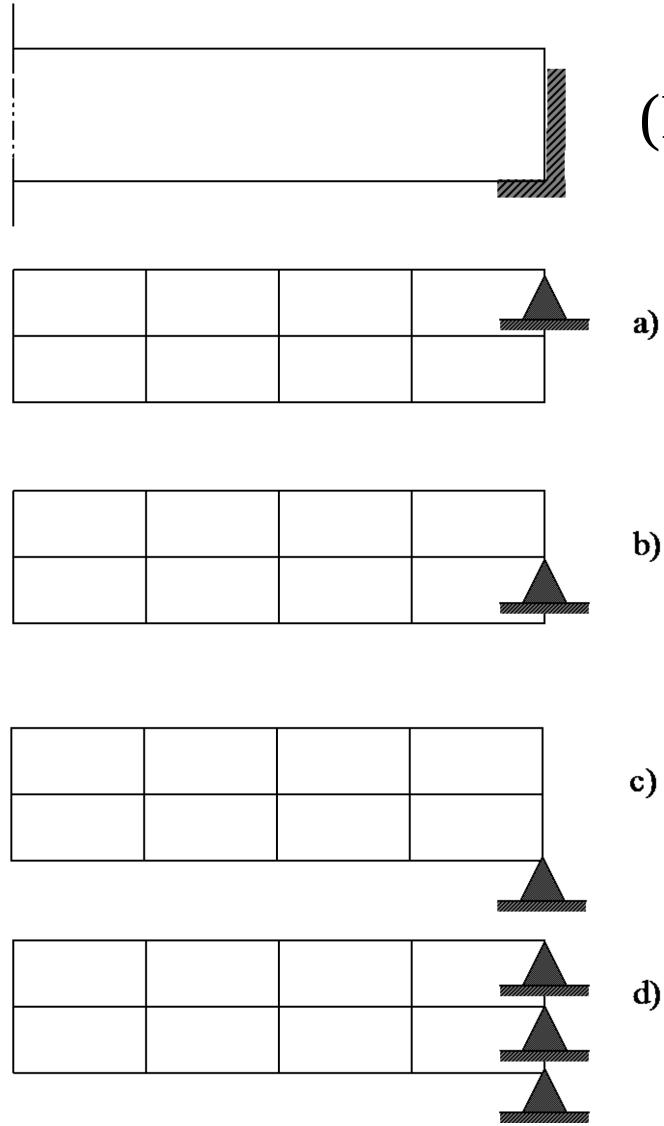


c) Fully clamped support

Boundary Conditions in Structural Problems

Example: Modelling of Supports

(Prop support of beam)



Symmetry

A structure possesses *symmetry* if its components are arranged in a periodic or reflective manner.

Types of Symmetry:

- Mirror (Reflective, bilateral) symmetry
- cyclical (Rotational) symmetry
- Axisymmetry
- Translational (Repetitive) symmetry
- . . .

Cautions:

In vibration and buckling analyses, symmetry concepts, in general, should not be used in FE solutions (works fine in modeling), since symmetric structures often have antisymmetric vibration or buckling modes.

Mirror symmetry

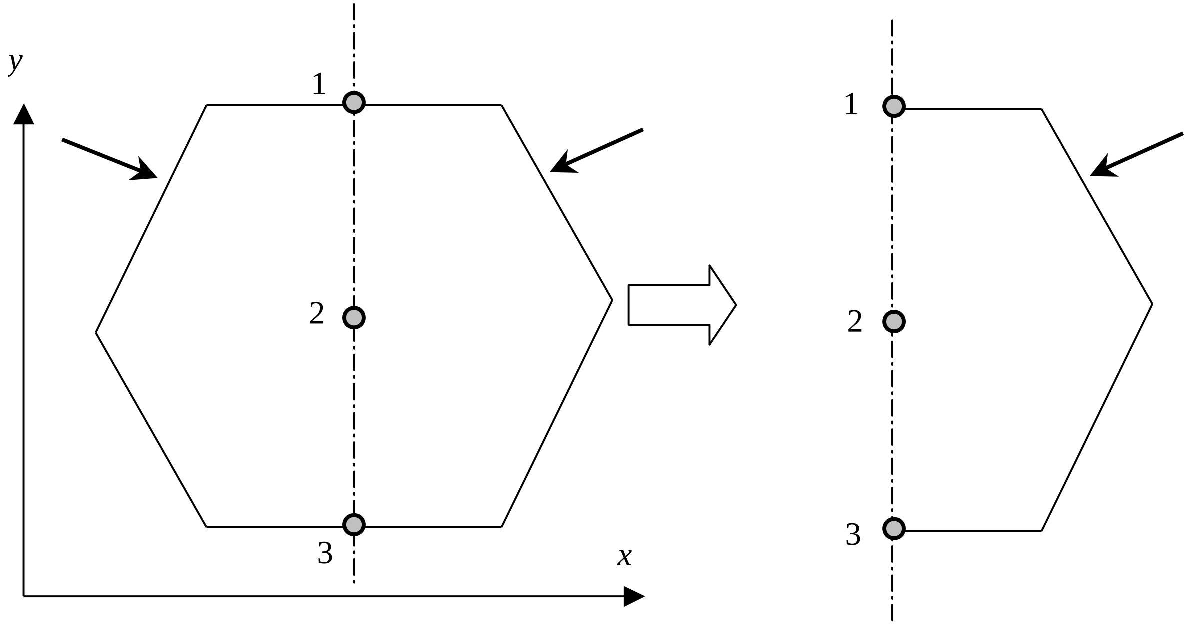
Consider a 2D symmetric solid:

$$u_{1x} = 0$$

$$u_{2x} = 0$$

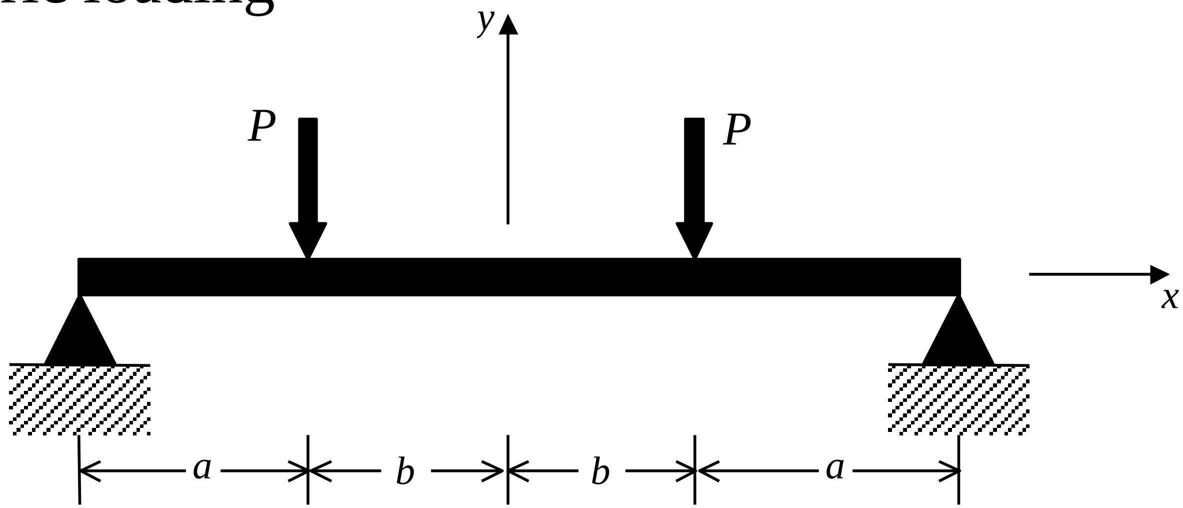
$$u_{3x} = 0$$

Single point constraints (SPC)

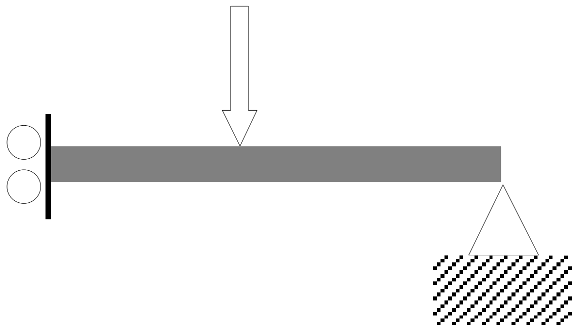


Mirror symmetry

Symmetric loading

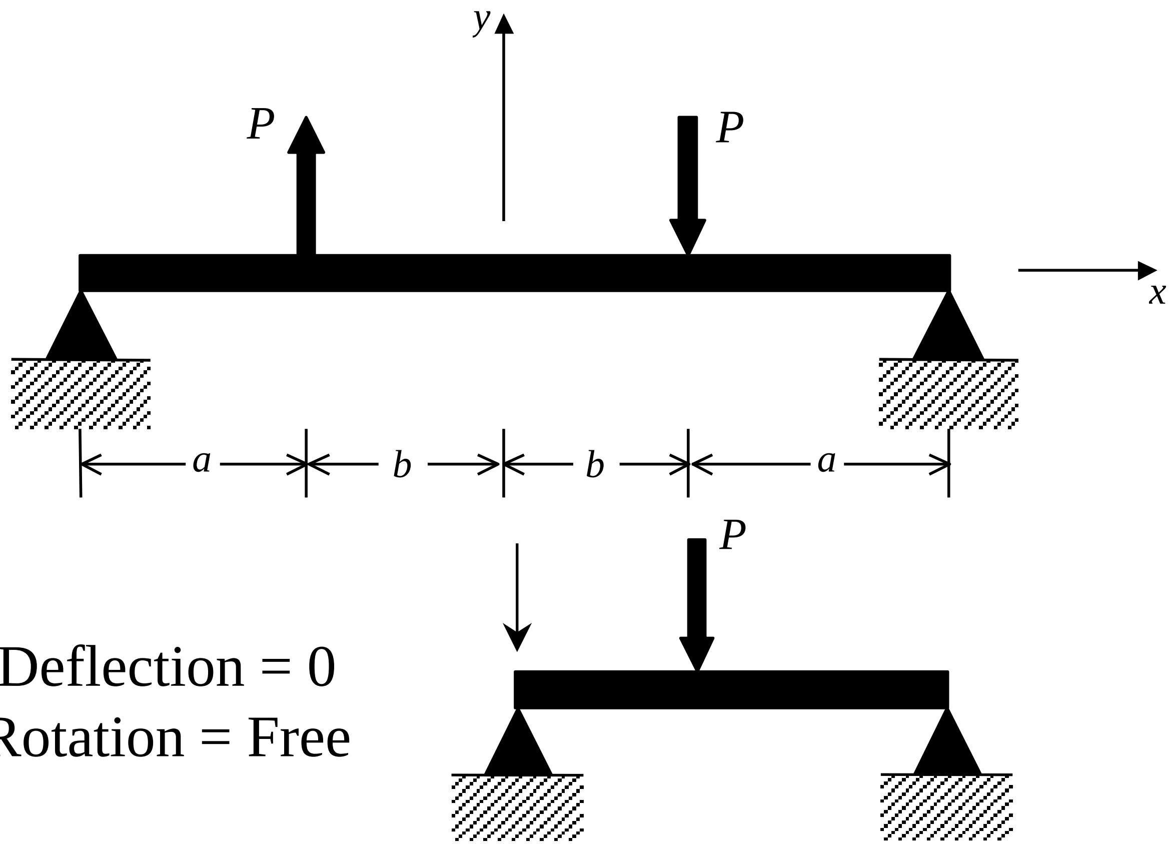


Deflection = Free
Rotation = 0



Mirror symmetry

Anti-symmetric loading



Deflection = 0
Rotation = Free

Mirror symmetry

Symmetric

- No translational displacement normal to symmetry plane
- No rotational components with respect to the axis parallel to symmetry plane

Plane of symmetry	u	v	w	θ_x	θ_y	θ_z
xy	Free	Free	Fix	Fix	Fix	Free
yz	Fix	Free	Free	Free	Fix	Fix
zx	Free	Fix	Free	Fix	Free	Fix

Mirror symmetry

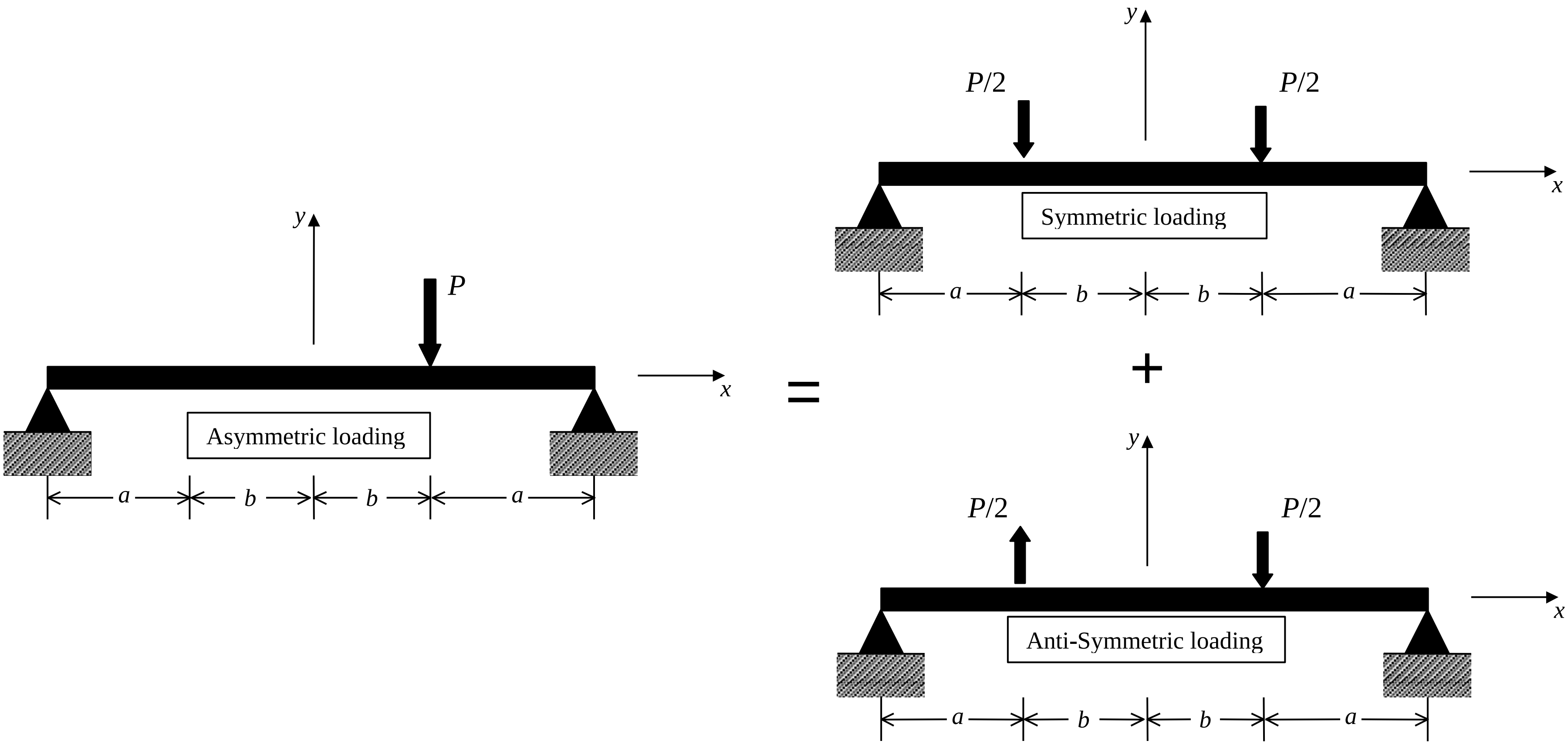
Anti-symmetric

- No translational displacement parallel to symmetry plane
- No rotational components with respect to the axis normal to symmetry plane

Plane of symmetry	u	v	w	θ_x	θ_y	θ_z
xy	Fix	Fix	Free	Free	Free	Fix
yz	Free	Fix	Fix	Fix	Free	Free
zx	Fix	Free	Fix	Free	Fix	Free

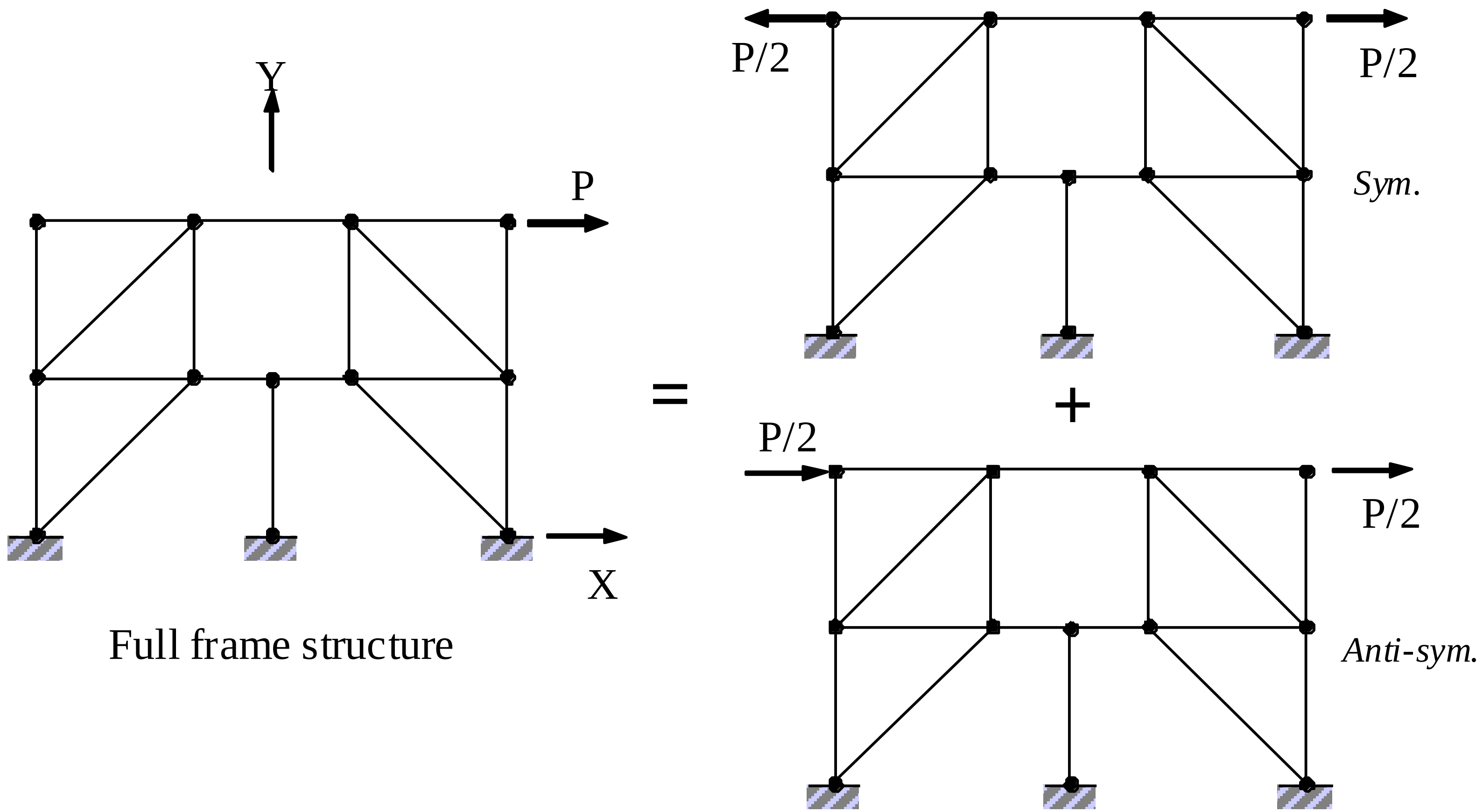
Mirror symmetry

- Any load can be decomposed to a symmetric and an anti-symmetric load



Symmetry

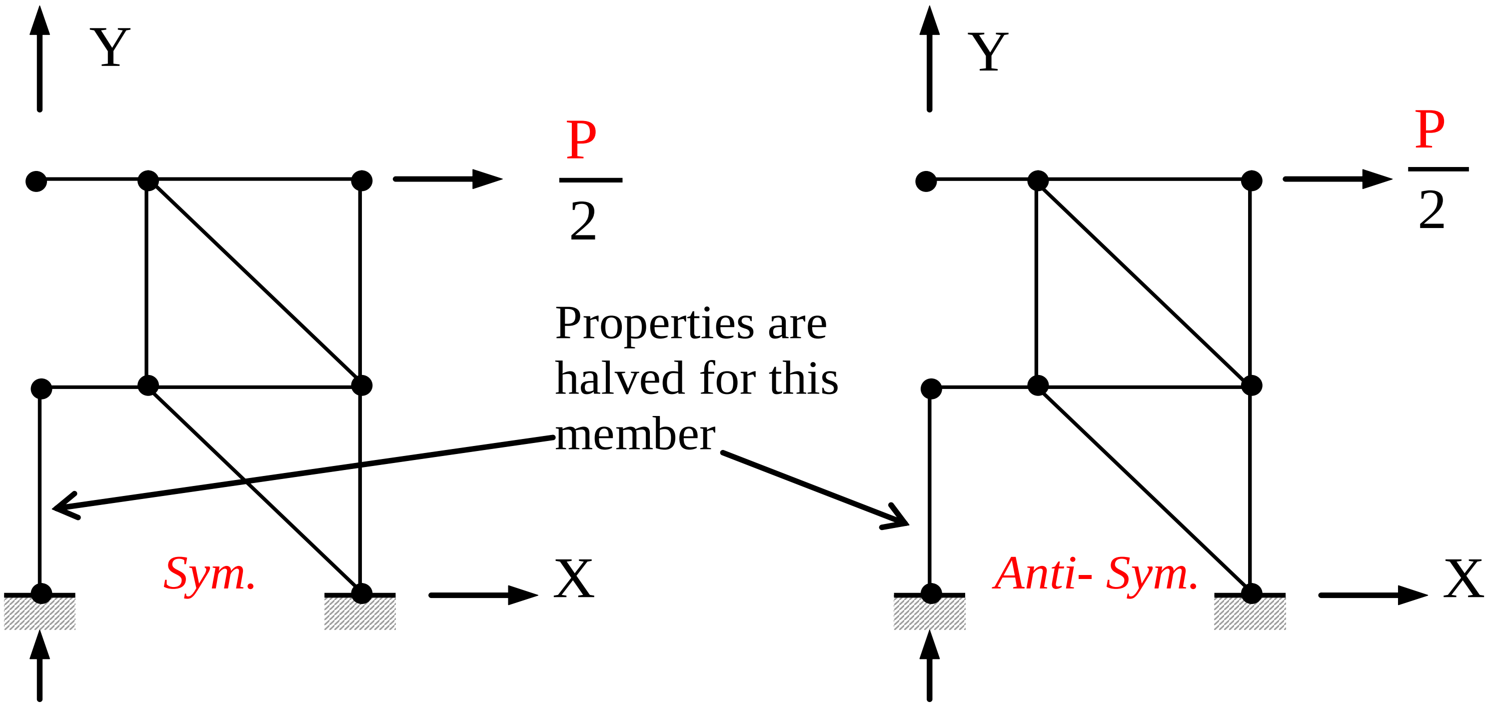
Mirror symmetry



Full frame structure

Symmetry

Mirror symmetry

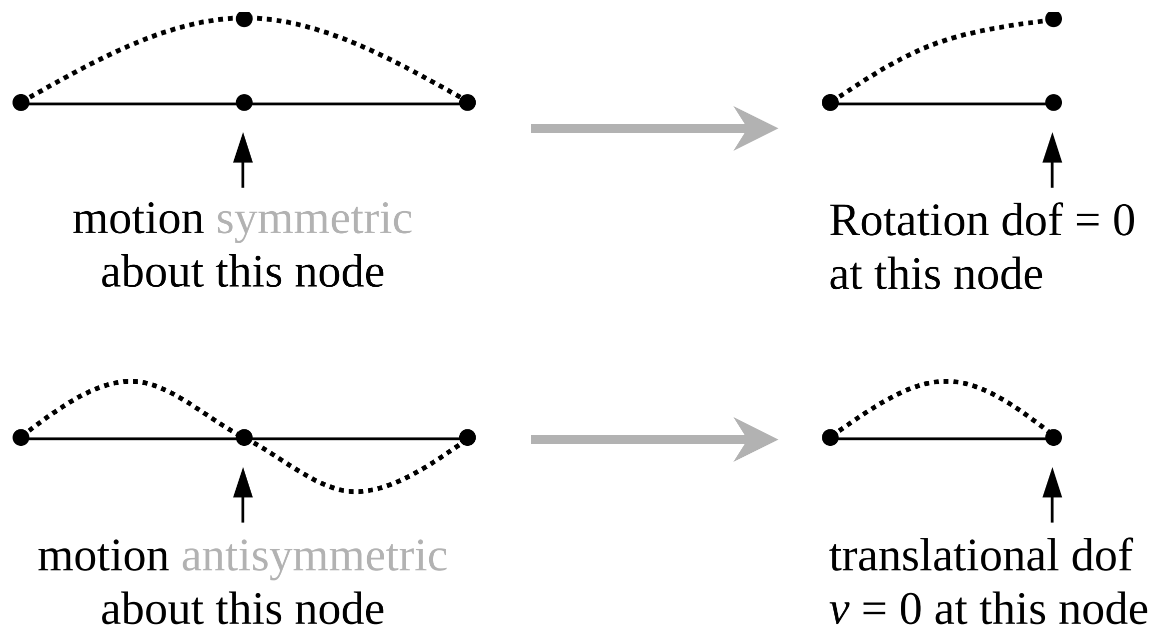


All nodes on this line fixed against the horizontal displacement and rotation.

All nodes on this line fixed against vertical displacement.

Mirror symmetry

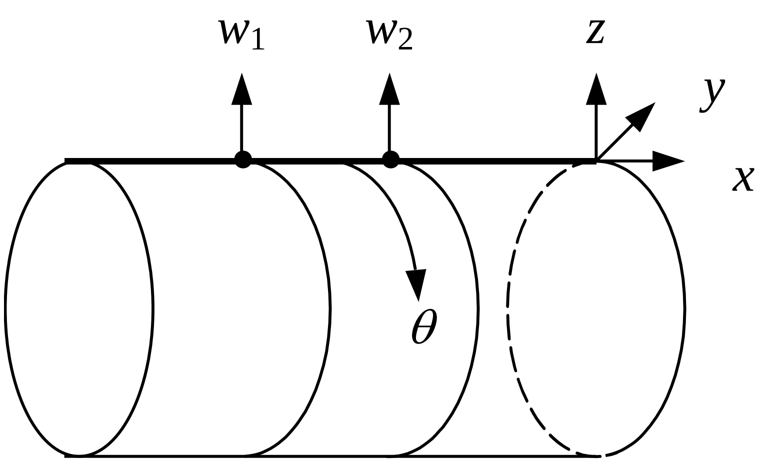
- Dynamic problems (e.g. two half models to get full set of eigenmodes in eigenvalue analysis)



Axial symmetry

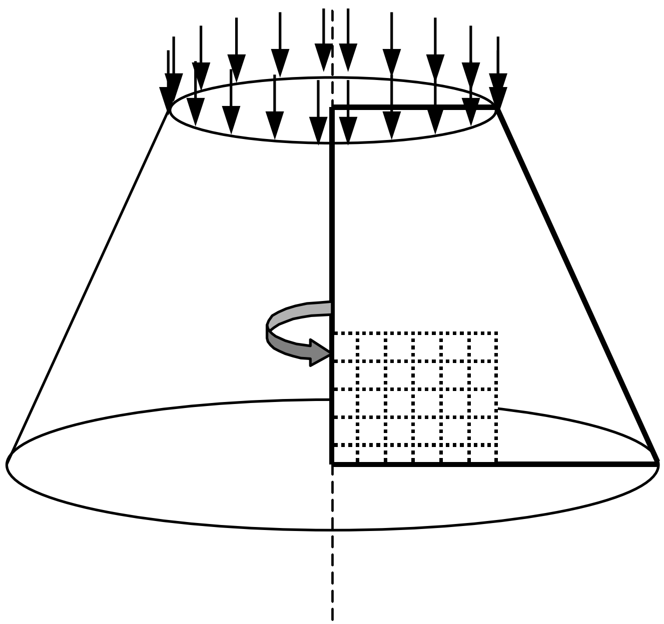
- Use of 1D or 2D axisymmetric elements

  Formulation similar to 1D and 2D elements except the use of polar coordinates



$$w = W \sin\theta$$

Cylindrical shell using 1D axisymmetric elements



3D structure using 2D axisymmetric elements

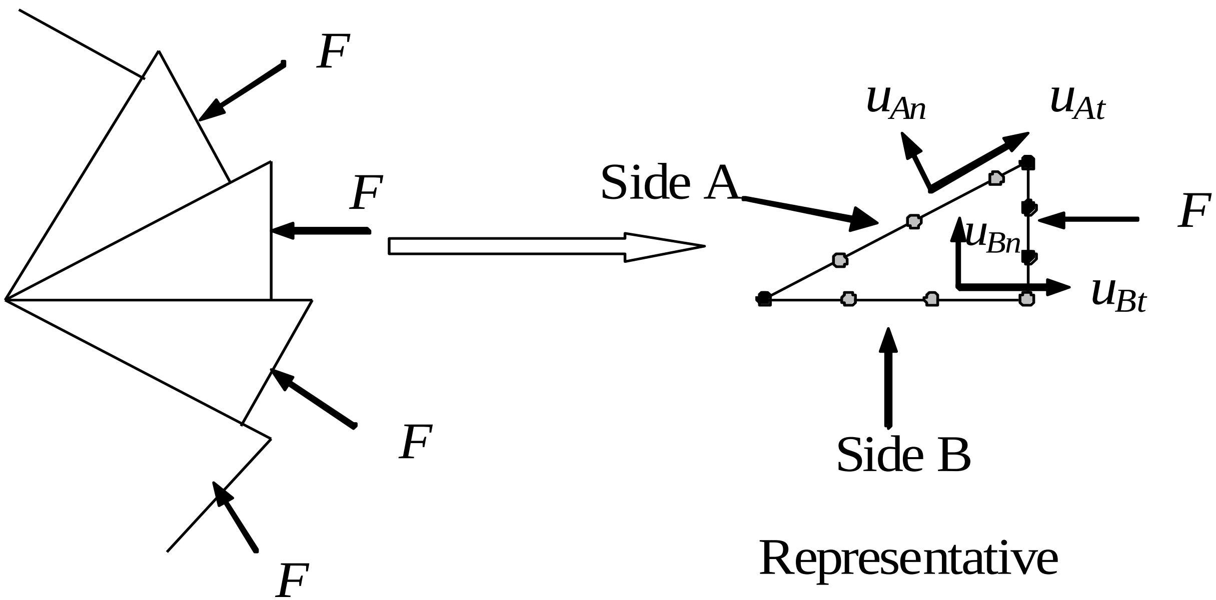
Symmetry

Cyclic symmetry

$$u_{An} = u_{Bn}$$

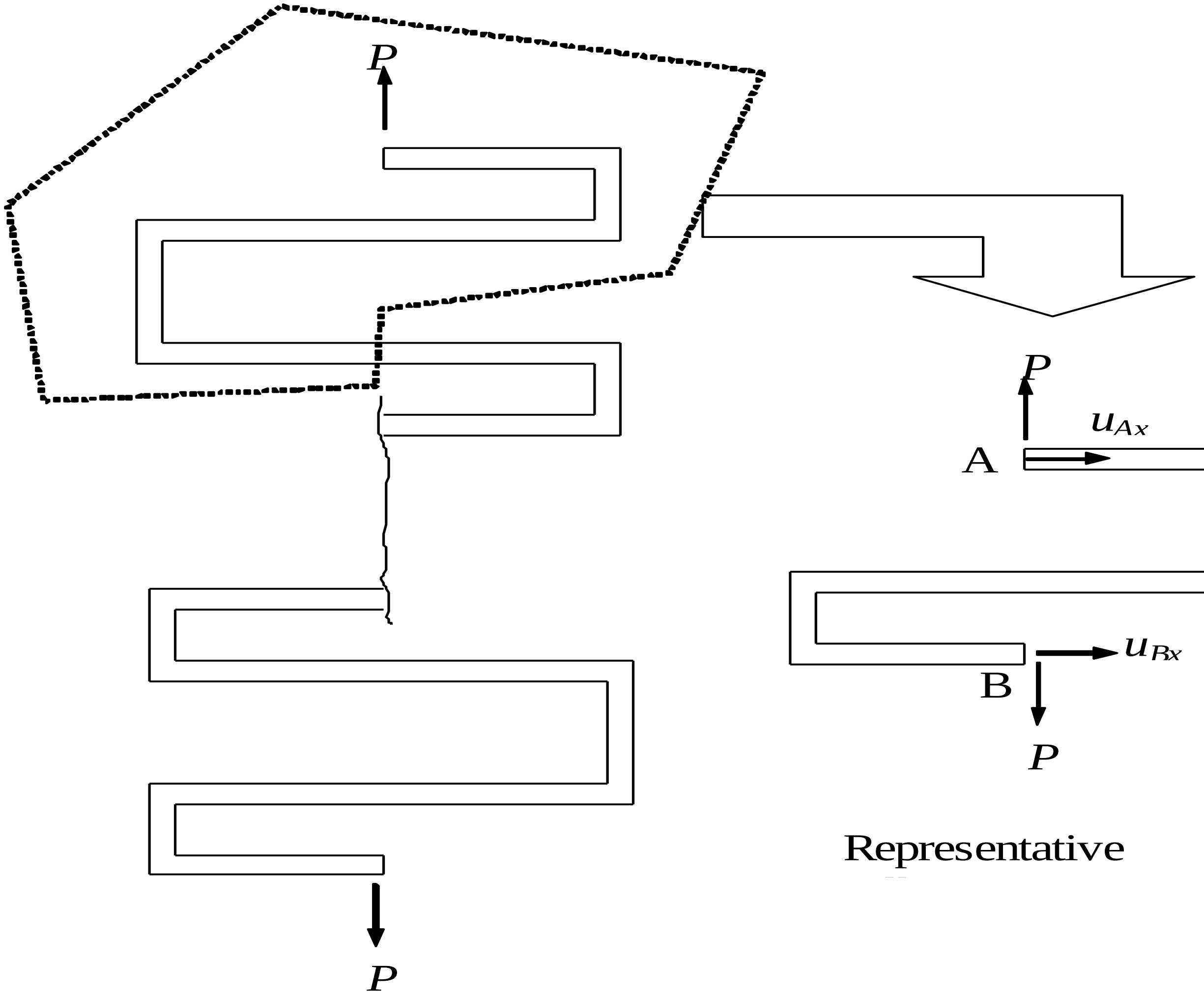
$$u_{At} = u_{Bt}$$

Multipoint constraints (MPC)



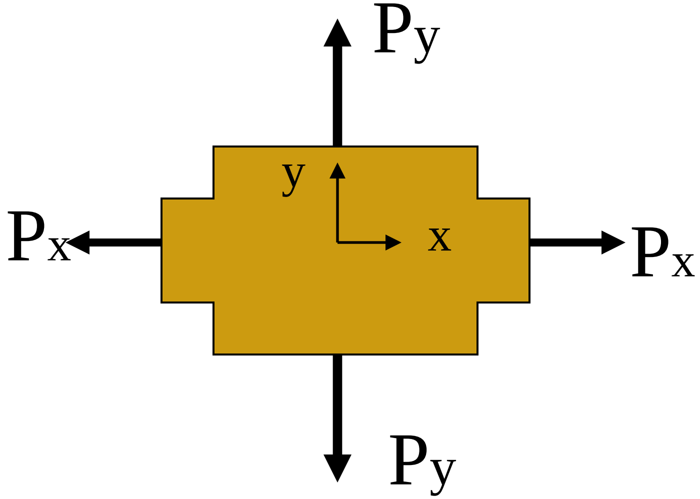
Repetitive symmetry

$$u_{Ax} = u_{Bx}$$

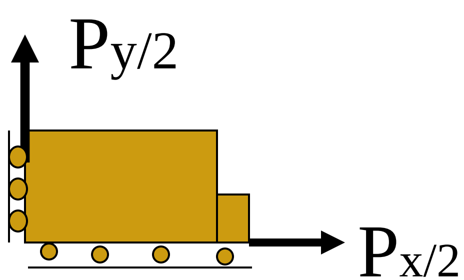


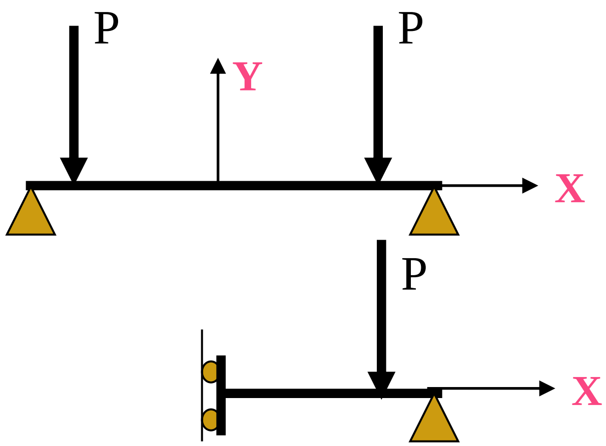
Symmetry

Example of Symmetry

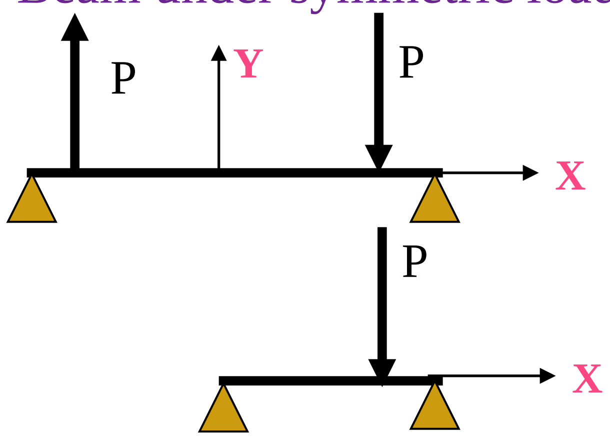


Plane structure having
Reflective symmetry



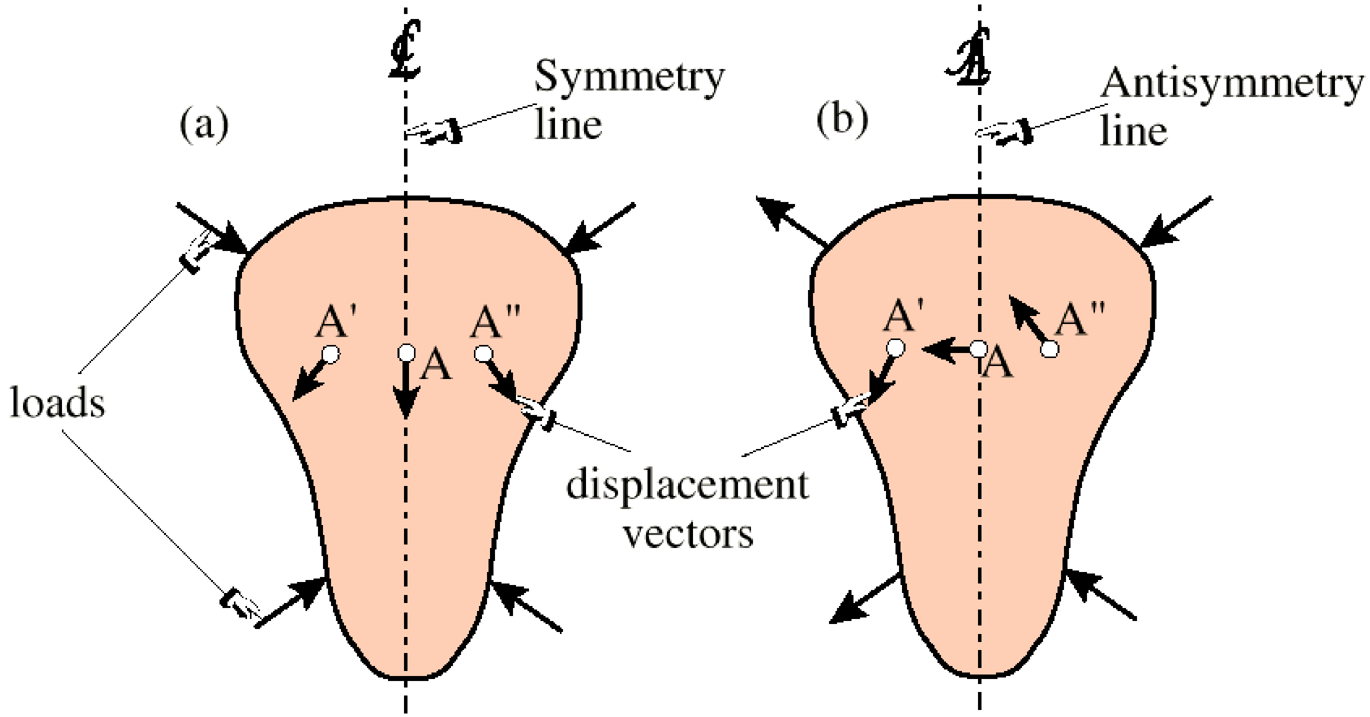


Beam under symmetric load

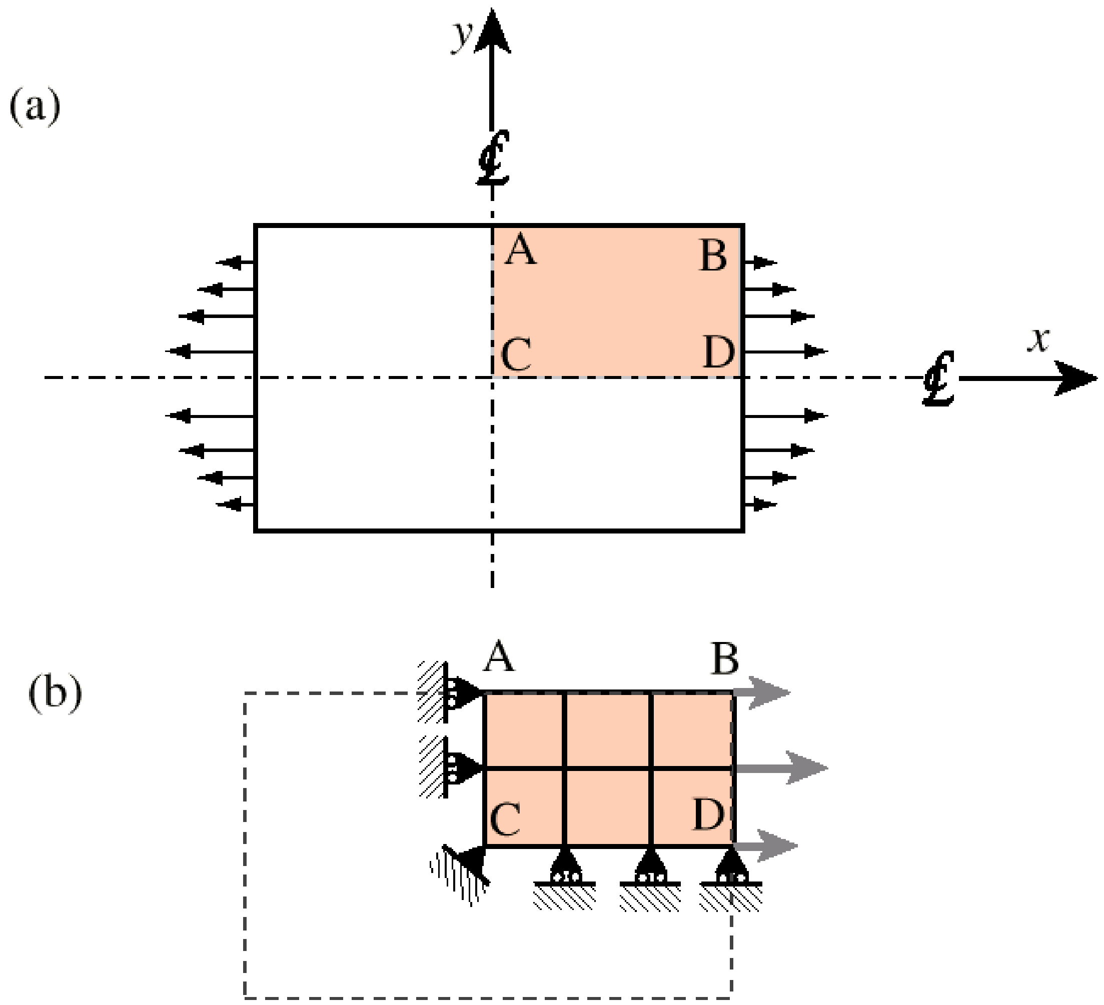


Beam under unsymmetric load

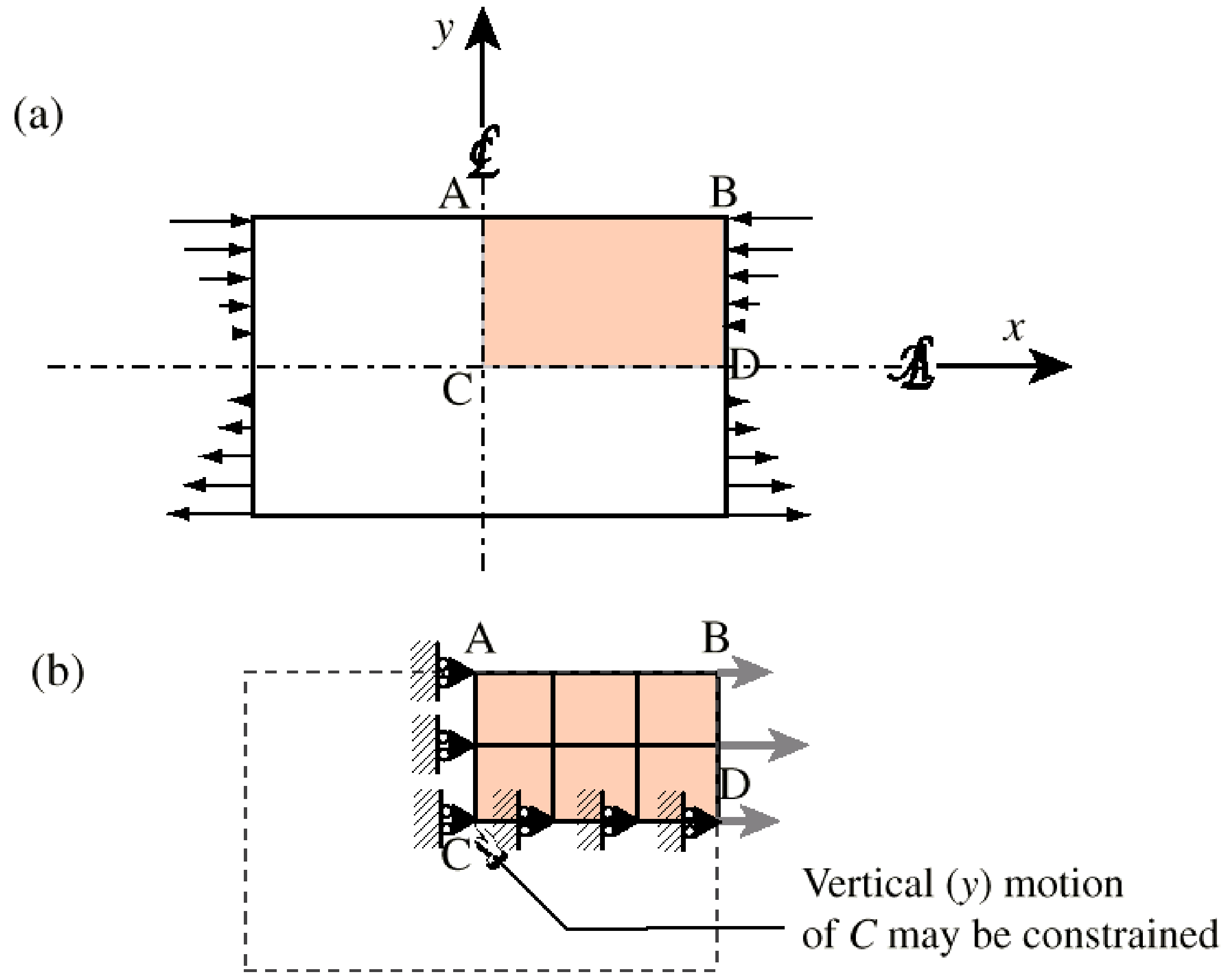
Example of Symmetry and Antisymmetry



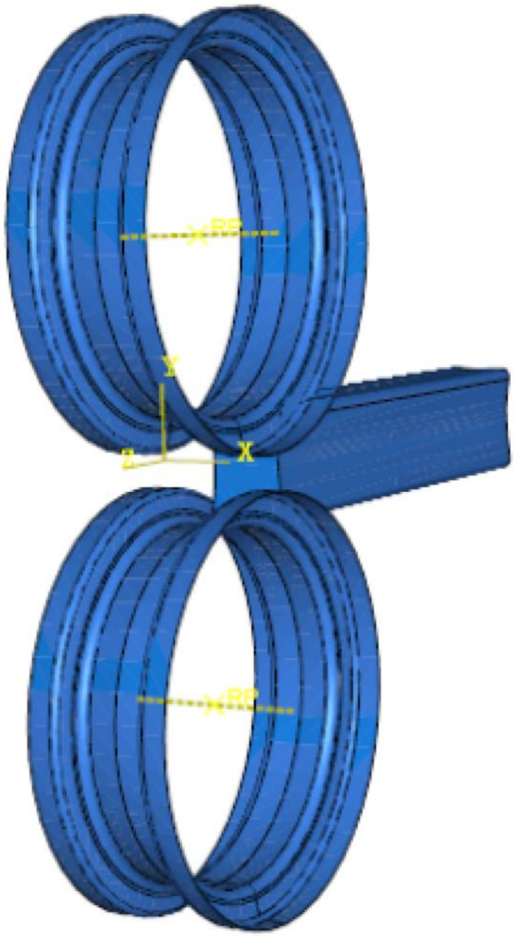
Example of Application of Symmetry BCs

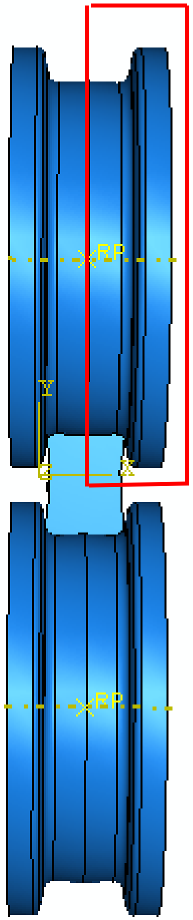


Example of Application of Antisymmetry BCs



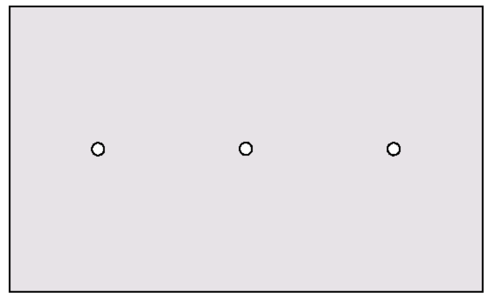
Example of Symmetry





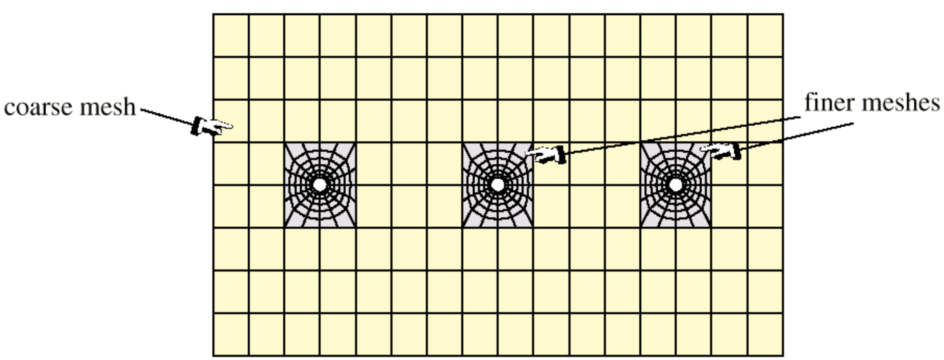
Global-Local Analysis (an instance of Multiscale Analysis)

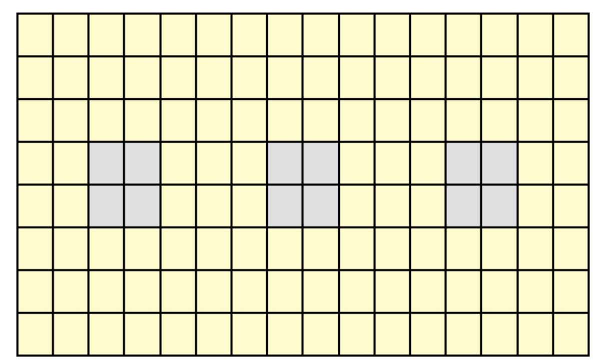
Complex engineering systems are often modeled in a multilevel fashion like substructure, … A related, but not identical, technique is multiscale analysis.



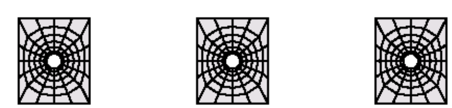
Example structure: panel with small holes

1- The whole system is first analyzed as a global entity, neglecting the detail
2- Local details are then analyzed using the results of the global analysis as boundary conditions.





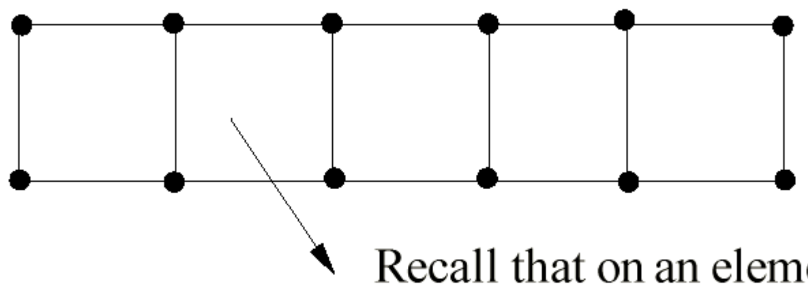
Global analysis with a coarse mesh, ignoring holes, followed by local analysis of the vicinity of the holes with finer meshes:



Nature of Finite Element Solutions

- FE Model – A mathematical model of the real structure, based on many approximations.
- Real Structure – Infinite number of nodes (physical points or particles), thus infinite number of DOF's.
- FE Model – finite number of nodes, thus finite number of DOF's.

⇒ Displacement field is controlled (or constrained) by the values at a limited number of nodes.



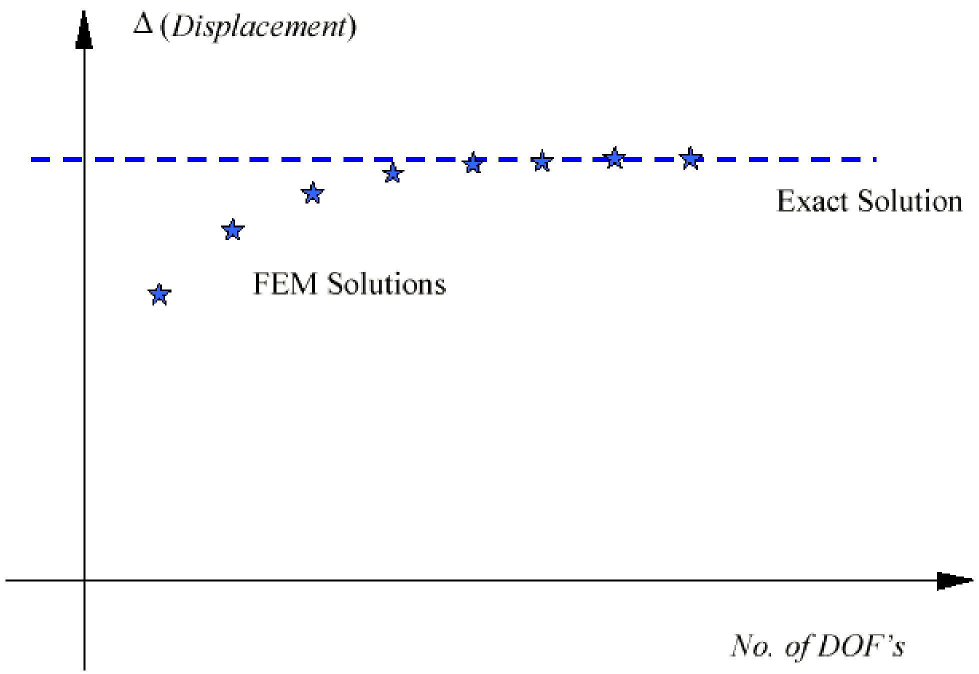
Recall that on an element :

$$u = \sum_{\alpha=1}^{4} N_{\alpha} u_{\alpha}$$

Stiffening Effect:

- FE Model is stiffer than the real structure.
- In general, displacement results are smaller in magnitudes than the exact values.

Hence, FEM solution of displacement provides a *lower bound* of the exact solution.



Numerical Error

Error $\neq$ Mistakes in FEM (modeling or solution).

Types of Error:

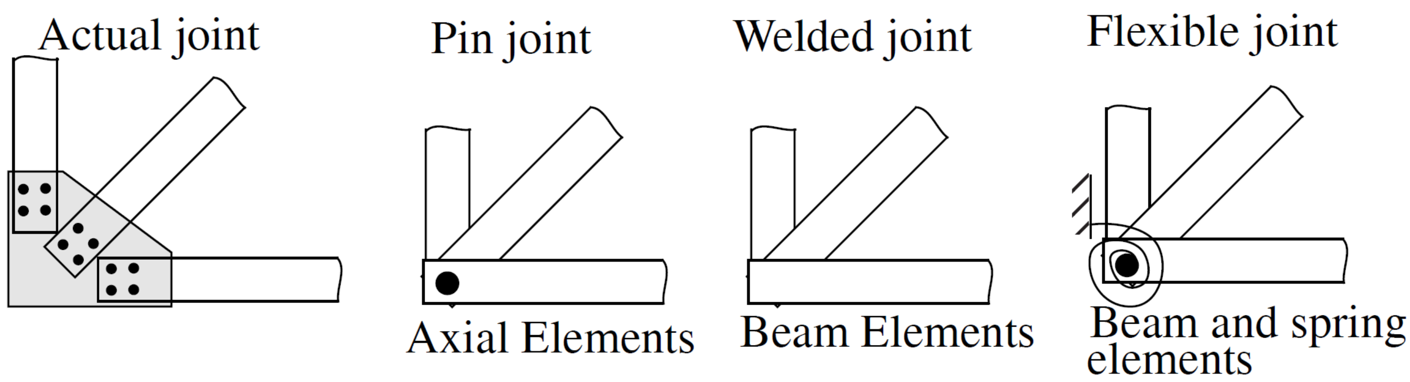
- Modeling Error (beam, plate … theories)
- Discretization Error (finite, piecewise …)
- Numerical Error (in solving FE equations)

Modeling error

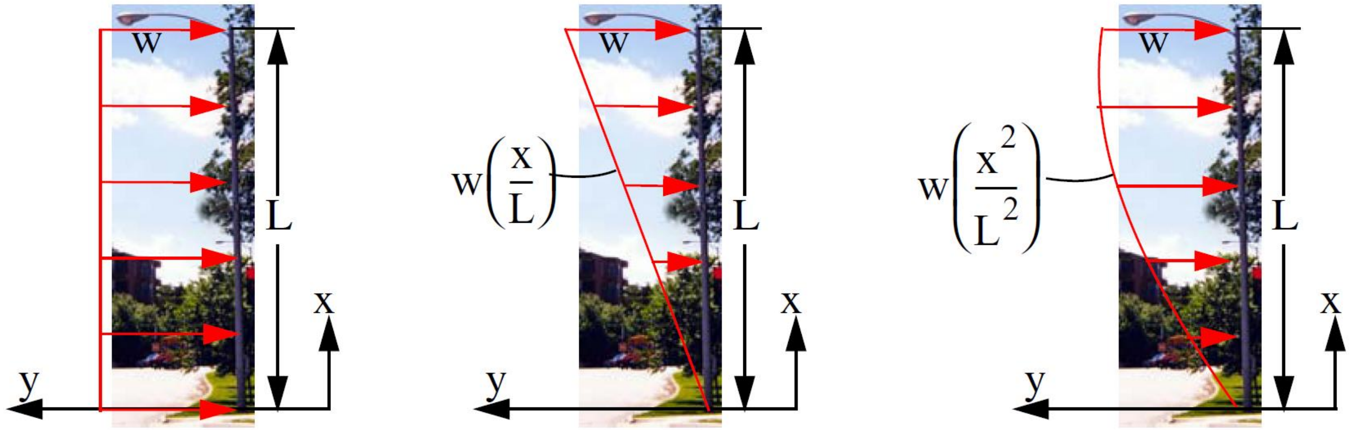
Error that arise from the description of the boundary value problem (BVP): Geometric description, material description, loading, boundary conditions, type of analysis.

- What physical details are important in the BVP description?

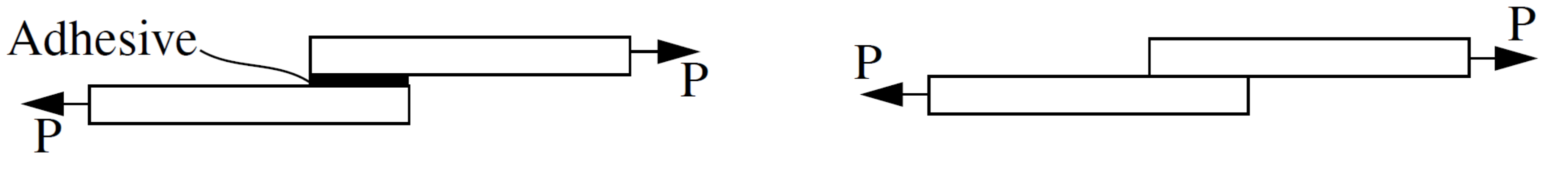
Should a mechanically fastened joint be modeled as a pin joint, welded joint, or a flexible joint.



How should the load be modeled?



Should the properties of the adhesive be included or ignored in a bonded joint?

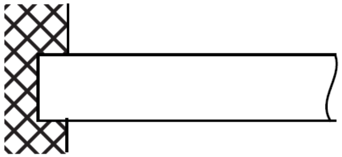


Modeling Error

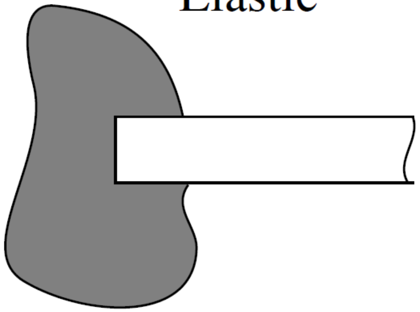
Should the material be modeled as isotropic or orthotropic?

How should the support be modeled? i.e., what are the appropriate boundary conditions.

Fixed

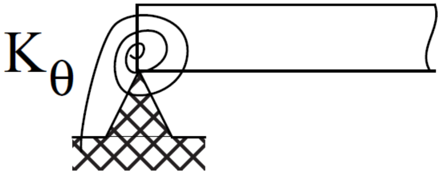


Elastic



$K_\theta = 0$ Simple support

$K_\theta = \infty$ Fixed support

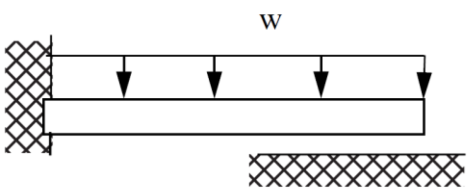


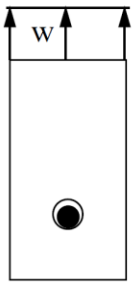
Modeling Error

- What type of analysis should be conducted?

Should you conduct a linear or non-linear analysis?

1. Material non-linearity: Stress and strain are non-linearly related.
2. Geometric non-linearity: Strain and displacement non-linearly related. (large deformation or strain)
3. Contact problem: The contact length changes with load.
   (*i*) No friction.
   (*ii*) With friction—need the slip ($F_f=\mu m$) and no slip boundary($F_f<\mu m$).





Should buckling analysis be conducted?

For time dependent problems should you conduct a dynamic or quasi-static analysis? Should the material be modeled as elastic or viscoelastic?

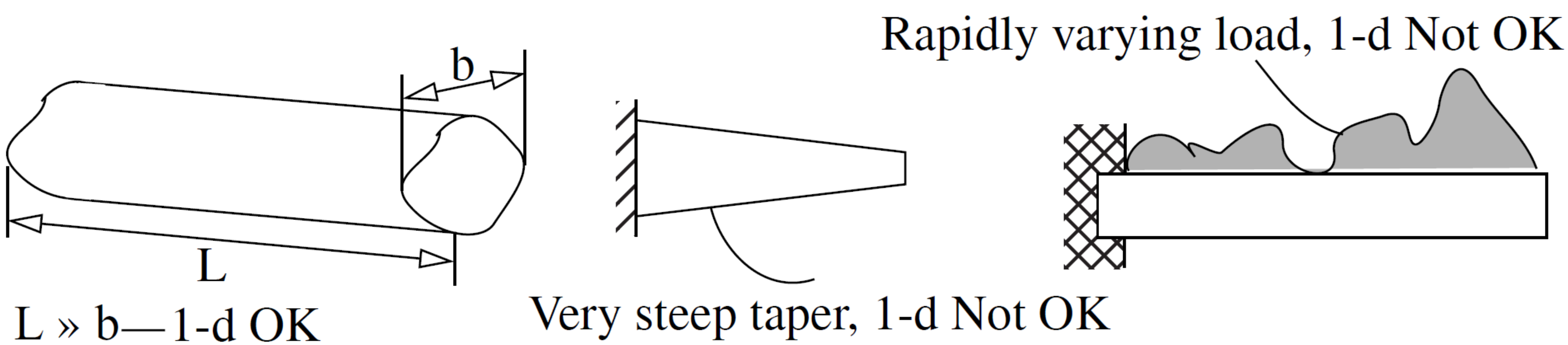
Discretization Error

- Errors that arises from creation of the mesh.

➢Elements in FEM are based on analytical models. All assumptions that are made in the analytical models are applicable to FEM elements.

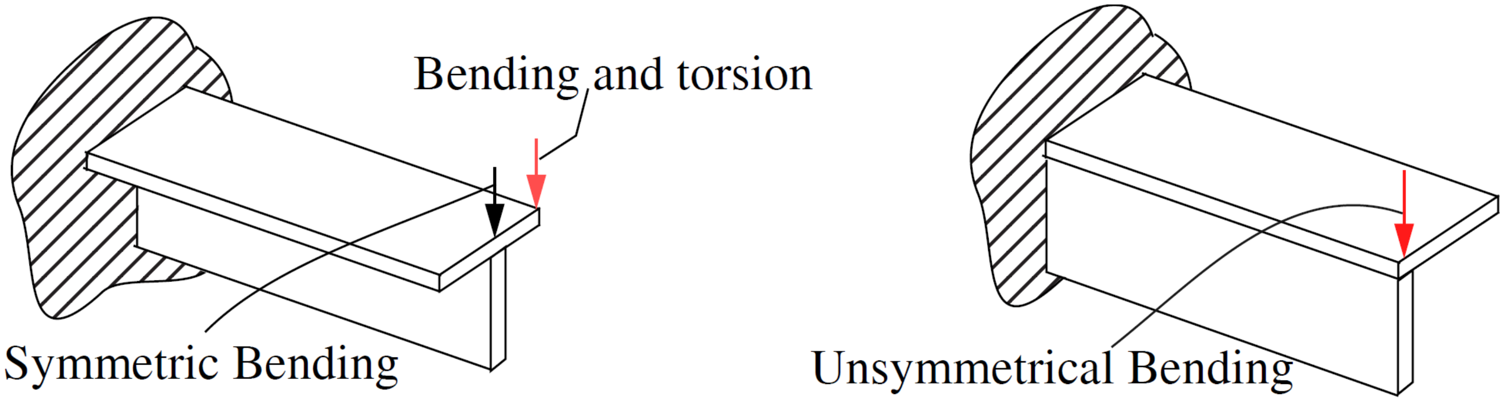
- What type of elements should be used?

➢Should 1-d element be used?



Discretization Error

➢Should beam element, which is based on symmetric bending, be used?



- What type of 2-d (plane stress, plane strain) or 3-d element should you use.?
- What mesh density should you use?

➢Too fine a mesh results in large computer time that may prevent optimization or parametric studies or non-linear analysis. Too coarse a mesh may result in high inaccuracies. Start with a coarse mesh, study the results and then refine the mesh as needed.

Discretization Error

- How accurately should the geometry be modeled?

➢Errors from modeling of geometric are generally small. For the same computational effort higher returns in accuracy are obtained in better modeling of displacement-Isoparametric elements are adequate.

Numerical Error

Errors that arise from finite digit arithmetic and use of numerical methods.

- Integration error

➢Few Gauss points leads to numerical instabilities. Large number of Gauss points are computationally expensive and may result in overly stiff elements leading to higher errors.

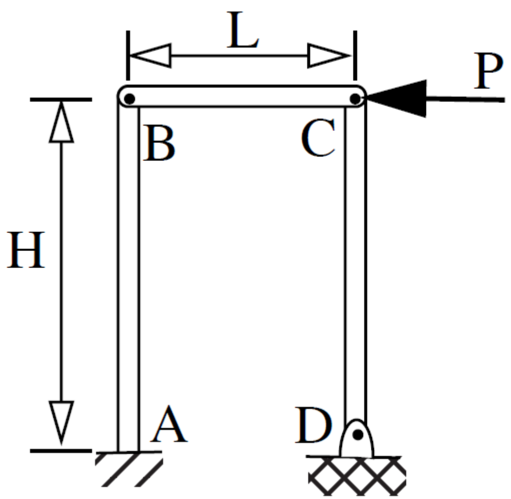
- Round off error

➢The finite digit arithmetic causes these errors, but the growth of round off errors are dictated by several factors. Need to avoid: adding or subtracting very large and very small numbers; dividing by small numbers.

(i) The manner in which algorithms are written in the computer codes. Non-dimensionalizing the problem will always help.

(ii) Large differences in physical dimensions.

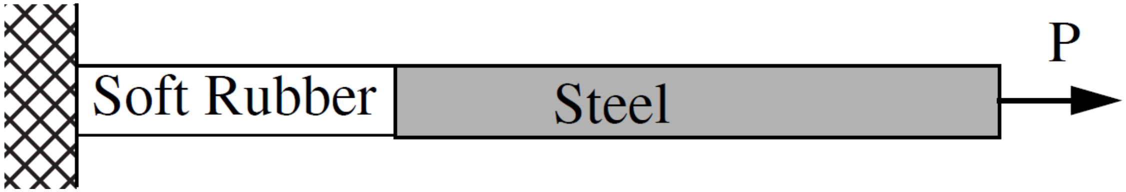
Numerical Error



$\frac{H}{L} \to \infty \text{Large}$ $\quad K_{BC} \gg K_{AB}$ $\quad K_{BC} \gg K_{CD}$

Can BC be modeled as rigid?

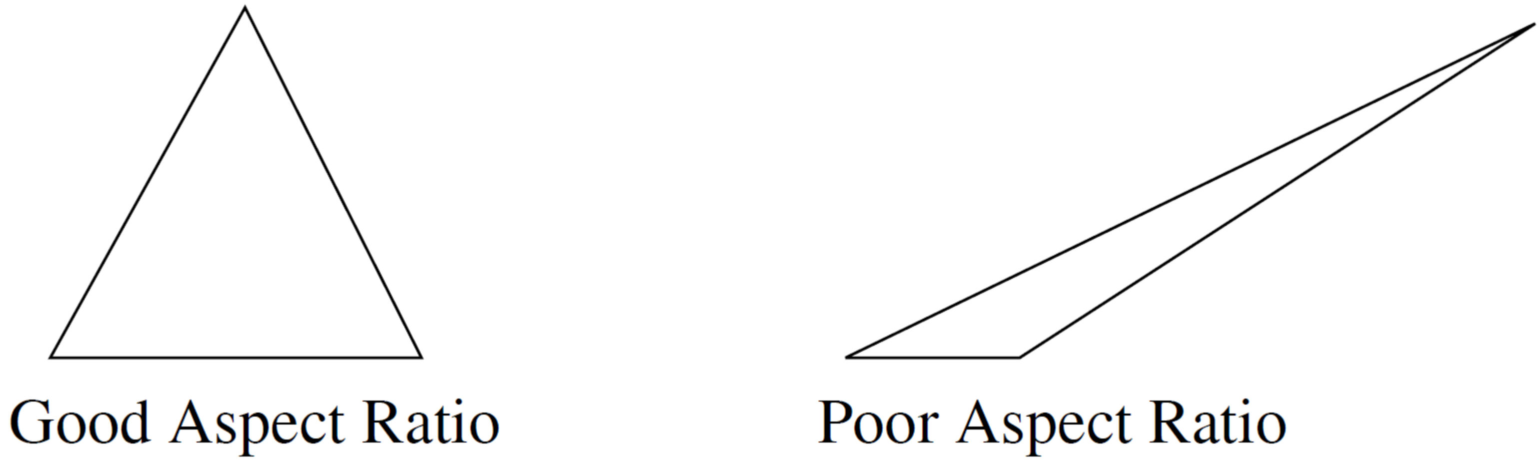
(iii) Large differences in stiffness caused my large differences in material properties (or dimensions).



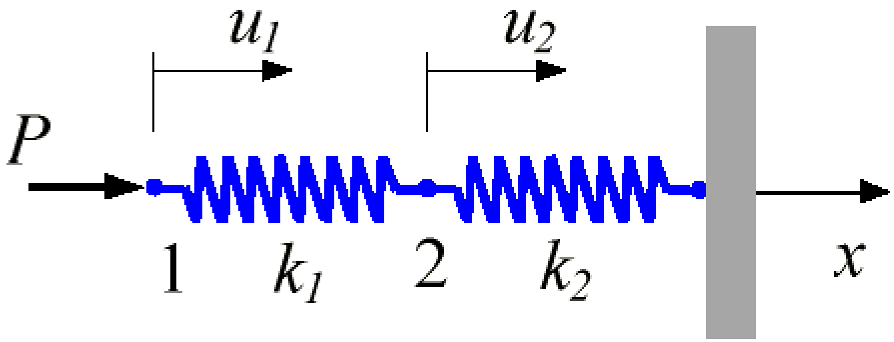
Can steel be modeled as rigid?

(iv) Elements with poor aspect ratio: ratio of largest to smallest dimension in an element.

Numerical Error



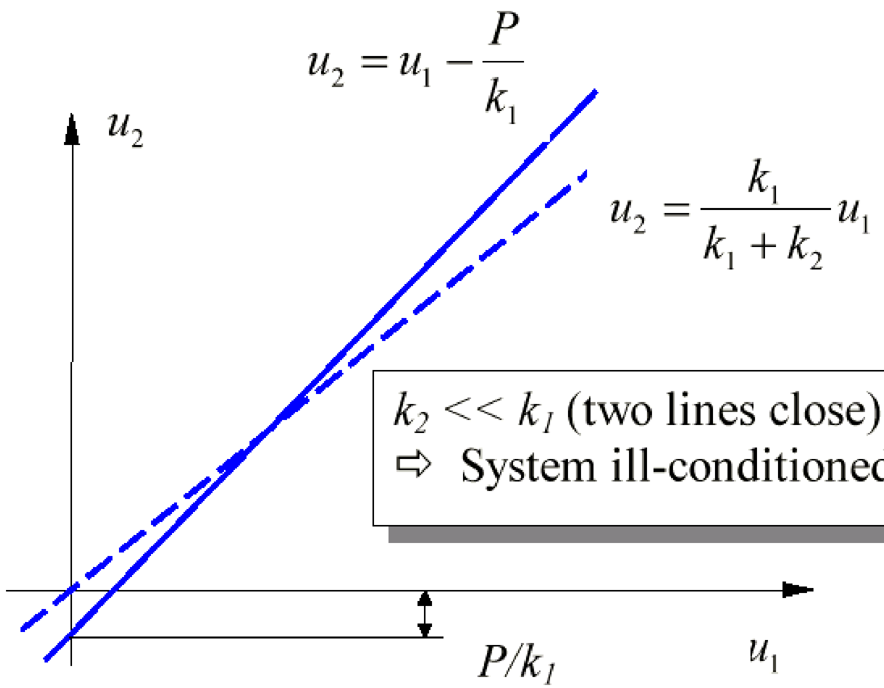
Example (numerical error):



FE Equations:

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1+k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P \\ 0 \end{Bmatrix} \quad \Rightarrow \quad Det\,\mathbf{K} = k_1 k_2 \,.$$

The system will be *singular* if k_2 is small compared with k_1.



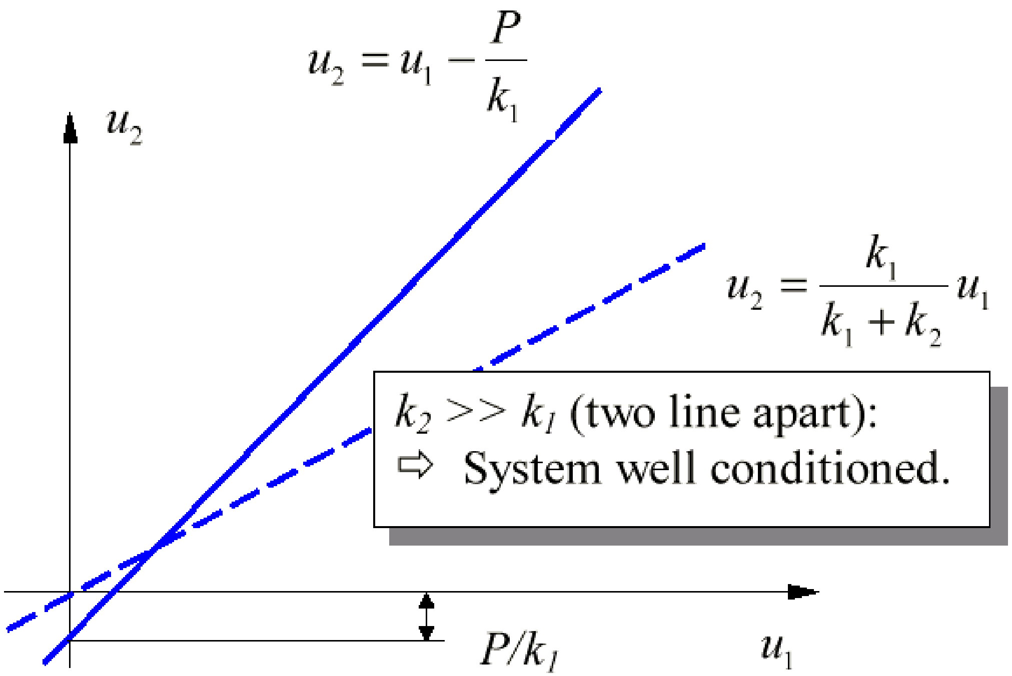
$k_2 << k_1$ (two lines close):
⇨ System ill-conditioned.

Numerical Error

$$\begin{bmatrix} 2 & 6.00001 \\ 2 & 6 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 8.00001 \\ 8 \end{bmatrix} \Rightarrow \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 1 \\ 1 \end{bmatrix}$$

$$\begin{bmatrix} 2 & 6.00001 \\ 2 & 6 \end{bmatrix}\begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} 8.00002 \\ 8 \end{bmatrix} \Rightarrow \begin{bmatrix} u_1 \\ u_2 \end{bmatrix} = \begin{bmatrix} -2 \\ 2 \end{bmatrix}$$

Numerical Error



- ❖ Large difference in stiffness of different parts in FE model may cause ill-conditioning in FE equations. Hence giving results with large errors.
- ❖ Ill-conditioned system of equations can lead to large changes in solution with small changes in input (right hand side vector).

Convergence Requirements for Finite Element Discretization

Convergence: discrete (FEM) solution approaches the analytical (math model) solution in some sense

Convergence = Consistency + Stability

Further Breakdown of Convergence Requirements

- Consistency
  - Completeness *individual elements*
  - Compatibility *element patches*
- Stability
  - Rank Sufficiency *individual elements*
  - Positive Jacobian *individual elements*

The Variational Index m

Bar

$$\Pi[u] = \int_0^L \left(\tfrac{1}{2}\, u' E A u' - q u\right) dx$$

$m = 1$

Beam

$$\Pi[v] = \int_0^L \left(\tfrac{1}{2}\, v'' E I v'' - q v\right) dx$$

$m = 2$

Element Patches

Nonconforming elements and the patch test
Conforming = compatible
Nonconforming = incompatible
Ideal: Conforming elements
Observation: Certain nonconforming elements also give good results, at the expense of nonmonotonic convergence

Nonconforming elements:

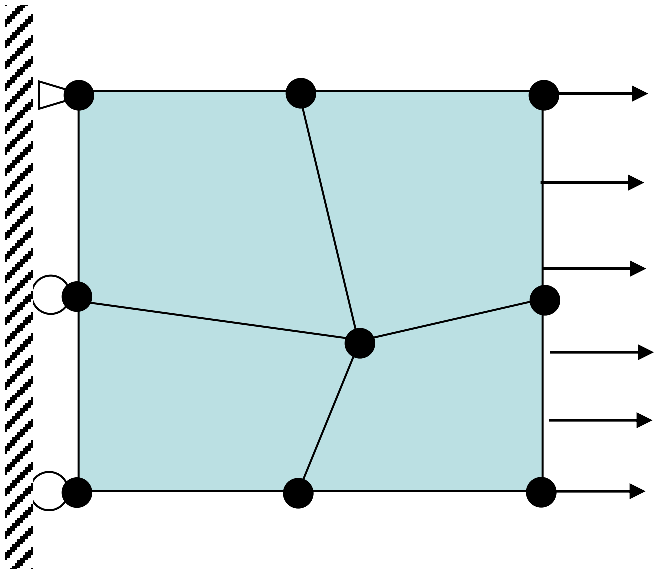
- *satisfy completeness*
- *do not satisfy compatibility*
- *result in at least nonmonotonic convergence if the element assemblage as a whole is complete, i.e., they satisfy the PATCH TEST*

PATCH TEST**:**

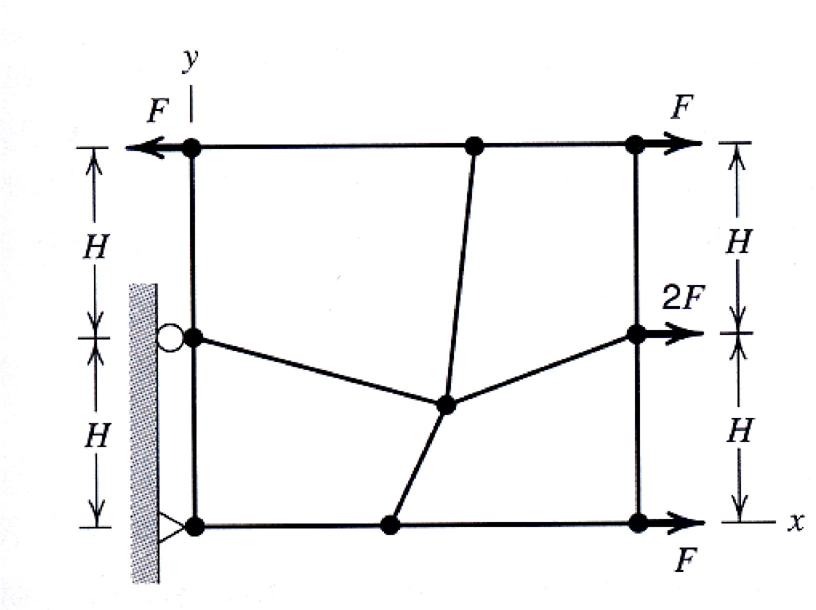
1. A patch of elements is subjected to the minimum displacement boundary conditions to eliminate all rigid body motions
2. Apply to boundary nodal points forces or displacements which should result in a state of constant stress within the assemblage
3. Nodes not on the boundary are neither loaded nor restrained.
4. Compute the displacements of nodes which do not have a prescribed value
5. Compute the stresses and strains

The patch test is **passed** if the computed stresses and strains match the expected values to the ***limit of computer precision***.

Element Patches



Patch Test - Procedure



(a)

Build a simple FE model

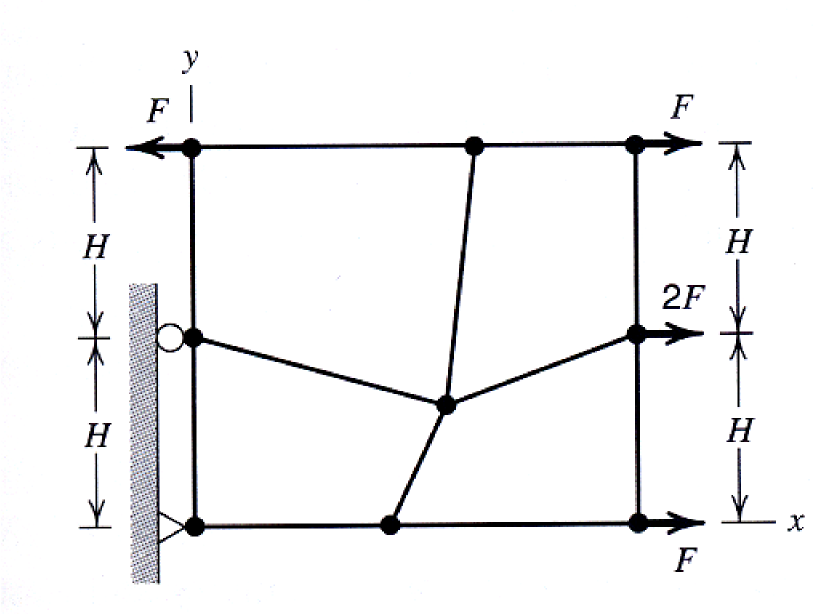
Consists of a Patch of Elements

At least one internal node

Load by nodal equivalent forces consistent with state of constant stress

Internal Node is unloaded and unsupported

Patch Test - Procedure



(a)

$$F = \frac{1}{2}(\sigma_x H t)$$

Compute results of FE patch

If

(computed σ_x) = (assumed σ_x)

test passed

NOTES:

1. This is a great way to debug a computer code
2. Conforming elements ALWAYS pass the patch test
3. Nodes not on the boundary are neither loaded nor restrained.
4. Since a patch may also consist of a single element, this test may be used to check the completeness of a single element
5. The number of constant stress states in a patch test depends on the actual number of constant stress states in the mathematical model (3 for plane stress analysis. 6 for a full 3D analysis)

Completeness & Compatibility in Terms of *m*

Completeness

The ***element shape functions*** must represent exactly all polynomial terms of order *m* in the Cartesian coordinates. A set of shape functions that satisfies this condition is call *m*-complete

Compatibility

The ***patch trial functions*** must be $C^{(m-1)}$ continuous between elements, and $C^{(m)}$ piecewise differentiable inside each Element.

Convergence Requirements

Plane Stress: *m = 1* in Two Dimensions

Completeness

The ***element shape functions*** must represent exactly all polynomial terms of order <=1 in the Cartesian coordinates. That means any *linear polynomial* in *x, y* with a *constant* as special case

Compatibility

The ***patch trial functions*** must be C^0 continuous between elements, and C^1 piecewise differentiable inside each element

Stability

Rank Sufficiency

The discrete model must possess the same solution uniqueness attributes of the mathematical model For displacement finite elements:
the rigid body modes (RBMs) must be preserved no zero-energy modes other than RBMs can be tested by the rank of the stiffness matrix

Positive Jacobian Determinant

The determinant of the Jacobian matrix that relates cartesian and natural coordinates must be everywhere *positive* within the element

Rank Sufficiency

The element stiffness matrix must not possess any zero-energy kinematic modes other than rigid body modes
This can be checked by verifying that the element stiffness matrix has the correct rank:

$$\boxed{\textit{correct rank} = \#\ \textit{of element DOF} - \#\ \textit{of RBMs}}$$

A stiffness matrix that has proper rank is called rank sufficient

Notation for Rank Analysis of Element Stiffness

n_F	number of element DOF
n_R	number of independent rigid body modes
n_G	number of Gauss points in integration rule for **K**
n_E	order of **E** (stress-strain) matrix
r_C	correct (proper) rank $n_F - n_R$
r	actual rank of stiffness matrix
d	rank deficiency $r_C - r$

General case

rank of **K**: $r = \min(n_F - n_R,\ n_E n_G)$

rank deficiency: $d = (n_F - n_R) - r$

Plane Stress, *n* nodes

$$n_F = 2n \qquad n_R = 3 \qquad n_E = 3$$

$$r = \min(2n - 3,\ 3n_G)$$

The element stiffness matrix must not possess any zero-energy kinematic mode other than rigid body modes.

This can be mathematically expressed as follows.

Let n_F be the number of element degrees of freedom, and n_R be the number of independent rigid body modes.

Let r denote the rank of **K**(e)**.** The element is called rank sufficient if $r = n_F - n_R$ and rank deficient if $r < n_F - n_R$.

In the latter case, $d = (n_F - n_R) - r$ is called the rank deficiency.

Rank Sufficiency for Numerically Integrated Finite Elements

If an isoparametric element is numerically integrated, let n_G be the number of Gauss points, while n_E denotes the order of the stress-strain matrix **E.**

Two additional assumptions are made:

(i) The element shape functions satisfy completeness in the sense that the rigid body modes are exactly captured by them.

(ii) Matrix **E** is of full rank.

Then each Gauss point adds n_E to the rank of **K**(e)**,** up to a maximum of $n_F - n_R$. Hence the rank of **K**(e) will be $r = min(\ n_F - n_R\ ,\ n_E\, n_G)$

Rank Sufficiency for Numerically Integrated Finite Elements

To attain rank sufficiency, $n_E n_G$ *must equal or exceed* $n_F - n_R$:

$$n_E n_G >= n_F - n_R$$

from which the appropriate Gauss integration rule can be selected.

In the plane stress problem, $n_E = 3$ *because*

E is a 3 × 3 matrix of elastic moduli;

Also $n_R = 3$. Consequently $r = \min(n_F - 3,\ 3n_G)$ and $3n_G \geq n_F - 3$.

EXAMPLE

Consider a plane stress 6-node quadratic triangle.
Then $n_F = 2 \times 6 = 12$. To attain the proper rank of
$12 - n_R = 12 - 3 = 9$, $n_G \geq 3$.
A 3-point Gauss rule makes the element rank sufficient.

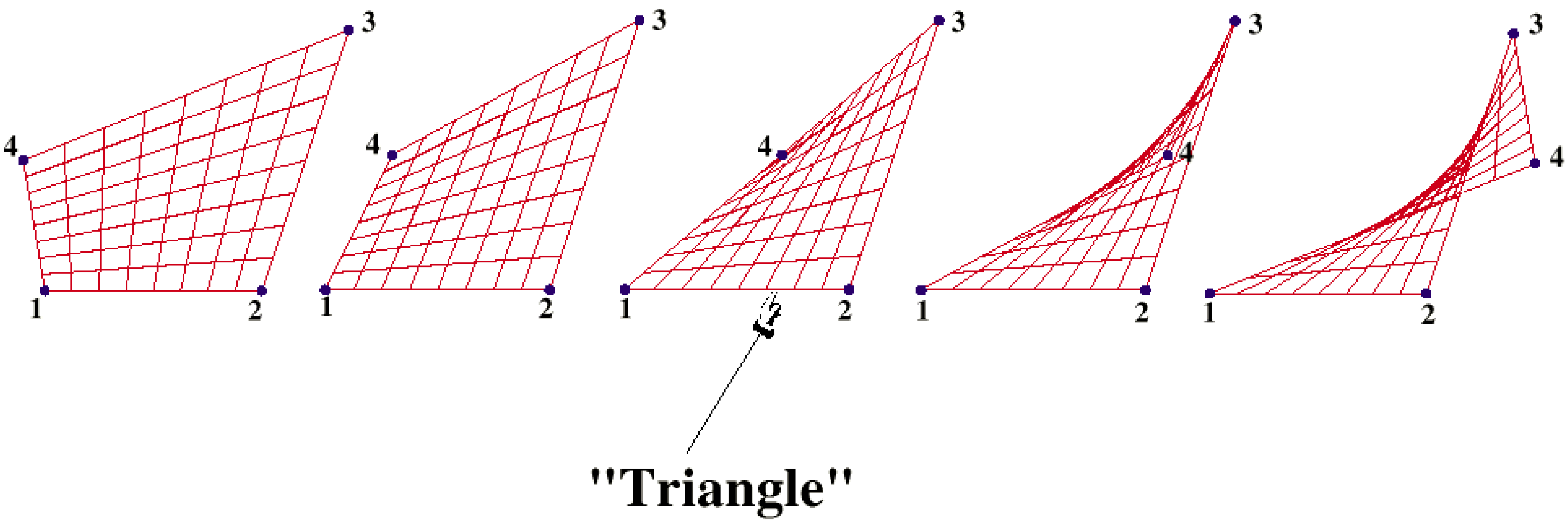
EXAMPLE

Consider a plane stress 9-node biquadratic quadrilateral. Then
$n_F = 2\times9 = 18$. To attain the proper rank of
$18 - n_R = 18 - 3 = 15$, $n_G \geq 5$. *The 2×2 product Gauss* rule is insufficient because $n_G = 4$. Hence a *3×3* rule, which yields $n_G = 9$, is required to attain rank sufficiency.

Rank Sufficiency for Numerically Integrated Finite Elements

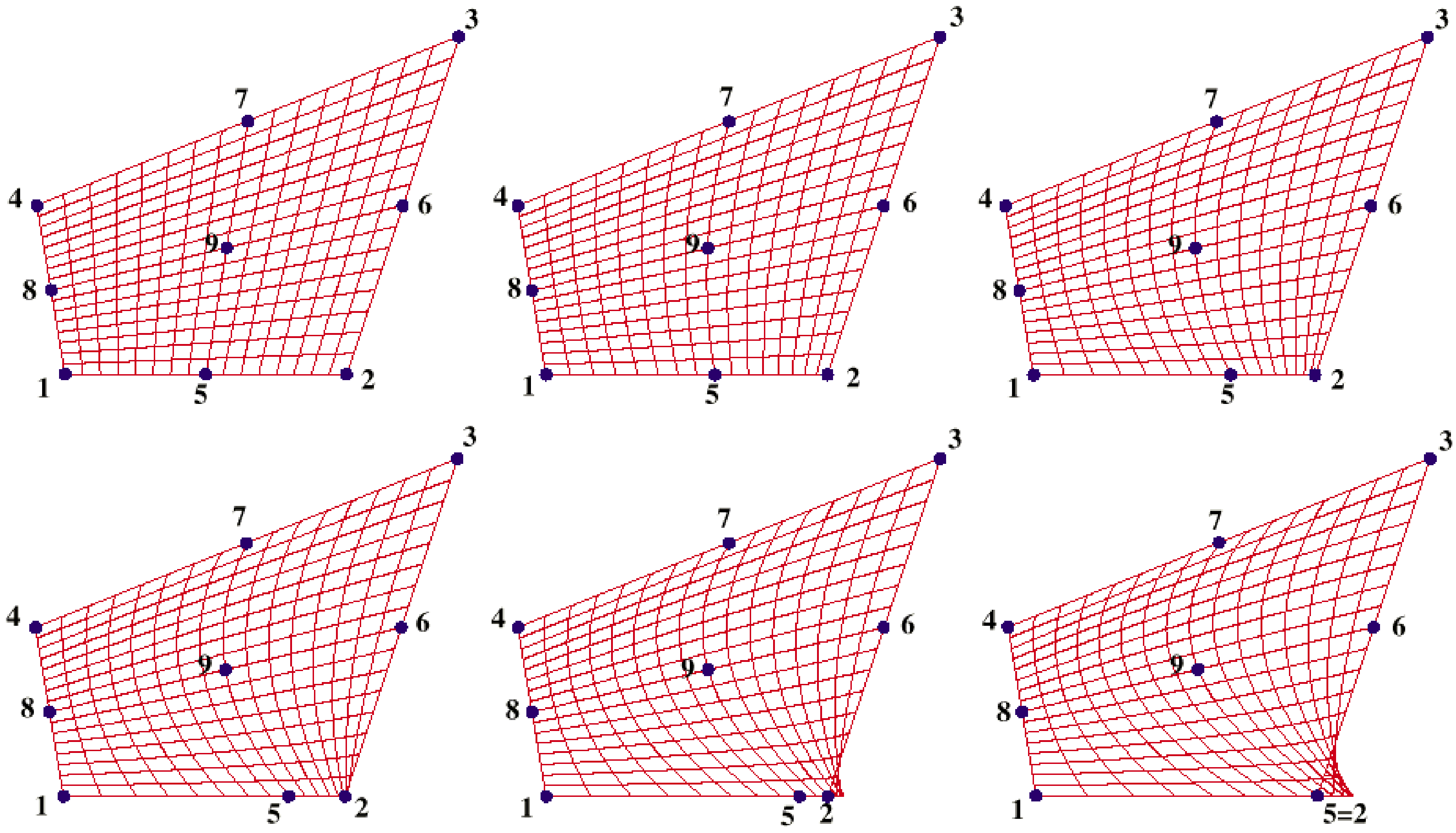
Element	n	n_F	$n_F - 3$	Min n_G	Recommended rule
3-node triangle	3	6	3	1	centroid*
6-node triangle	6	12	9	3	3-midpoint rule*
10-node triangle	10	20	17	6	7-point rule*
4-node quadrilateral	4	8	5	2	2 x 2
8-node quadrilateral	8	16	13	5	3 x 3
9-node quadrilateral	9	18	15	5	3 x 3
16-node quadrilateral	16	32	29	10	4 x 4

Displacing a Corner Node of 4-Node Quad

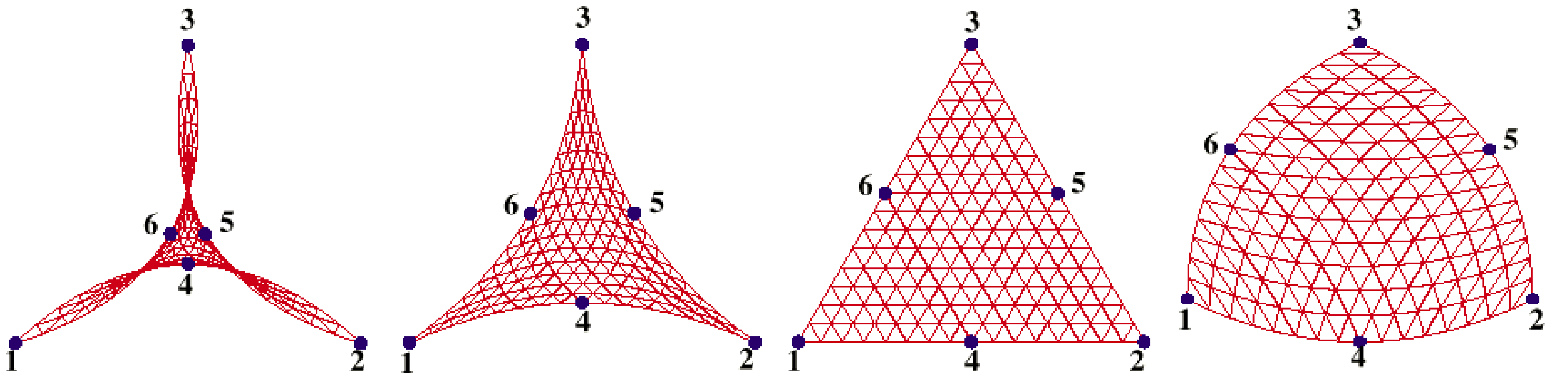


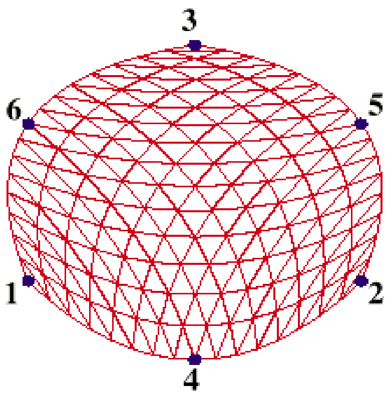
Positive Jacobian Requirement

Displacing a Midside Node of 9-Node Quad

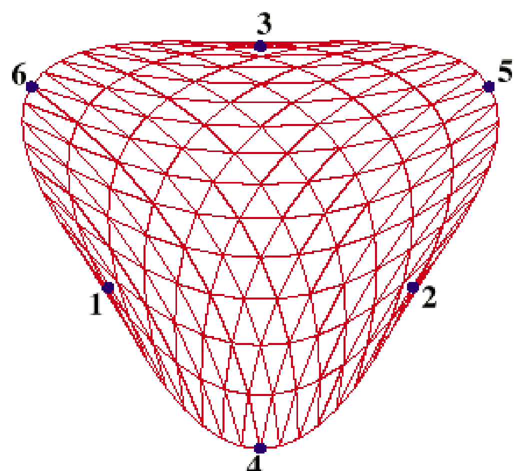


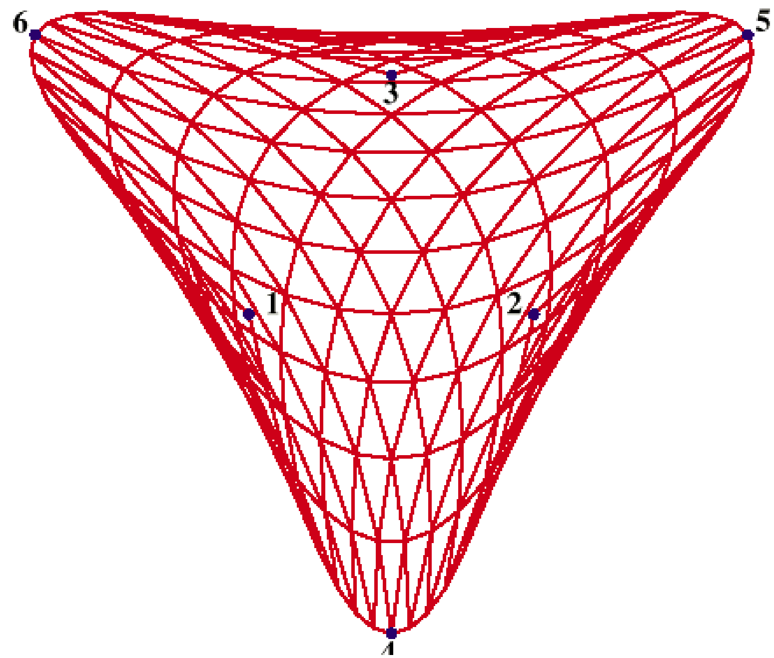
Positive Jacobian Requirement





"Circle"
(looks a bit "squashed"
because of plot scaling)





Convergence of FE Solution

As the mesh in an FE model is "refined" repeatedly, the FE solution will converge to the exact solution of the mathematical model of the problem (the model based on bar, beam, plane stress/strain, plate, shell, or 3-D elasticity theories or assumptions).

Types of Refinement:

h-refinement: reduce the size of the element ("*h*" refers to the typical size of the elements);

p-refinement: Increase the order of the polynomials on an element (linear to quadratic, etc.; "*h*" refers to the highest order in a polynomial);

r-refinement: re-arrange the nodes in the mesh;

hp-refinement: Combination of the h- and p-refinements (better results!).

Adaptivity (h-, p-, and hp-Methods)

- Future of FE applications
- Automatic refinement of FE meshes until converged results are obtained
- User's responsibility reduced: only need to generate a good initial mesh

Error Indicators:

Define,

σ : element by element stress field (discontinuous),

σ^* : averaged or smooth stress (continuous),

$\sigma_E = \sigma - \sigma^*$: the error stress field.

Adaptivity (h-, p-, and hp-Methods)

Compute strain energy,

$$U = \sum_{i=1}^{M} U_i, \qquad U_i = \int_{V_i} \frac{1}{2} \mathbf{s}^T \mathbf{E}^{-1} \mathbf{s}\, dV;$$

$$U^* = \sum_{i=1}^{M} U_i^*, \qquad U_i^* = \int_{V_i} \frac{1}{2} \mathbf{s}^{*T} \mathbf{E}^{-1} \mathbf{s}^*\, dV;$$

$$U_E = \sum_{i=1}^{M} U_{Ei}, \qquad U_{Ei} = \int_{V_i} \frac{1}{2} \mathbf{s}_E^T \mathbf{E}^{-1} \mathbf{s}_E\, dV;$$

where M is the total number of elements, V_i is the volume of the element i.

One error indicator --- the relative energy error:

$$\eta = \left[\frac{U_E}{U + U_E} \right]^{1/2}. \qquad (0 \le \eta \le 1)$$

Adaptivity (h-, p-, and hp-Methods)

The indicator η is computed after each FE solution. Refinement of the FE model continues until, say

η <= 0.05. => converged FE solution.