Introduction to FEA Process

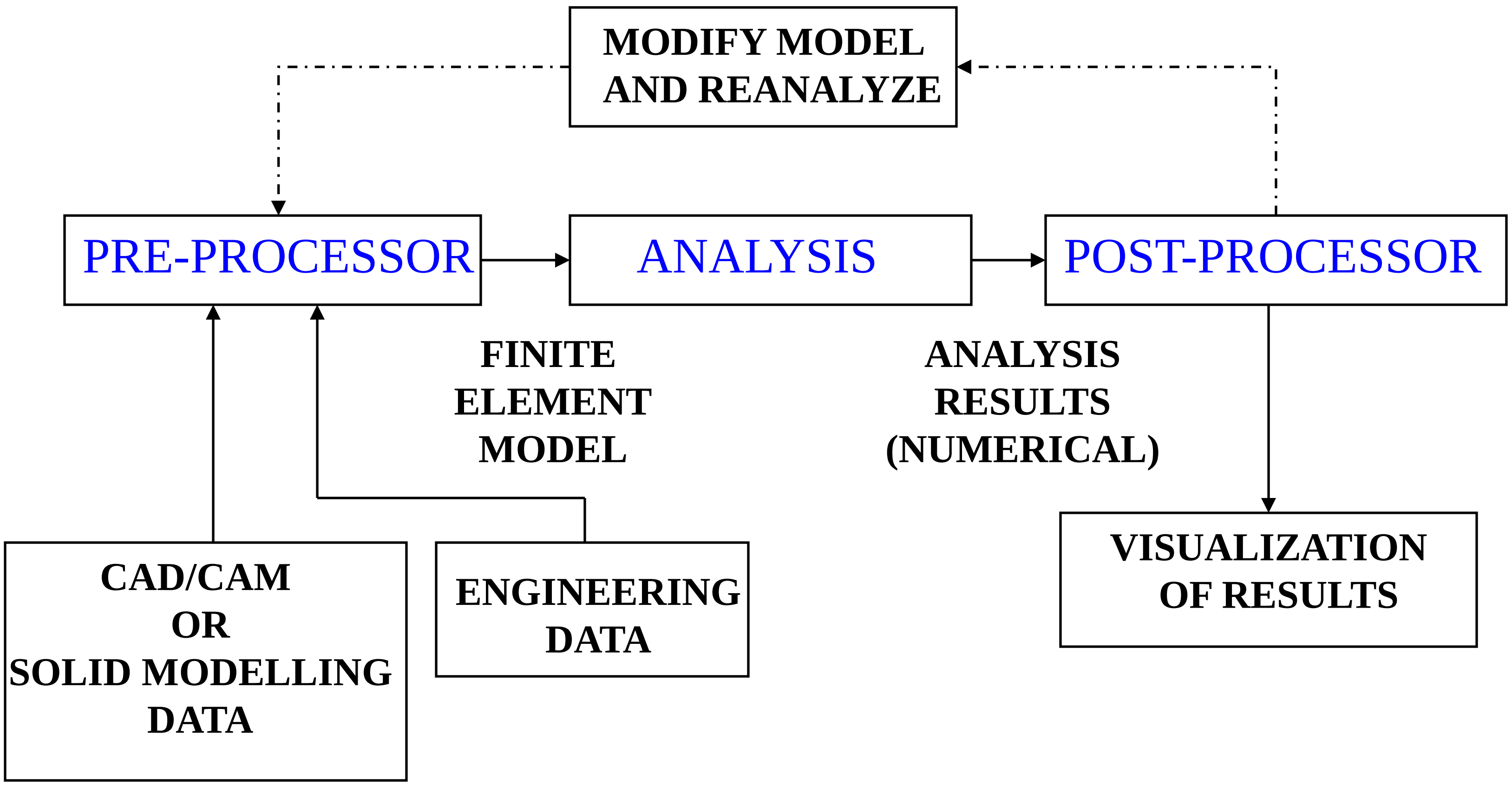
Session Objectives

- At the end of the session the delegates should have understood
  - FEA Process
  - Preprocessing
  - Meshing techniques and guidelines
  - Element Quality Checks
  - Other miscellaneous topics

FEA

- The basic theme of FEA is to make calculation at only limited finite) no of points and then interpolate the results for entire domain(surface or volume)
- Any continuous object has infinite degrees of freedom and its just not possible to solve the problem in this format,
- The finite element method reduces degrees of freedom from infinite to finite with help of discritiization.
- The FEA is based on mathematical equations. It does not under stand/ recognize any other physically separated component/ element unless and until mathematical relation is defined between them.

Finite Element Analysis Process



Finite Element Analysis Process

- PRE-PROCESSING (HYPERMESH, MSC/PATRAN, ANSYS)
  - Geometric approximation/Idealization
  - Finite Element mesh generation
  - Geometric properties of elements
  - Material properties
  - Loads
  - Boundary conditions
- ANALYIS (MSC/NASTRAN, LS-DYNA, ANSYS)
  - Element stiffness and load matrix generation
  - Assembly of elemental matrices
  - Solution of equations for structural displacements
- POST- PROCESSING (MSC/PATRAN, LS-POST, ANSYS)
  - Calculations of stresses, strains, reaction forces etc
  - Graphical visualization of results

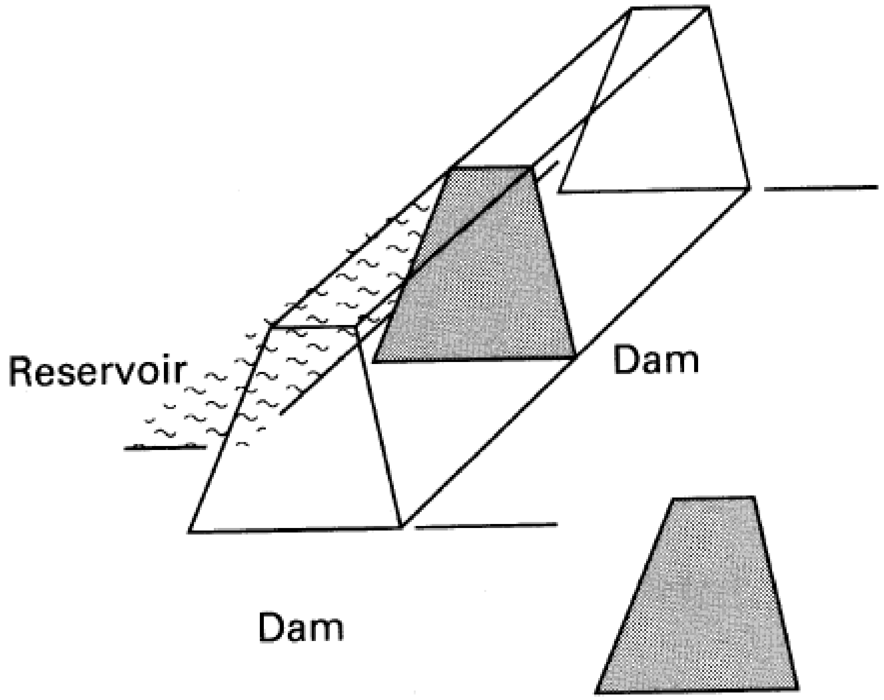
PRE-PROCESSING

- Geometric approximation/Idealization
- Finite Element mesh generation
- Geometric properties of elements
- Material properties
- Loads
- Boundary conditions

Geometric Approximations/Idealization

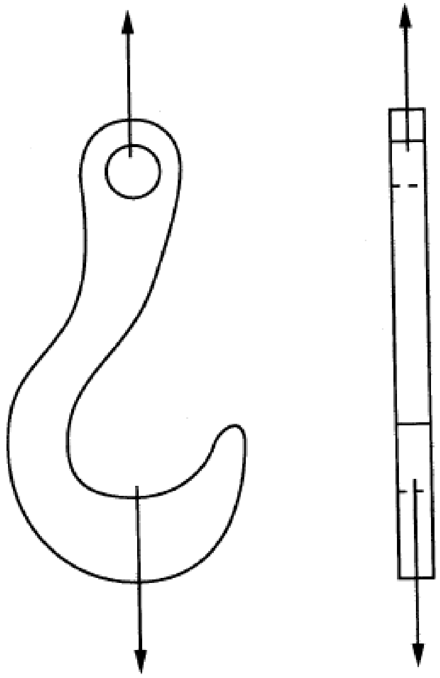
- All structures are 3D in Engineering systems but geometrical approximations are made to facilitate simple stress analysis.
- **2D MODELS**
  - **(a) Plane Strain**
  - When the geometry and loads of a problem can be completely described in one plane.

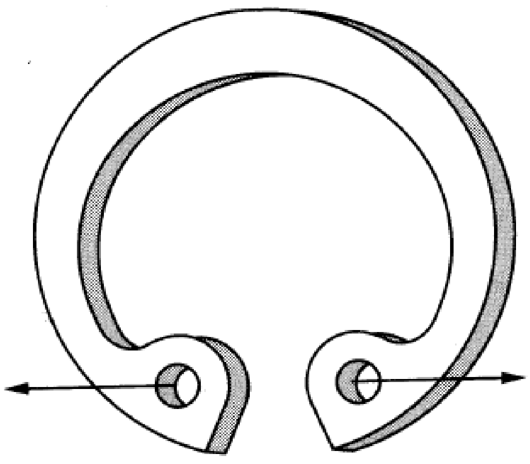
Geometric Approximations/Idealization



Long bodies whose geometry and loading do not vary significantly in the longitudinal direction

Geometric Approximations/Idealization

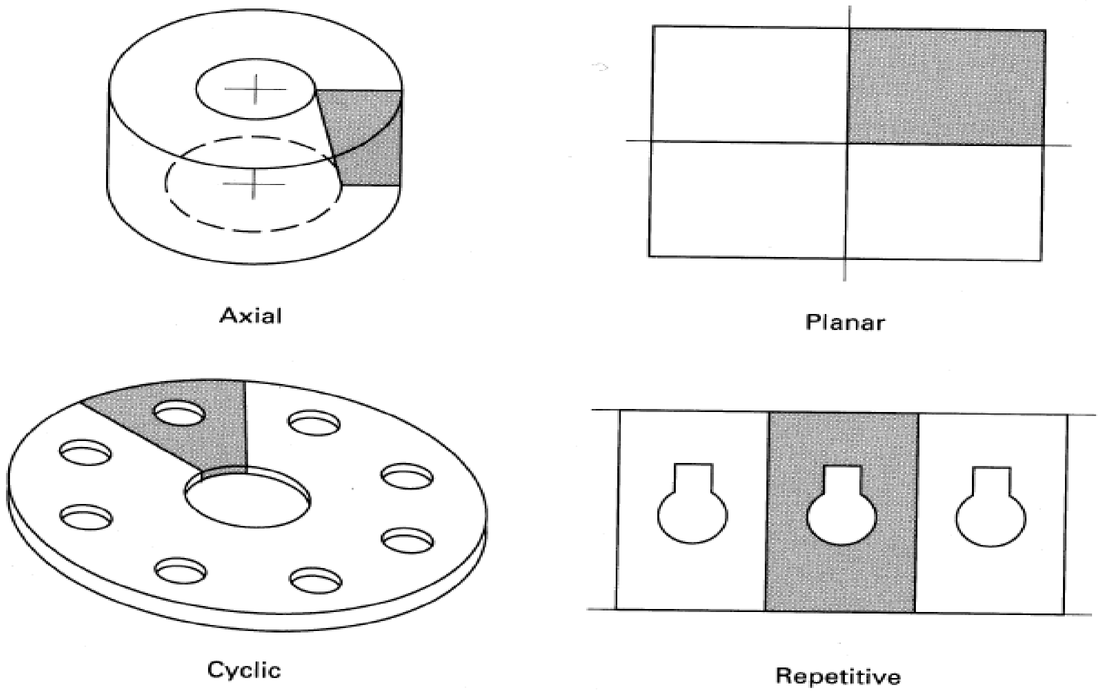




Bodies with negligible dimensions in one direction and loaded in the plane of the body

Simplification through Symmetry

There are 4 common types of symmetry in engineering problems: Axial, Planar, Cyclic and Repetitive

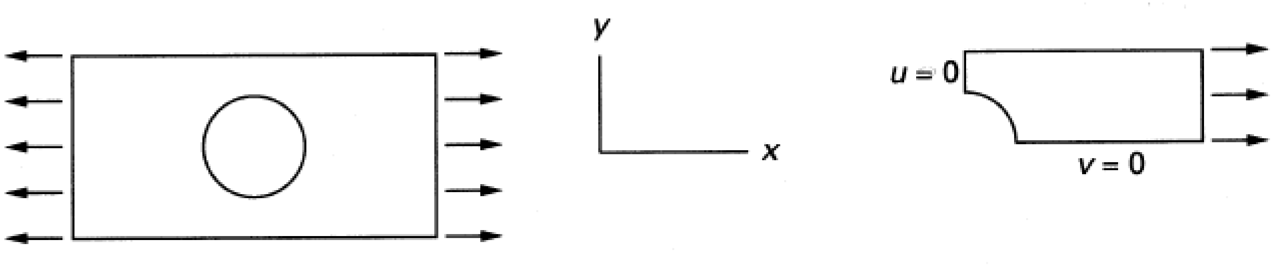


Simplification only possible if the geometry of the structure and the loading conditions have the same symmetry

Simplification through Symmetry

Axial symmetry: Is a feature of many engineering problems so all FE programs contain special ax symmetric elements. These automatically allow for the circumferential variation being constant.

Planar symmetry: The correct constraint conditions on the lines of symmetry must be specified

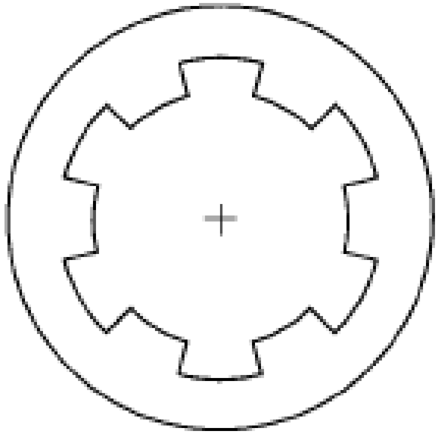


Complete problem

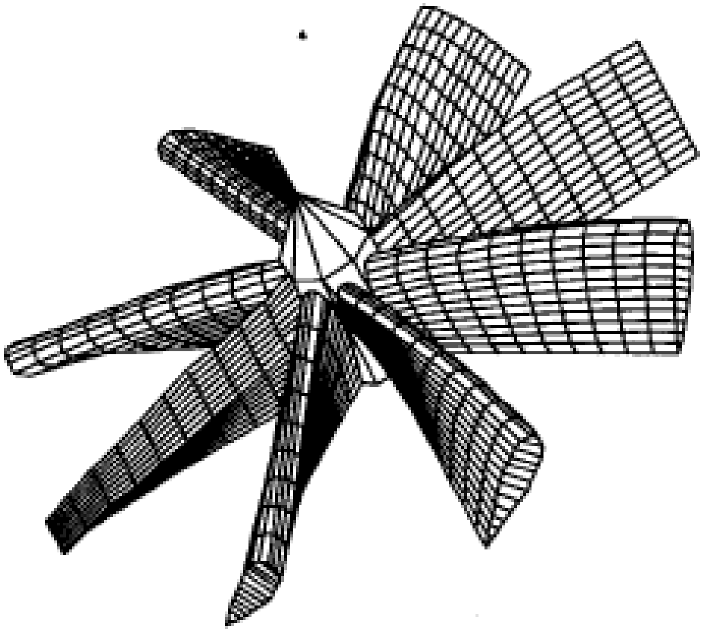
Finite element model

Simplification through Symmetry

- Cyclic symmetry**:** Similar to plane symmetry, but now described in a cylindrical coordinate system



Spline fitting

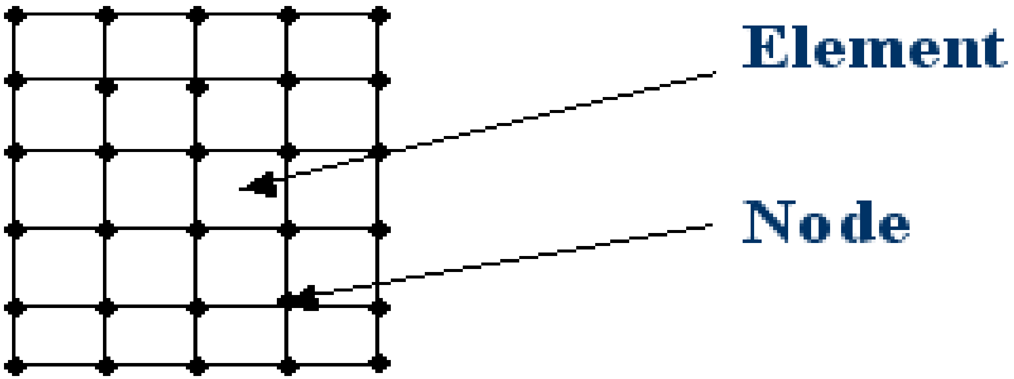


Propeller

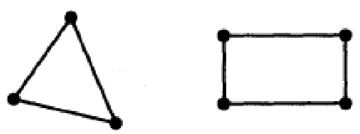
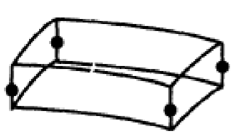
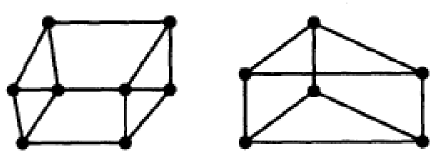
Finite Element Method

The actual structure is discretized into simple geometric shapes, each of which is assumed to behave as a continuous structural member called a finite element

- Elements are defined by corner points called nodes.

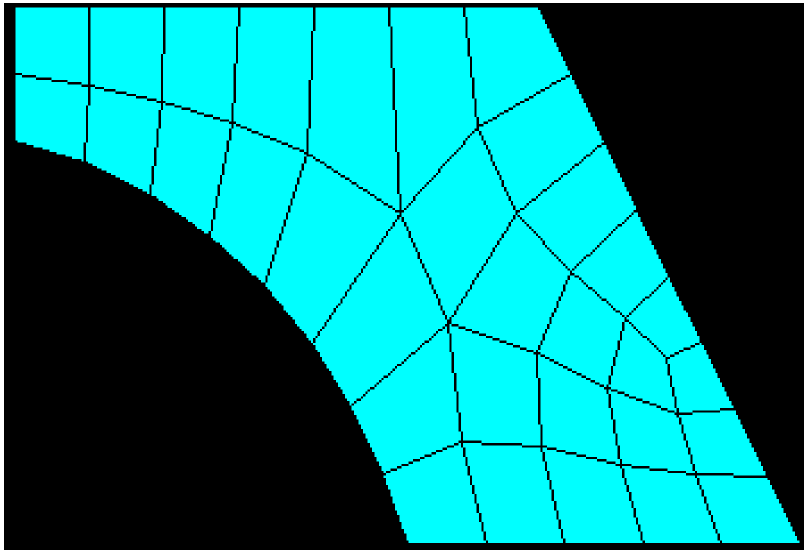


Basic Element Types - Shapes

Shape	Type	Geometry
Point	Mass	
Line	Spring, beam, spar, gap	
Area	2D solid, axisymmetric solid, plate	
Curved area	Shell	
Volume	3D solid	

Mesh Methods

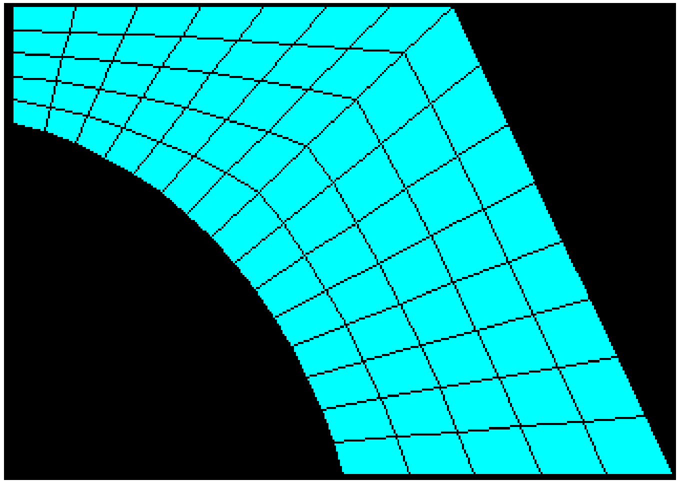
Free Mesh/Automated Mesh/Unstructured Mesh



- Has no element shape restrictions.
- The mesh does not follow any pattern
- Suitable for complex shaped areas and volumes

Mesh Methods

Mapped Mesh/Manual Mesh/Structured Mesh



- Restricts element shapes to quadrilaterals for areas and hexahedra (bricks) for volumes
- Typically has a regular pattern with obvious rows of elements
- Suitable only for “regular” areas and volumes such as rectangles and bricks

Comparison

Free Mesh

- Easy to create; no need to divide complex shapes into regular shapes
- Volume meshes can contain only tetrahedra, resulting in a large number of elements
- Only higher-order (10-node) tetrahedral elements are acceptable, so the number of DOF can be very high

Mapped Mesh

- Generally contains a lower number of elements
- Lower-order elements may be acceptable, so the number of DOF is lower
- Aesthetically pleasing
- Areas and volumes must be “regular” in shape, and mesh divisions must meet certain criteria
- Very difficult to achieve, especially for complex shaped volumes

Meshing Techniques and Guidelines

GUIDELINES

- Identify portions of the structure to be modeled with 1, 2 & 3-D elements
- Obtain global element size based on total number of elements required and ‘feel’ for the problem
- Generally, fine mesh is required for modeling important features like fillets, bolt holes, expected stress-concentration areas
- Unimportant features can either be neglected in the geometry or still be modeled with the same global size (basically to capture stiffness effects)
- Use at least four linear element divisions to turn a 90 degree arc

Meshing Techniques and Guidelines

- Use about two layers of concentric radial mesh around bolt holes. Useful for applying bolt pressures, preloads etc.
- Different techniques for modeling seam welds - rigid links, shell elements
- Techniques for modeling spot welds - rigid links, co-incident node, mesh refinement with solid elements
- Try to maintain a mesh ‘pattern’
- Use smaller element size along directions of high stress gradient and coarser element size along other directions; biasing
- Generally, good stress results require a finer mesh and good displacement results can be obtained with a coarse mesh also - Mode Shapes

Meshing Techniques and Guidelines

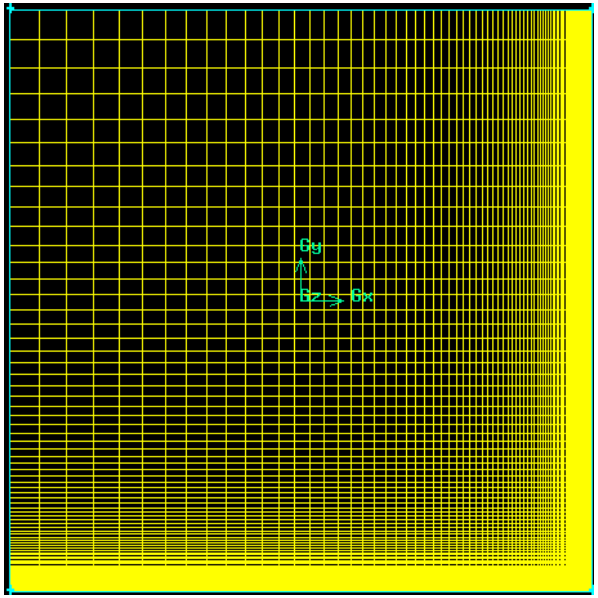
GUIDELINES

- Possible to combine shell, beam and solid elements to model a single continuum structure.
- Use at least 2 elements through the thickness when modeling thick plates and shells. Four elements will be generally adequate. About eight are required to capture through-the-thickness variations fairly accurately
- Minimize the use of triangular and wedge elements to less than about 5 per cent of the total number of elements in the model. Necessary at sharp corners.
- Make a mesh plan - particularly important for solid elements

Mesh Biasing

- One technique for improving the mesh near an area of interest is to bias mesh, which is accomplished with either solids or shells.
- Fig shows surface mesh biased towards one corner

- Biasing



Element Quality Checks

ELEMENT QUALITY ATTRIBUTES

- Different softwares have different definitions
  - Aspect Ratio : Measure of longest to shortest dimension < 10
  - Skew : Crossed, butterfly elements
  - Warping Angle : Measure of out-of-planeness for four noded quadrilaterals < 300 (loose) < 150 (tight)
  - Taper : Measure tapered vis-à-vis straight edges

Check software for definitions

  - Distortion : ‘Jacobian’, a measure of deviation from > 0.4
  - squareness, cubeness
  - Check software for definitions. Jacobian = 1, for a square or cube element
  - Internal Angles : Quadrilaterals 1350 < θ < 450
  - Triangles 1050 < θ < 150

Other Miscellaneous Topics

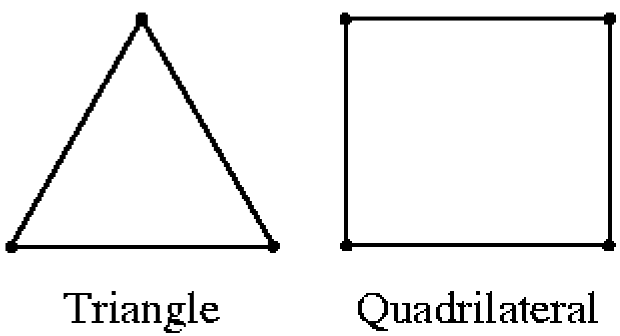
NODE AND ELEMENT NUMBERING

- **Bandwidth minimization**
  - Node renumbering
  - Several algorithms
  - Pre-processor assigns node numbers and element numbers
  - These node numbers are matched with a set of internally assigned node numbers. The internal node numbers are the result of the bandwidth minimization algorithm.
  - Efficient storage of the stiffness matrix
  - Example using a rectangular mesh
- **Wavefront minimization**
  - Element renumbering

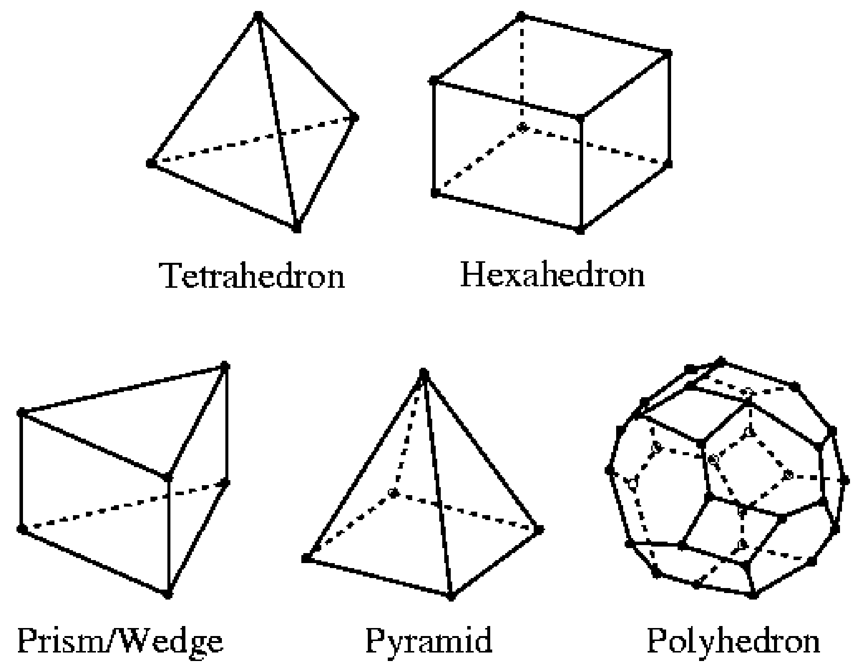
Other Miscellaneous Topics

- Many different cell/element and grid types are available Choice depends on the problem and the solver capabilities
- Cell or element types:

2D



3D

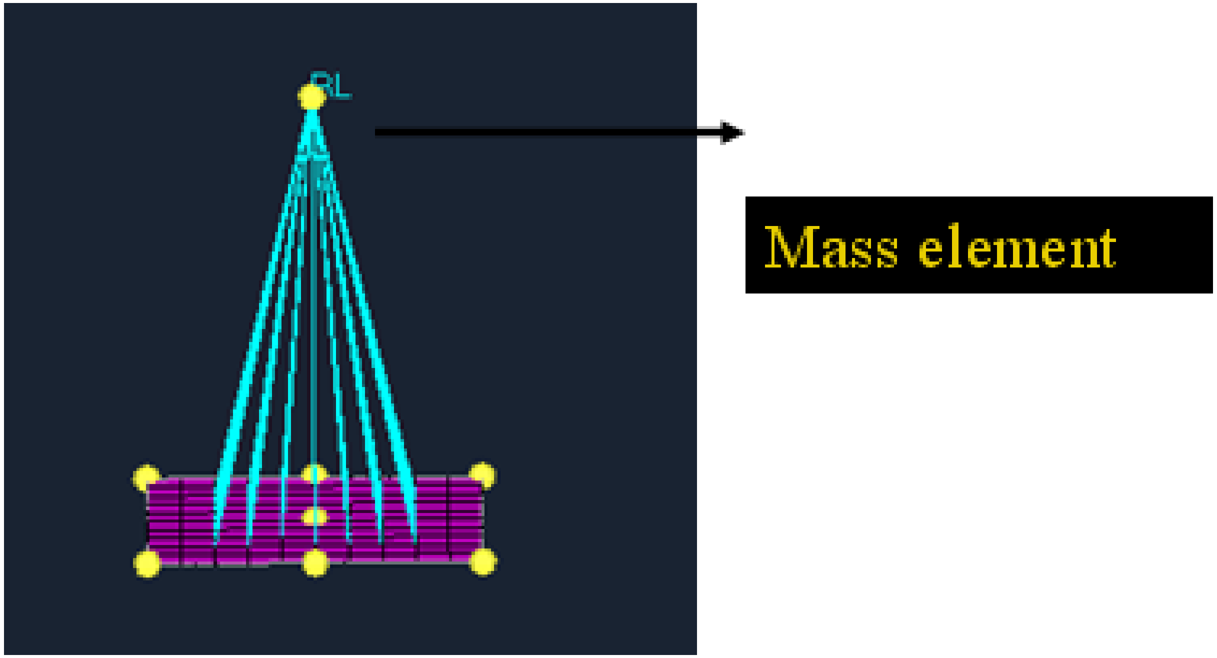


Other Miscellaneous Topics

Mass elements are used to define concentrated mass in a model. They are useful in parts of the model where there is geometry missing or there is more material than there should be (due to the oversimplification at the feature suppression stage, e.g. filled-in holes).

- Mass element is zero dimension or point element and is applied at cg of component.
- To connect the mass with structure, rigid or multi point constraint elements are used

- Mass element is frequently used for dynamic and NVH applications. And also it is very good option for restricting total dof's of model
- Person sitting on stool replaced by concentrated mass at CG



- Gap element is an air between components
- Gap element-used to define a contact between two surfaces. On the creation of the elements, a set of slave nodes and master nodes are defined. Contact occurs if either group of nodes attempts to penetrate the other. Friction properties can usually be applied to the element.
- When force is applied on upper plate it will deflect till it comes in contact with bottom plate then both the plates will deflect together, when this broblem is solved in fea without defining Gap elements or contact the results will be….i.e the bottom plate not taking any load.(stress=0)
- As with gap elements, they are usually NOT recommended for use with higher order elements

Geometric Properties of Elements

• **Line/Rod Elements**	:	Cross-Section Area
• **Beam Elements**	:	Cross-Section Area,Moments of Inertia,Height of the Beam
• **Plate or Shell Elements**	:	Uniform thickness or Varying thickness,if it is a Composite material, need to specify No.of Layers,Fibre thickness and fibre orientation
• **Spring Elements**	:	Stiffness value (Transnational Stiffness or torsional Stiffness)
• **Mass Elements**	:	Mass/Inertia values in different directions
• **Plane Elements***	:	No need to specify any geometric properties
• **Solid Elements***	:	No need to specify any geometric properties

Geometric Properties of Elements

Rigid elements are generally used to model rigid parts of a structures without having to assign the computationally expensive usual elastic elements. They are useful in dynamic analysis to account for distributed mass and inertia. They are also regularly used in kinematic type simulations away from any areas of interest in the model.

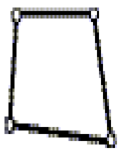
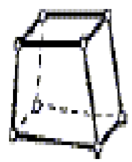
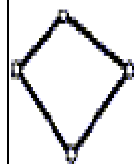
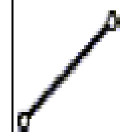
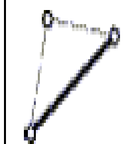
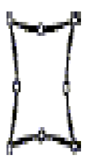
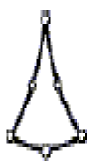
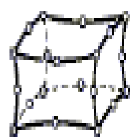
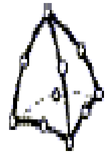
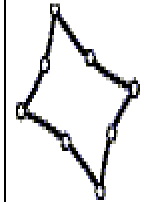
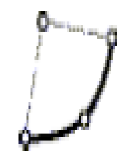
Coupling elements are used to couple two independent meshes at an interface. They are particularly useful when two parts of the model were created independently of each other. There are variants for coupling surfaces to surfaces, beams to surfaces and shell edges to surfaces.

Choosing Right Element for Structural Analysis

Picking of an element type from the large library of elements in Ansys,

- Out of the almost 200 available which one should select?
- Are there 181 kinds of shell elements?
- Type of component(1D,or 2d, or 3D)
- Type of Analysis
- Type of loads
- Resource available
- Idealization of problem (assumptions made to simplify the problem)

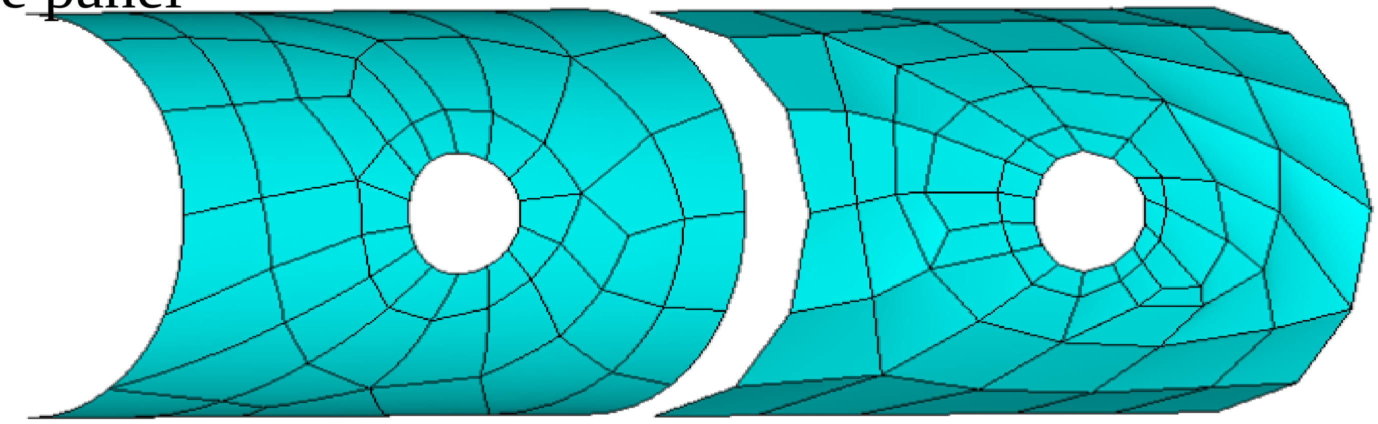
Commonly used Elements Types for Structural Stress Analyses

Element Order	2D Solid	3D Solid	3D Shell	Line Elements
Linear	 PLANE42 PLANE182	 SOLID45 SOLID185	 SHELL63 SHELL181	 BEAM3/44  BEAM188
Quadratic	 PLANE82/183  PLANE2	 SOLID95/186  SOLID92/187	 SHELL93	 BEAM189

Commonly used Elements Types for Structural Stress Analyses

- The element order refers to the interpolation of an elements nodal results to the interior of the element, and determines how results can very across an element.
- Plate is bending is a good example of this phenomenon, where strain is changing sign over a potentially very small distance, here element can linear or quadratic.
- The linear elements do not have midsize nodes. So strain can only vary linearly from one node to another.which is complicated in Ansys with many linear elements having so called extra shapes or being fully integrated. This makes elements behave more like quadratic element.
- Naturally a linear element is computationally faster than quadratic element.

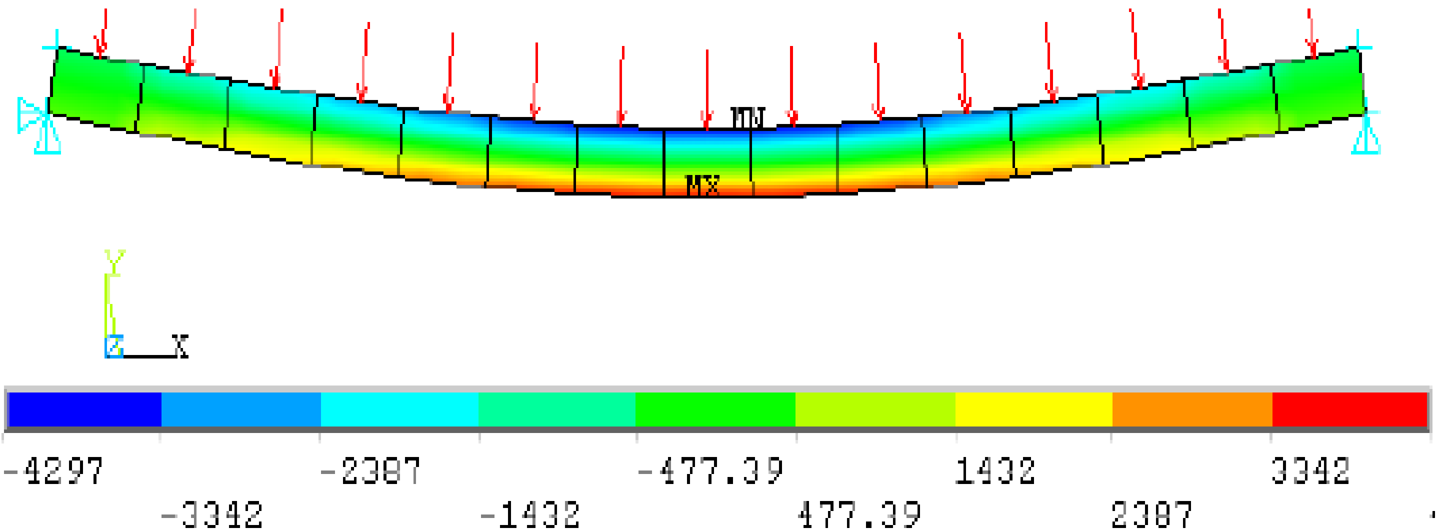
- The quadratic have mid side nodes. The shape function for strains varies in some nonlinear fashion between the corner nodes.
- Where as linear elements are flat on both sides and in- plane , a quadratic element can fallow a curvature in both directions and is more accurate for a given no of nodes in the model
- For example in a tube with a hole, the curvature is maintained around the edges of the hole as well as on the curved edge of the panel



SHELL93 Quadratic Elements (8 nodes) SHELL181 Linear Elements (4 nodes)

Considerations in Picking an Element Order

- For in plane bending type problems, quadratic elements are much better than linear elements. Even several linear elements with extra shapes or full integration turned on are not as good one quadratic element



SHELL93 Quadratic (8 noded) Default Options

Considerations in Picking an Element Order

- A linear element is probably better in a mapped mesh than a free mesh
- A finer mesh of linear elements is computationally more efficient, robust, stable and more accurate than coarser quadratic mesh with a comparable number of mesh

2d solid elements

How can 2d element be a solid?

That's a question most new analyzer ask.

There are two reasons , firstly they are considered solid because they are not a shell, or a beam element, secondly , they are solids because they fully represent a solid chunky part by modeling a **cross section** of the part

Conclusions

- Engineering Design Process have been dealt in brief
- Sources of Error in the FEM have been dealt in brief
- Advantages and Disadvantages of FEM have been dealt in brief
- Covered the basic Steps in the Finite Element Method

Thank You