IE360: CAD/CAM

Computer Aided Design and Computer Aided Manufacturing

Lecture (8)
Finite Element Analysis (FEA)

Dr Ashraf Afifi

Industrial Engineering Department

King Saud University

Outline

- Introduction
  - Why FEA?
  - FEA: Basic concept
  - FEA vs. analytical methods
- FEA Method
- Automatic Mesh Generation
- Cautions

Introduction:

- Finite Element Analysis (FEA) is a numerical method of solving engineering problems.

- FEA may be applied in (discussion will be limited to structural problems):
  - Structural,
  - Heat transfer,
  - Fluid flow.

- **Why FEA?**
  - Used in problems where analytical solution not easily obtained.
  - Mathematical expressions required for solution not simple because of complex:
    - geometries
    - loadings
    - material properties

FEA: Basic concept

- Replace continuous geometry with a set of objects with a finite number of degree of freedoms (DOF).

- Divide body into finite number of simpler units (elements).

- Elements connected at *nodal* points.
    - points common to two or more adjacent elements.
    - set of elements referred to as "*mesh*".

FEA vs. analytical methods

- Analytical methods involve solving for entire system in one operation.

- FEA involves defining equations for each element and combining to obtain system solution.
    - is therefore an approximation technique.

- An example of FEA Mesh is shown in the following figure

FEA Method:

- The most common technique is *Displacement Method*
    - Loads are known
    - Resistance to deformation of elements known
    - Displacements are unknown values

- Solve for displacements
    - Stress is a secondary solution, derived from displacements.

Structural FEA

- Structural problems defined in terms of:
  - loads (forces)
  - resistance to deformation (stiffness)
  - displacements

- Given by: $[k]\{d\} = \{F\}$
  - k = stiffness, d = displacement, F = force

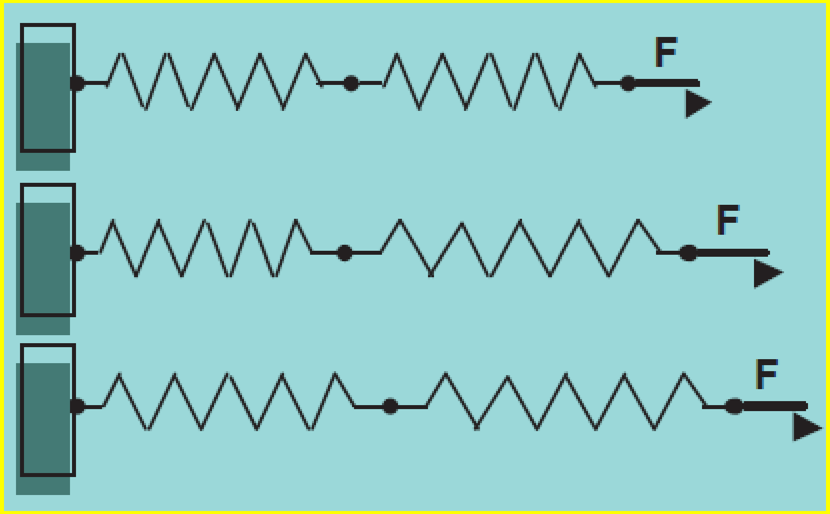
FEA concept

- Assume that variation of displacement across element is a simple function.

- Results in a set of relationships for displacement at nodal points for each element.

- Combine for entire mesh.
  - problem is converted to large number of simple algebraic equations.

- Spring element displacement



- Computer use in FEA
  - Because of
    - the relatively simple nature of equations,
    - connectivity between elements (resultants from adjacent elements applied),
    - combining of solutions for individual elements,
    - large number of equations to solve.
  - FEA well suited to computational automation.

- FEA steps:
  - Three steps:
    - pre-processing
    - analysis
    - post-processing

- *Pre-processing*
  - Requires definition of:
    - system geometry
    - restraints on the system (boundary conditions)
    - loads applied
    - type and properties of elements
    - material properties

- Older systems
  - Prior to use of GUIs (graphic user interfaces)
    - analyst would define input by hand
    - each nodal location, element type, constraint, etc. will be input one at a time to a data file.

- Use of solid modeling in FEA
  - Solid modeling use simplifies and enhances FEA.
  - Model database used as pre-processor input.
  - May permit definition of all preprocessing data.
- Modern solid modeling systems
  - Allow direct definition of element type.
  - Have automatic mesh generators.
  - Permit definition of loads and boundary conditions graphically, directly upon model geometry.
  - Allow specification of other data through menu input.

Automated mesh generation:

- Mesh shape considerations
  - Shape of mesh critical to analysis.
  - Higher density improves solution at cost of computational time.
  - Simple geometry requires fewer elements, more complexity requires increased density.
  - Mesh shape related to loads, and boundary conditions.

Defined geometry within CAD package

First pass mesh

- Automeshing exists for 2D and 3D systems.

- Intelligent meshing systems consider geometry and topology of model.

- Some systems support bi-directional associativity.

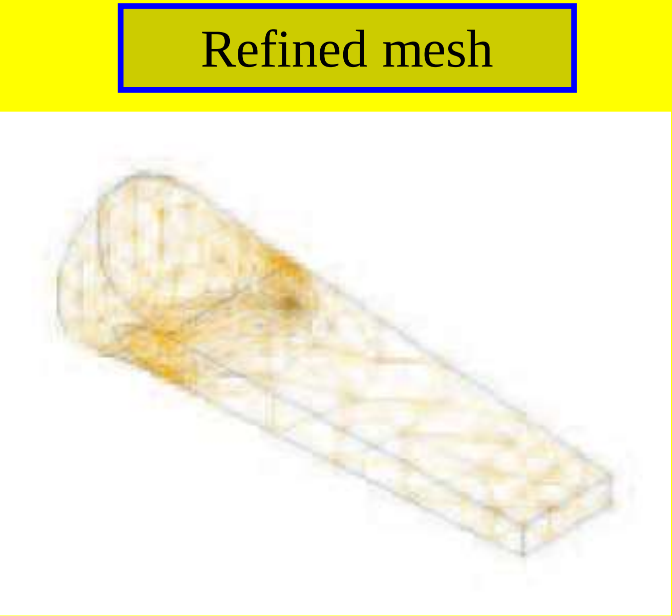
- Changes in model geometry will produce changes in mesh.

- However, changes in topology (additional edges) would require remeshing.

- Different mesh cases may be defined for the same model.

- Early analysis may involve coarse mesh.
  - Low mesh density.
  - Faster computation time.

- Mesh is refined for further analysis.



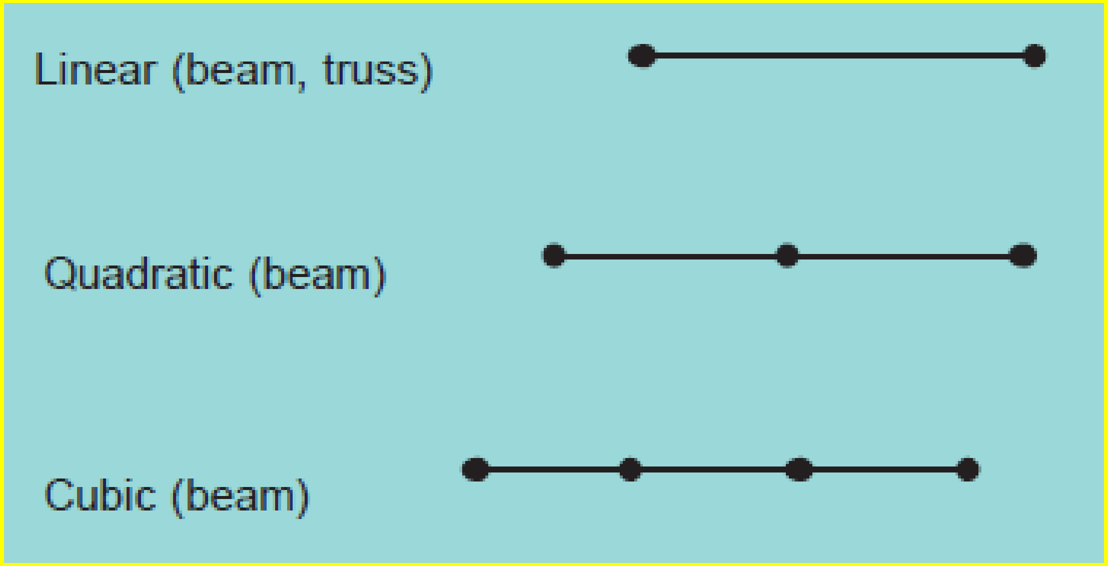
- Model case
  - In addition to mesh cases
    - load cases,
    - constraint cases, may be defined.

  - As with meshes, early analysis may involve simplified loading and constraints, later refined.

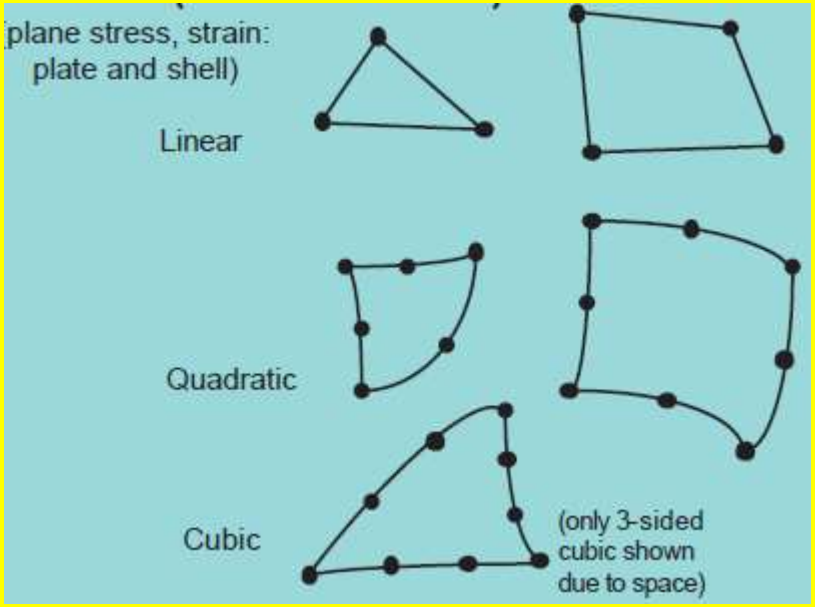
- Boundary conditions and loads
  - Much of specification is automated.
  - Specification not limited to nodal.
  - Possible to specify restrains and loads for high level geometric entities (edges, faces).
  - System applies appropriate nodal properties
- Pre-processing within CAD package

- Automated mesh generation concerns:
  - Mesh shape should consider loads and restraints.
  - Automated systems may not do so.
  - This can be of high concern when defining meshes for different load cases.

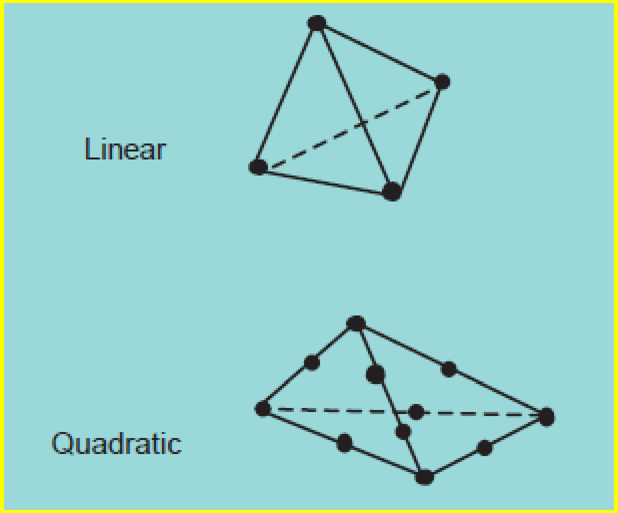
- Element Types:
  - Dimensional elements



- 2 D (area elements)



- 3 D (volume elements)



- Load cases (mechanical): Rules of thumb
  - Moments
    - apply at single node, apply at nodes on an edge
  - Point forces:
    - apply to single node, nodes along edges, nodes on surface.
  - Surface pressure:
    - may be uniform or non-uniform.
    - applied to edge or surface.
    - may be nonconservative (load normal to surface in large displacement cases).

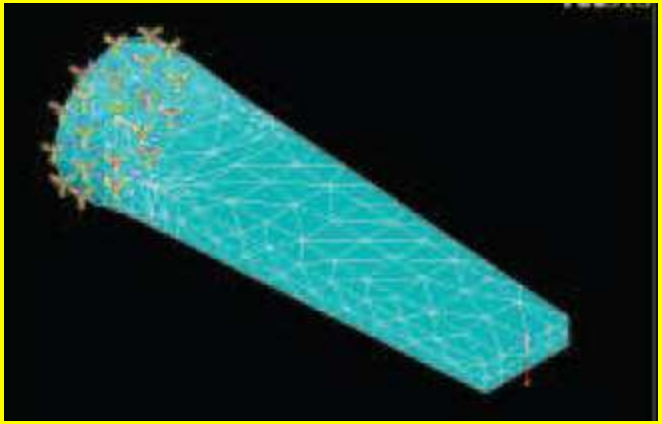
- *Post-processing*
  - Output of FEA data.
  - Desire simplicity for speed in design evaluation.
  - Many systems support enhanced graphics display.

- Output Examples
  - Extreme values reported in list form.
  - Extreme values displayed in color on rendered solid model.
    - typically include color index.
  - Note that max value occur at surface unless internal loads present.

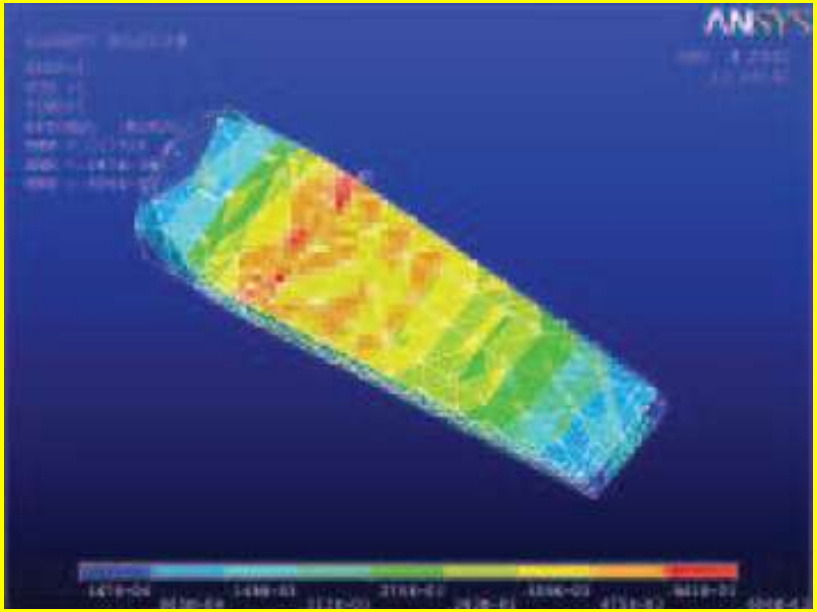
➢ *Design evaluation*

- Von Mises stress
- Tresca stress
- Principle stress

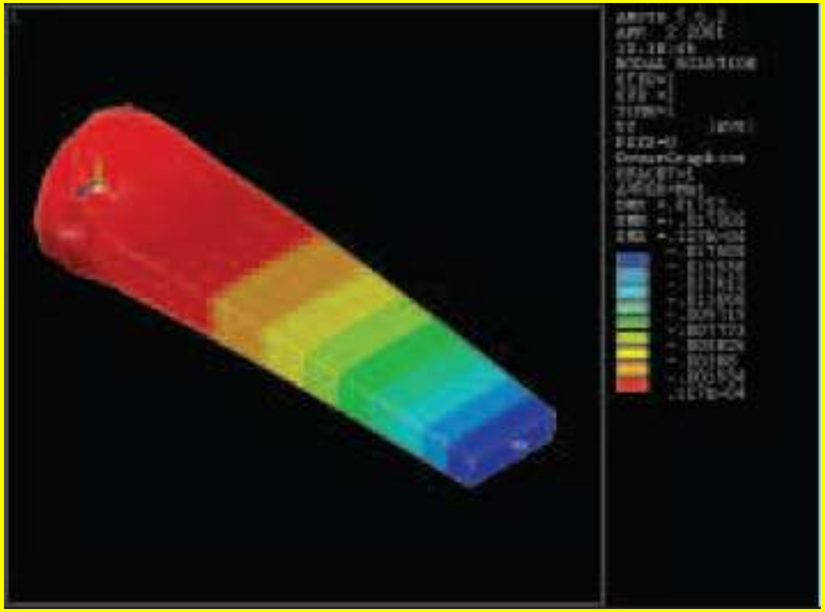
FEA mesh, load and boundary conditions



Strain analysis output



Stress analysis output



Cautions:

- Results only as good as the job done in creating mesh, applying loads and boundary conditions.
  - if not calculated and applied correctly, results of little use.

- Must understand
  - mechanics principles.
  - material and physical properties.

- Mesh generation is often a critical aspect.

- Proper mesh can reduce errors in primary results (such as displacements) by half.

- Point loads produce inaccurate local deformation.

- Some loads such as bearing loads not well defined in current software.
  - can be difficult to define and apply.