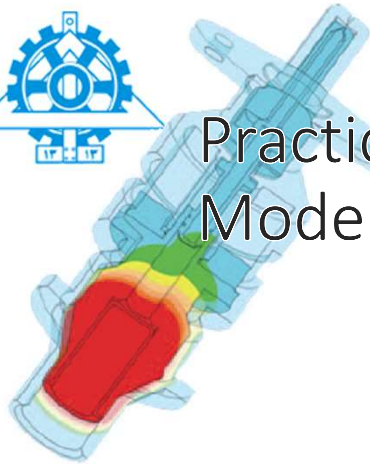
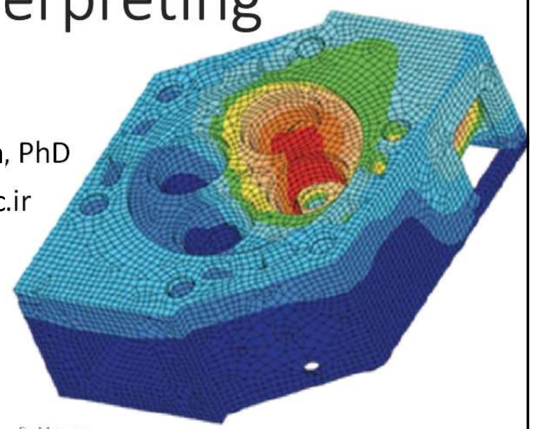


Practical Considerations in Modeling and Interpreting Results

By: Maryam Mahnama, PhD
m.mahnama@ut.ac.ir



Contents

In this chapter, we will describe some modeling guidelines, including:

- ✓ Generally recommended mesh size
- ✓ Natural subdivisions modeling around concentrated loads
- ✓ Use of symmetry and associated boundary conditions
- ✓discussion of equilibrium, compatibility, and convergence of solution.

The presentation is **"recipe oriented"** and illustrated by specific examples from structural mechanics.
No attempt is made at rigorous justification of rules and recommendations, because that would require mathematical tools <u>beyond the scope of this course.</u>

General Notes on Modelling

- Finite element modeling is partly an art guided by visualizing physical interactions taking place within the body.
- One appears to acquire good modeling techniques through:
  - Experience
  - Working with experienced people
  - Searching in the literatures

General Notes

- In modeling an experienced user should
- Understand the physical behavior taking place
- Choosing the proper type of element
- Understanding the boundary conditions imposed on the problem
- Determine the kinds of loads that must be applied to a body and their magnitudes and locations.

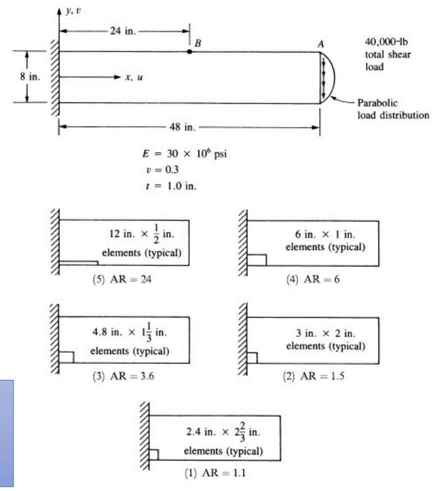
Aspect Ratio and Element Shape

Definition: the ratio of the longest dimension to the shortest dimension of a quadrilateral element.

Five different finite element models used to analyze a beam subjected to bending :

In general, as the aspect ratio increases, the inaccuracy of the solution increases.

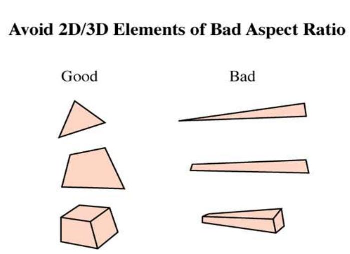
Exception: if the stress gradient is close to zero at some location of the actual problem, then large aspect ratios at that location still produce reasonable results.

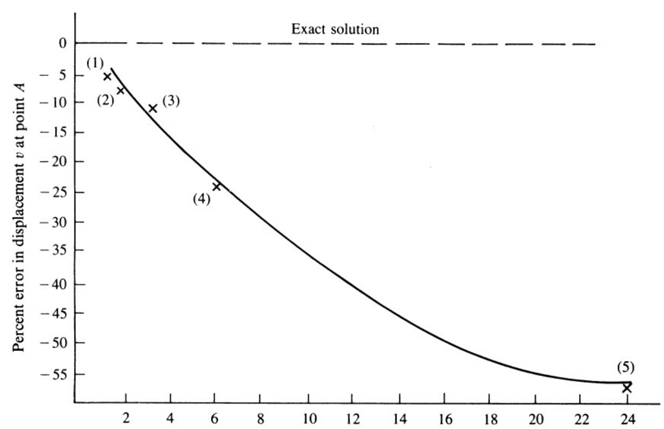


Aspect Ratio and Element Shape

In general, an element yields best results if its shape is compact and regular:

(1) Low aspect ratio,
(2) Near equal corner angles

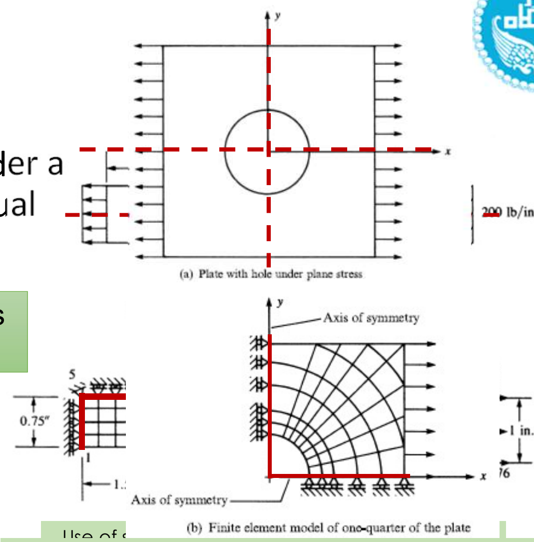




Use of Symmetry

- Use of symmetry allows us to consider a reduced problem instead of the actual problem

⇨ we can use a finer subdivision of elements with less labor and computer costs

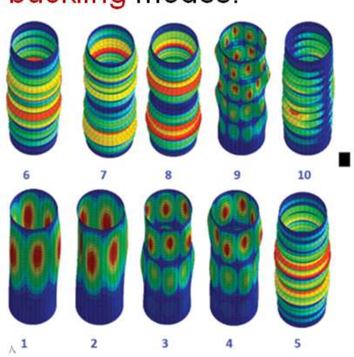


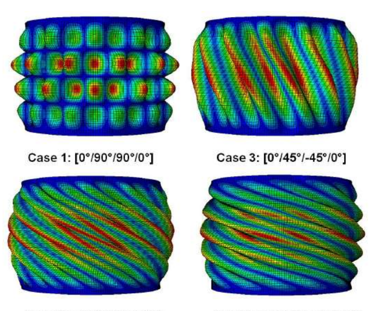
Problem reduction using axes of symmetry applied to a plate with a hole subjected to tensile force

Use of Symmetry

- Important Note:

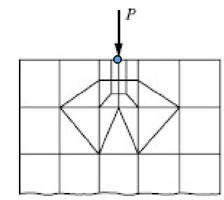
In vibration and buckling problems, symmetry must be used with caution since symmetry in geometry does not imply symmetry in all vibration or buckling modes.



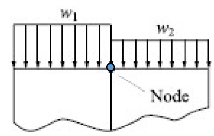


Natural Subdivisions at Discontinuities

Nodes: locations of concentrated loads or discontinuity in loads

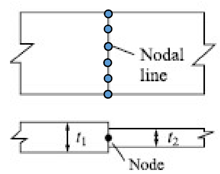


Concentrated load

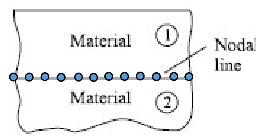


Abrupt change of distributed load

Nodal line: abrupt thickness change or changes in material properties.



Abrupt change of plate thickness

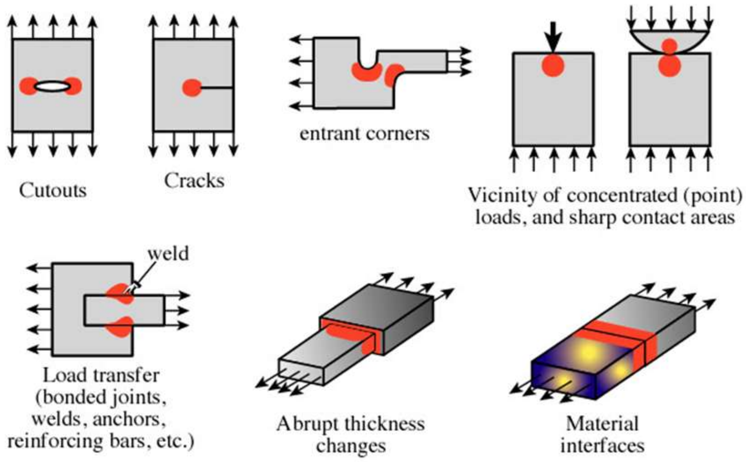


Abrupt change of material properties

Sizing of Elements and the *h* and *p* Methods of Refinement

- There are two distinct methods in FEM to refine a finite element mesh to improve the results: *h* method and *p* method.
- These methods are then used to revise or refine a finite element mesh ⇒ improvement of the results in the next refined analysis.
- The goal of the analyst is to refine the mesh to obtain the necessary accuracy by using only as many degrees of freedom as necessary.
- The final objective of this so called adaptive refinement is to obtain equal distribution of an error indicator over all elements.
- discretization depends on the geometry of the structure, the loading pattern and the boundary conditions.

Sizing of Elements and the *h* and *p* Methods of Refinement



regions of stress concentration or high stress gradient due to fillets, holes, or re-entrant corners require a finer mesh near those regions

Sizing of Elements and the h and p Methods of Refinement

h method of refinement:

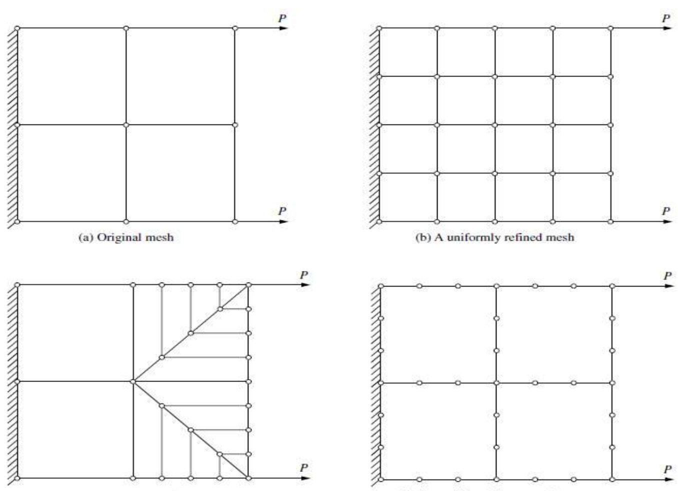
- ✓ In the *h* method of refinement, we use the particular element based on the shape functions for that element
- ✓ For example, linear functions for the bar, quadratic for the beam, bilinear for the CST and …
- ✓ We then add elements of the same kind to refine or make smaller elements in the model
- ✓ The mesh refinement is continued until the results from one mesh compare closely to those of the previously refined mesh.

Sizing of Elements and the h and p Methods of Refinement

p method of refinement:

- ✓ In the *p* method of refinement, the polynomial *p* is increased from perhaps quadratic to a higher-order polynomial based on the degree of accuracy specified by the user
- ✓ The goal is to better fit the conditions of the problem, such as the boundary conditions, the loading, and the geometry changes
- ✓ The order of the polynomial is normally increased while the element geometry remains the same and the problem is solved again
- ✓ The *p* refinement may consist of adding DOF to existing nodes, adding nodes on existing boundaries between elements, and/or adding internal DOF.
- ✓ The results of the iterations are compared to some set of convergence criteria specified by the user

Examples of *h* and *p* Refinement Methods



(a) Original mesh

(b) A uniformly refined mesh

(c) A possible nonuniform *h* refinement

(d) A possible uniform *p* refinement

Equilibrium and Compatibility of Finite Element Results

An approximate solution for a stress analysis problem using the finite element method based on **assumed** displacement fields *does not generally satisfy* all the requirements for **equilibrium** and **compatibility** that an **exact** theory-of-elasticity solution satisfies.

Remember that relatively **few exact solutions** exist!

Hence, **FEM** is a very practical method for obtaining **reasonable**, but **approximate**, numerical solutions.

There are some of the approximations generally *inherent* in finite element solutions.

Equilibrium and Compatibility of Finite Element Results

1. Equilibrium of nodal forces and moments is satisfied

This is true because the global equation $F = Kd$ is a *nodal* equilibrium equation whose solution for d is such that the sums of all forces and moments applied to each node are zero.

Equilibrium of the **whole structure** is also satisfied because the **structure reactions** are included in the nodal equilibrium equations.

Equilibrium and Compatibility of Finite Element Results

2. Equilibrium within an element is not always satisfied

Does the displacement function introduced for the element, satisfies the Differential equation corresponding to the element?

The answer is yes for Bar, Beam and Constant-Strain Triangular (CST) elements.

These elements have equilibrium within element

The answer is no for Linear-Strain Triangular (LST), Axisymmetric and Rectangular elements.

Equilibrium and Compatibility of Finite Element Results

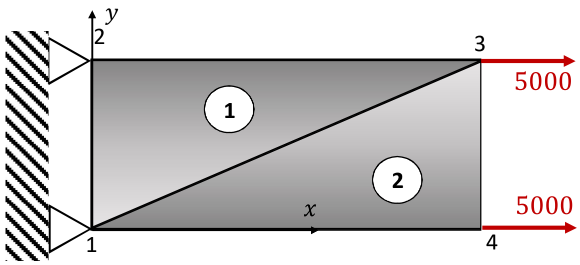
3. Equilibrium is not usually satisfied between elements

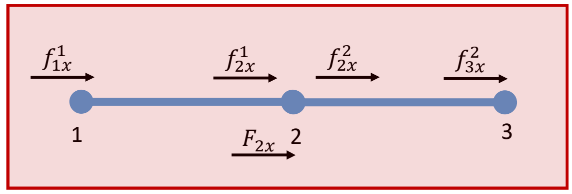
For line elements, such as used for truss and frame analysis, interelement equilibrium is satisfied.

However, for two- and three dimensional elements, interelement equilibrium is not usually satisfied.

A differential element including parts of two adjacent finite elements is usually not in equilibrium.

Equilibrium and Compatibility of Finite Element Results





Element#1

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} \boxed{1005} \\ \boxed{301} \\ \boxed{2.4} \end{Bmatrix} \text{psi}$$

Element#2

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{Bmatrix} \boxed{995} \\ \boxed{-1.2} \\ \boxed{-2.4} \end{Bmatrix} \text{psi}$$

Equilibrium and Compatibility of Finite Element Results

4. Compatibility within an element

Compatibility is satisfied within an element as long as the element displacement field is continuous. Hence, individual elements do not tear apart.

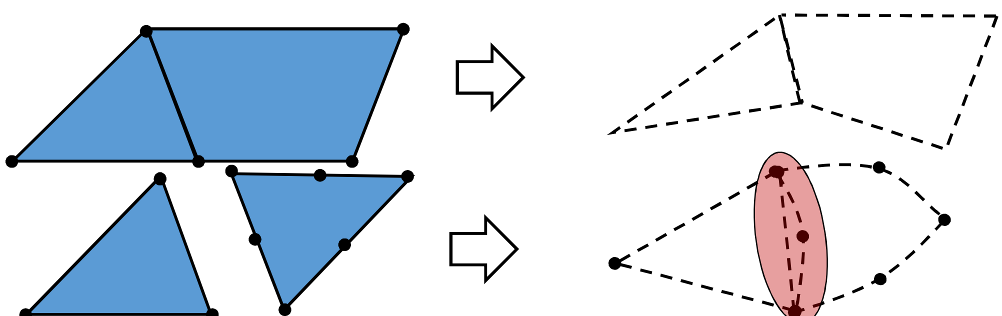
5. Compatibility between elements at nodes

In the formulation of the element equations, compatibility is invoked at the nodes. ⇒ Elements remain connected at their common nodes. Similarly, the structure remains connected to its support nodes because boundary conditions are invoked at these nodes.

Equilibrium and Compatibility of Finite Element Results

6. Compatibility between elements in edges

Compatibility may or may not be satisfied along interelement boundaries.



Opening or overclosure may happen

Complete displacement function

The approximation function should allow for rigid-body displacement (constant motion of the body without straining) and for a state of constant strain within the element.

This idea of completeness also means in general that the lower order term cannot be omitted in favor of the higher-order term.

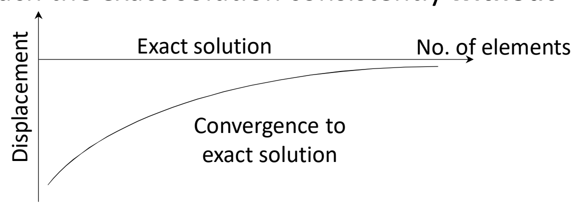
For the simple linear function $u = a + bx$, this means a cannot be omitted while keeping bx .

Convergence of Solution

Monotonic convergence:

Completeness of a function is a **necessary** condition for *monotonic convergence* to the exact answer.

This type of convergence is the process in which successive approximation solutions (FE solutions) approach the exact solution consistently **without changing <u>sign</u>** or **<u>direction</u>.**



Convergence of a finite element solution based on the *compatible* and *complete* displacement formulation