Finite Element Analysis

• Numerical method of solving engineering problems.
• May be applied in:
  – structural
  – heat transfer
  – fluid flow
• We will limit discussion to structural problems.

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 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 involving defining equations for each element and combining to obtain system solution.
  – is therefore an approximation technique

Example of FEA Mesh

FEA method

• 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.

Computational methods

- 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 interface)
  - analyst would define input by hand
  - each nodal location, element type, constraint, etc. input one at a time to a data file.

Use of SM in FEA

- Solid modeling use simplifies and enhances FEA.
- Model database used as pre-processor input.
- May permit definition of all pre-processing data

Modern SM systems

- allow direct definition of element type
- have automatic mesh generators
- permit definition of loads and boundary condition graphically, directly upon model geometry.
- allow specification of other data through menu input

Mesh shape considerations

- shape of mesh critical to analysis
- higher density improves solution at cost of computational time
- simple geometry require fewer elements, more complexity requires increased density.
- mesh shape related to loads, BCs.

Defined Geometry Within CAD Package

First Pass Mesh
Automated mesh generation

- Automeshing exists for 2D and 3D systems
- Intelligent meshing systems consider geometry and topology of model

Automated mesh generation

- some systems support bi-directional associativity
- changes in model geometry will produce changes in mesh
- however, changes in topology (additional edges) would require remeshing

Automated mesh generation

- different mesh cases may be defined for same model
- early analysis may involved 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 restraints 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

1 Dimensional elements

- Linear (beam, truss)
- Quadratic (beam)
- Cubic (beam)

2 D (area elements)

- Linear
- Quadratic
- Cubic

Triangular Surface Mesh

3 D (volume elements)

- Linear
- Quadratic
Tetrahedral Solid Mesh

Element size definitions

Max Edge Length
- start at 1/6 maximum overall dimension for coarse mesh
- refine to increase accuracy

Max Face Aspect Ratio
- desire value less than 4
- 1.1547 would require equilateral triangles
  - very difficult condition for solids

Minimum Face Edge Ratio
- value of 1.0 would require equilateral triangles
- start at 0.5, move toward 0.7
- at 0.7:
  - max edge difference would be 30%
  - most edges would only differ by 20%
  - minimum corner angle 41 degrees
Load cases (mechanical)

• Rules of thumb:
  • Moments
    – shell elements:
      • apply at single node, apply at nodes on an edge
    – solids
    – apply as force couples acting at nodes
  • Point Forces
    – apply to single node, nodes along edges, nodes on surface

Rules of thumb (con’t)

• Surface Pressure
  – may be uniform or non-uniform
    • function of active coordinate system
  – applied to edge or surface
  – may be nonconservative (load normal to surface is 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 is 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

• for homogeneous ductile materials
  – maximum Von Mises stress less than material’s yield stress
  – maximum Tresca stress less than material’s shear strength
• for homogeneous brittle materials
  – maximum principle stress less than ultimate tensile strength of material

FEA Mesh, Load and Boundary Conditions
Strain Analysis Output

Stress Analysis Output

FEA Result Car Roof Crush

Physical Crush Test

Comparison of Physical and Numerical Results

Example Analysis of shaft
Two different geometries examined

Cautions
• results only as good as the job done in creating mesh, applying loads and BCs
  – if not calculated and applied correctly, results of little use
• must understand
  – mechanics principles
  – material and physical properties

Cautions
• mesh generation often critical aspect
• proper mesh can reduce errors in primary results (such as displacements) by half

Cautions
• 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