AU424 - Finite Element Methods and Analysis

Unit -5 – *Computer Implementation*

Contents:

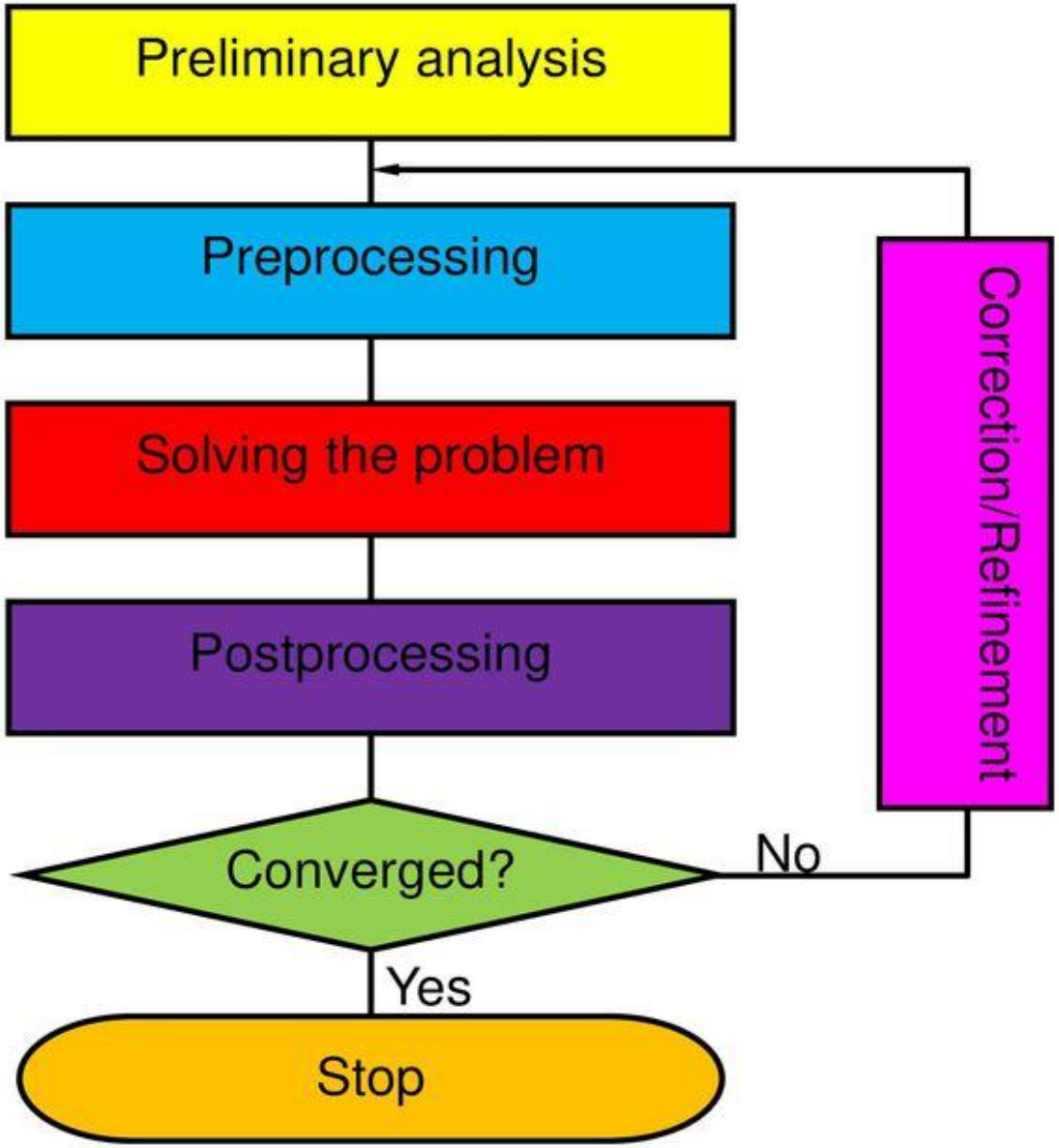
- ✓An overview of FE Analysis Program
- ✓Preprocessing
- ✓Solution
- ✓Post Processing

Prepared by
Mr. D. Rajesh Kumar, AP/Auto
SNSCT

FINITE ELEMENT PROCEDURE

- **Discretization**: dividing the structure into a set of simple-shaped, contiguous elements, connected by sharing nodes
- Nodal displacements are unknown DOFs
- Element level matrix equations are **assembled** to form global level equations
- Specify **displacement BC** and **applied loads**
- The global matrix equations are solved for the unknown DOFs
- From the displacements at the nodes, calculate **strains** and then **stresses** in each element
- Difficulties
  - How to ***model*** the problem using finite elements?
  - ***What kind*** of elements and ***how many*** elements should be used?
  - How the BCs and loads should be specified?
  - How to ***interpret*** the results?

FINITE ELEMENT PROCEDURE *cont.*



PRELIMINARY ANALYSIS

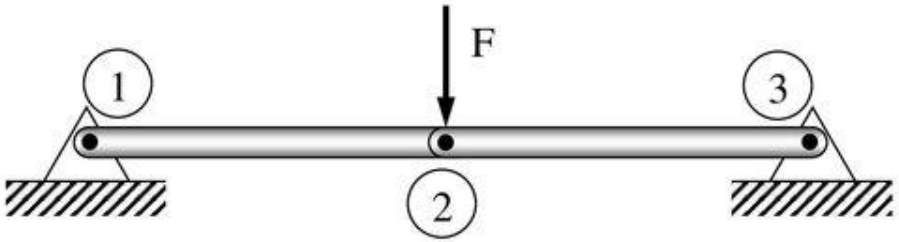
- One of the most important steps in FEA procedure
- Often ignored by many engineers
- Provide an insight into the problem and predict behavior
- Use analytical methods to estimate the expected solution (FBD, equilibriums, mechanics of materials, etc)
- Simplify the problem using bars and beams
- Predict level of displacement and stress as well as critical locations
- **Before FEA, engineers should know the range of expected solution and candidates of critical locations**

PREPROCESSING

- preparing a model for finite element analysis
  - Modeling a physical problem using finite elements
  - Choosing types and number of elements
  - Applying displacement boundary conditions
  - Applying external loads
- **The finite element model is not a replication of the physical model, but a mathematical representation of the physical model**
- Finite element model can be different from physical model.
  - One or two beam elements for the complex space rocket system if the interest is in the max bending moment of the rocket.
  - The plate with a hole can be modeled using plane stress elements with the thickness

PREPROCESSING *cont.*

- The behavior of FE model is different from that of physics

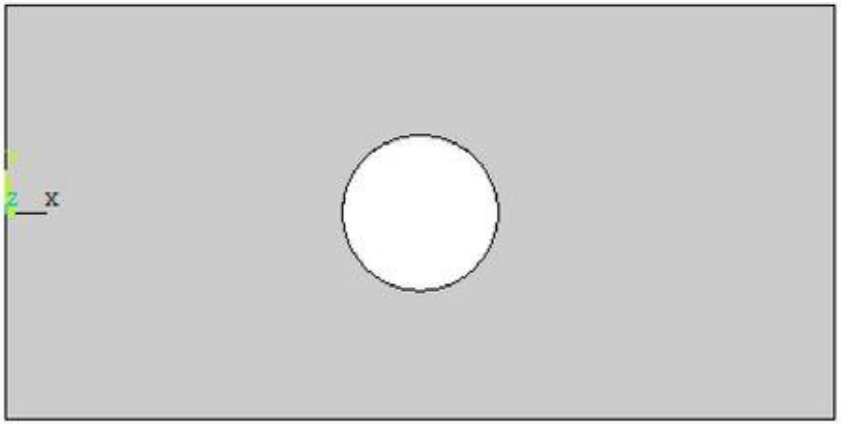


$$\frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -1 & 0 & 2 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix} = \begin{Bmatrix} R_{1x} \\ R_{1y} \\ 0 \\ -F \\ R_{3x} \\ R_{3y} \end{Bmatrix} \Longrightarrow \frac{EA}{L}\begin{bmatrix} 2 & 0 \\ 0 & 0 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -F \end{Bmatrix}$$

- No stiffness in the vertical direction!

PREPROCESSING *cont.*

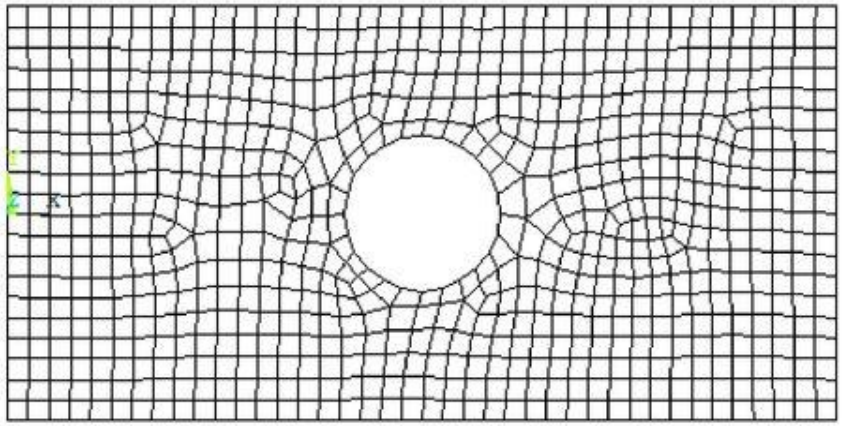
- Units
  - **STUPIDEST mistakes come from UNITS!**
  - Consistent units must be used throughout FE procedure
  - In SI unit, order of deformation ~ 10^{-6}m, order of stress ~ 10^{8}Pa
- Automatic mesh generation
  - Many commercial programs can automatically generate nodes and elements using GUI
  - Work with solid model



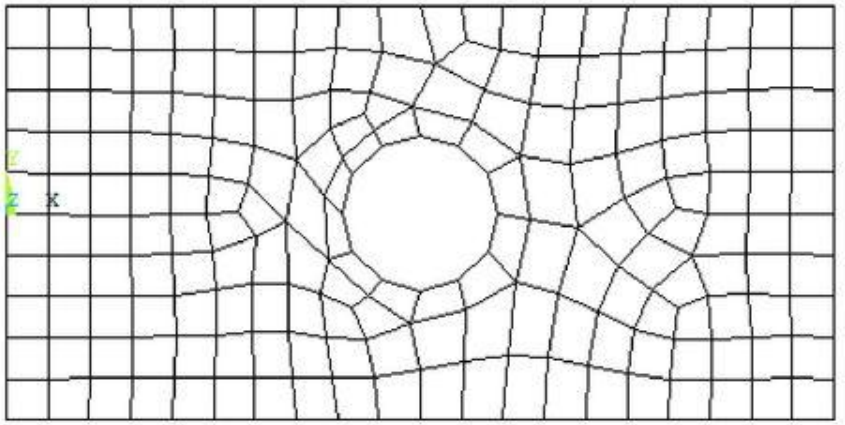
PREPROCESSING *cont.*

- Mesh control
  - Provide mesh parameters that define the size and type of elements and other attributes
  - Global or local element size, curvature-based element size
  - Smaller element size for location of interest

Element size = 0.1

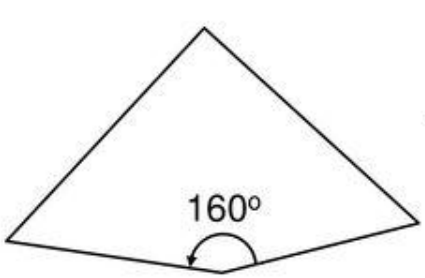


Element size = 0.2

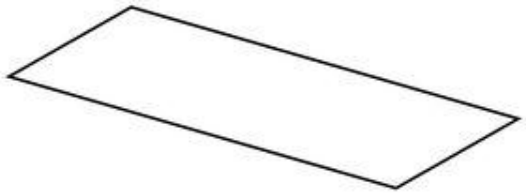


PREPROCESSING *cont.*

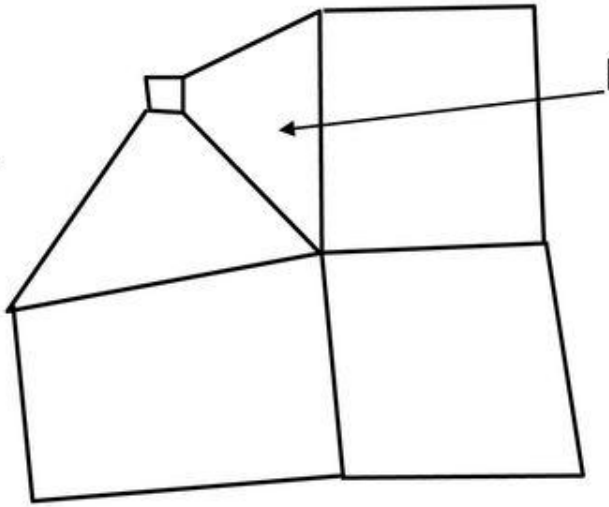
- Mesh quality
  - **A good quality mesh is a recipe of success in finite element analysis**
  - Element shape: Best for square element
  - Aspect ratio: Large aspect ratio elements should be avoided
  - Element size: Quick transition from small to large elements should be avoided
  - Smaller elements must be used where stresses change quickly



Distorted element

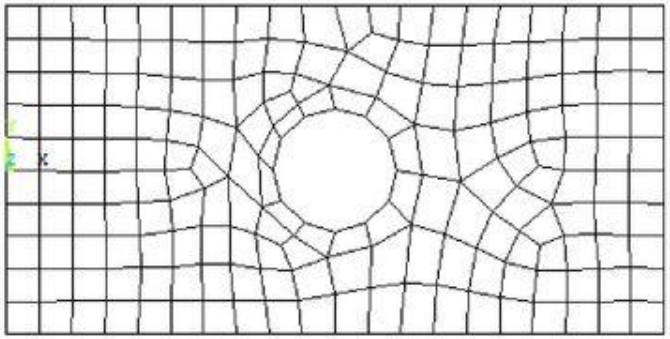


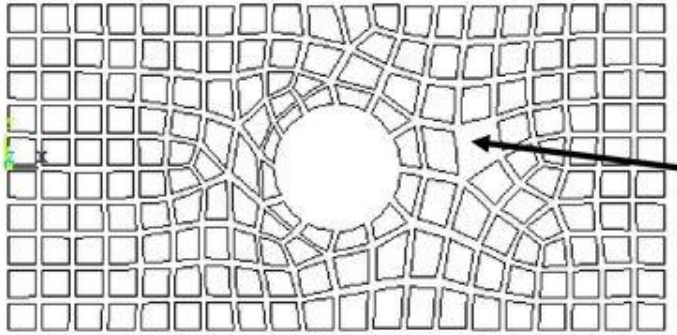
Large aspect ratio



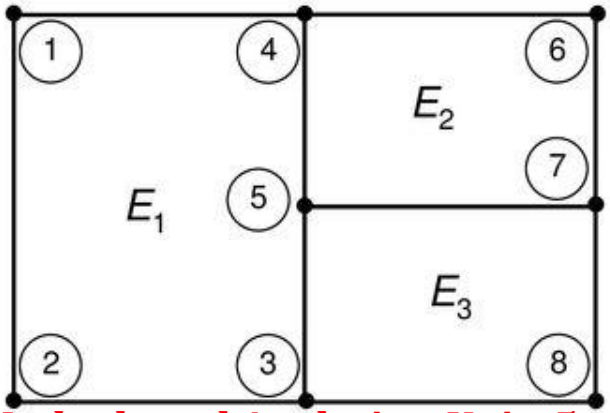
PREPROCESSING *cont.*

- Checking the mesh
  - Duplicated nodes: Two nodes at the same location are associated with different elements; artificial crack in the model
  - Missing elements: Can be detected using shrink plot of elements





  - Mismatched boundary: Produce artificial crack

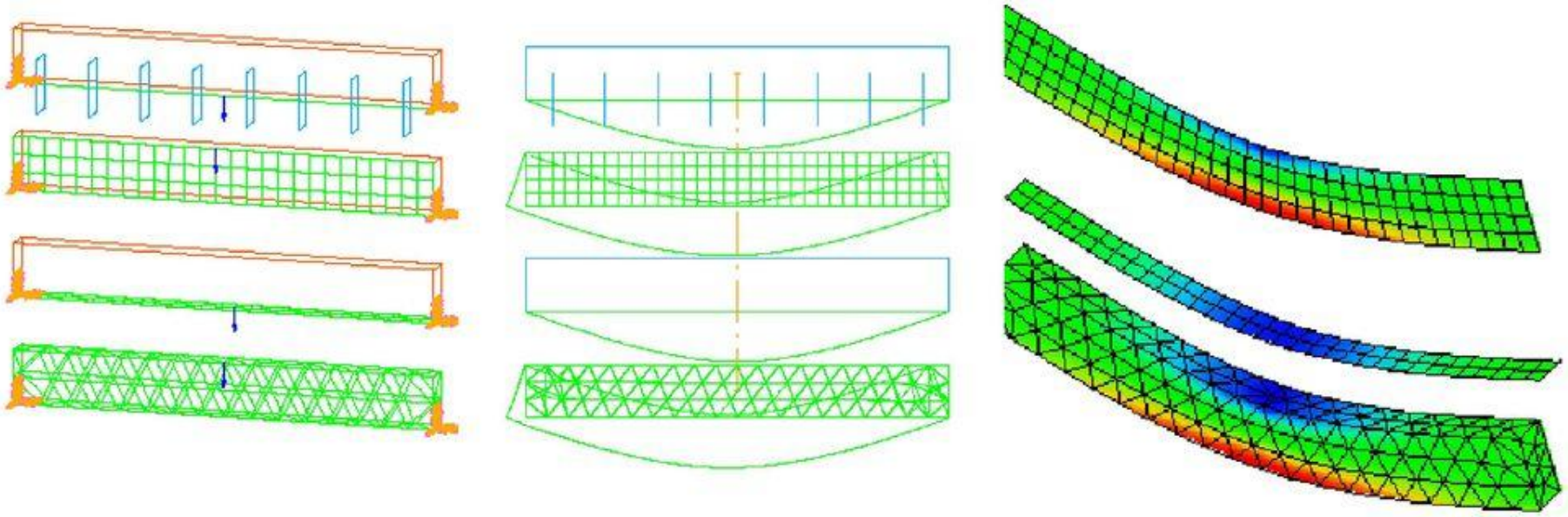


PREPROCESSING *cont.*

- Material properties
  - Isotropic, linear elastic material: Young's modulus, shear modulus, Poisson's ratio
  - Only two are independent
  - Sometimes, failure stress is required for estimating safety
  - Anisotropic material, composite material, elasto-plastic material, etc
  - Unit of material properties must be consistent with that of FE model

PREPROCESSING *cont.*

- Choosing Element Type and Size
  - Different elements and models can be used for solving the same problem
  - Engineers should understand the capability of the elements and models so that proper elements should used

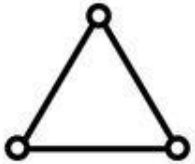
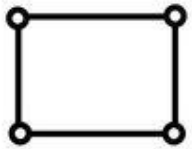
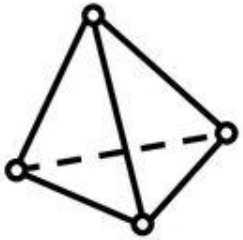
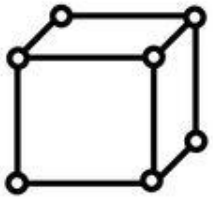


PREPROCESSING *cont.*

- Solid element:
  - Can represent structural details, but computationally expensive
- Shell/plate element
  - The sheet or plate can be represented using 2D plane with thickness
  - More efficient than solid element
  - Good for thin wall where bending and in-plane forces are important
- Beam/frame element
  - Most efficient way of modeling
  - Good for predicting the overall deflection and bending moments of slender member
  - Limited to predicting local stress concentrations at the point of applied load or at junction

PREPROCESSING *cont.*

Element Types

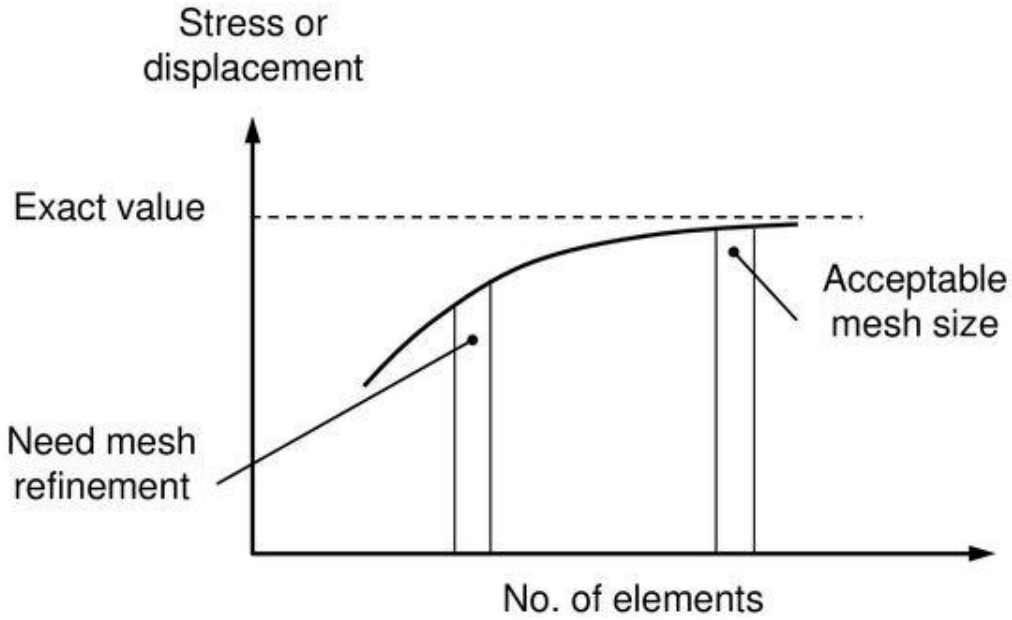
Element	Name
	1D linear element
	2D triangular element
	2D rectangular element
	3D tetrahedral element
	3D hexahedral element

PREPROCESSING *cont.*

- Element order
  - We only learned linear elements
  - **Linear elements**: 2-node bar, 3-node triangular, 4-node quadrilateral, 4-node tetrahedral, 8-node hexahedral elements
  - **Parabolic elements**: 3-node bar, 6-node triangular, 8-node quadrilateral, 10-node tetrahedral, 20-node hexahedral elements
  - **Cubic elements**: 4-node bar, 9-node triangular, 12-node quadrilateral, 16-node tetrahedral, 32-node hexahedral elements
- Linear elements have two nodes along each edge, parabolic have three, and cubic have four.
- A higher-order element is more accurate than a lower-order element

PREPROCESSING *cont.*

- How to choose element size?
  - Critically important in obtaining good results
  - Preliminary analysis can help
- Is the size proper? (Error analysis and convergence analysis)
  - Mesh refinement improves solution accuracy.
  - How small is good enough?



PREPROCESSING *cont.*

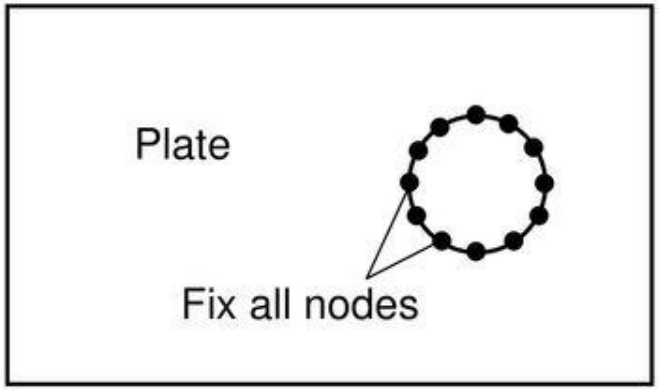
- Convergence rate
  - Calculate the function of interest at three different meshes
  - Let h_1, h_2, and h_3 be the sizes of elements, ordered by $h_1 > h_2 > h_3$
  - Usually $h_1 = 2h_2 = 4h_3$
  - The ratio in difference

$$\frac{\left\| u_{h_3} - u_{h_2} \right\|}{\left\| u_{h_3} - u_{h_1} \right\|} \approx \left(\frac{h_2}{h_1} \right)^{\alpha}$$

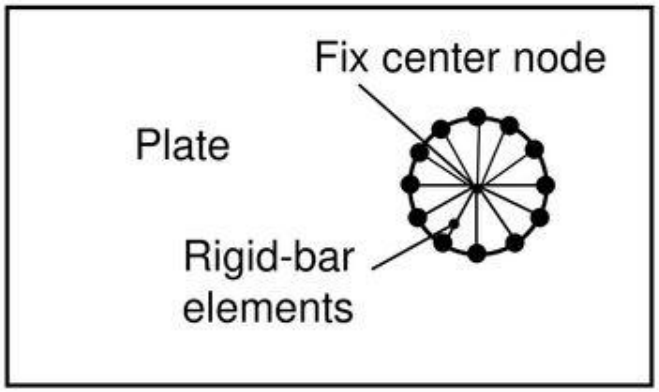
  - **Convergence rate** α: indicates how fast the solution will converge to the exact one

PREPROCESSING *cont.*

- Applying displacement boundary conditions
  - **FE model should be properly restrained so that it is not free to move in any direction even if there are no applied forces in that direction**
  - Errors in BC will not disappear no matter how much you refine the model
  - Any unexplained high stress may be due to a wrong boundary condition



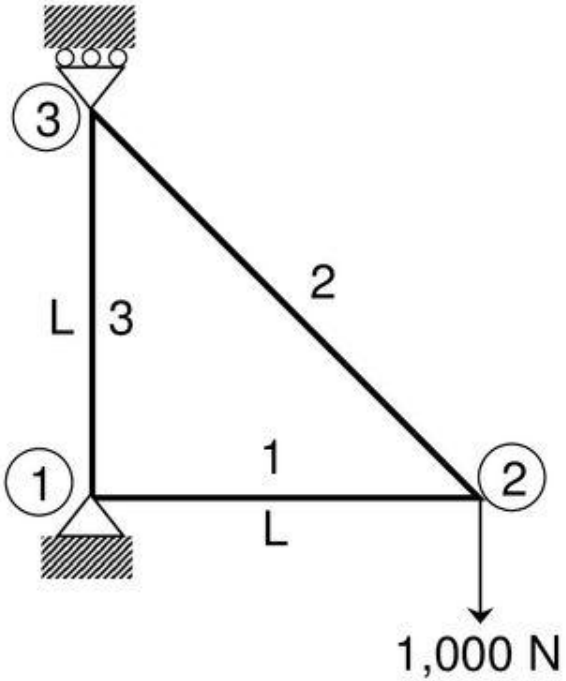
Not allowed to translate/rotate



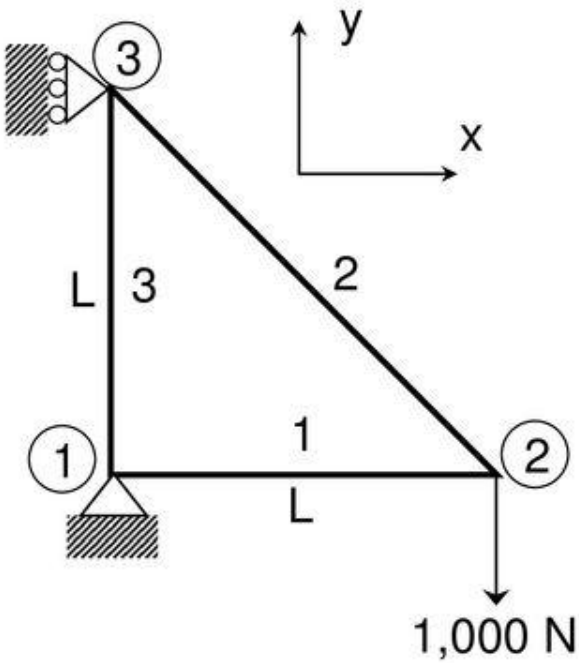
Not allowed to translate

PREPROCESSING *cont.*

- Example of error in BC



(a) Improper case

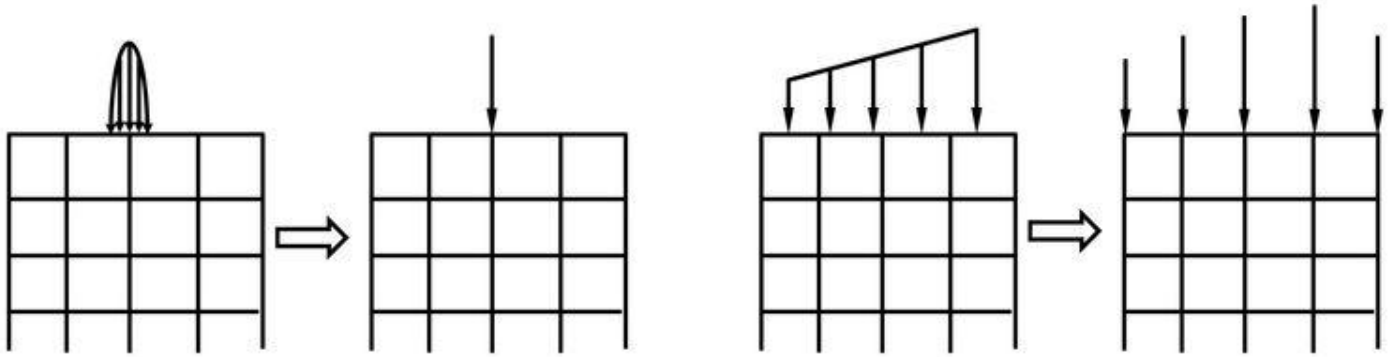


(b) Proper case

PREPROCESSING *cont.*

- Applying external forces
  - Forces are applied through a complex mechanism
  - It is often simplified when the interest region is far from the load application location
  - FE results near the load application location are not accurate due to approximation involved in the force
- Applying a concentrated force
  - Theoretically infinite stress (zero area)
  - Practically, all forces are distributed in a region
  - Concentrated force in FE is an idealization of distributed forces in a small region

PREPROCESSING *cont.*

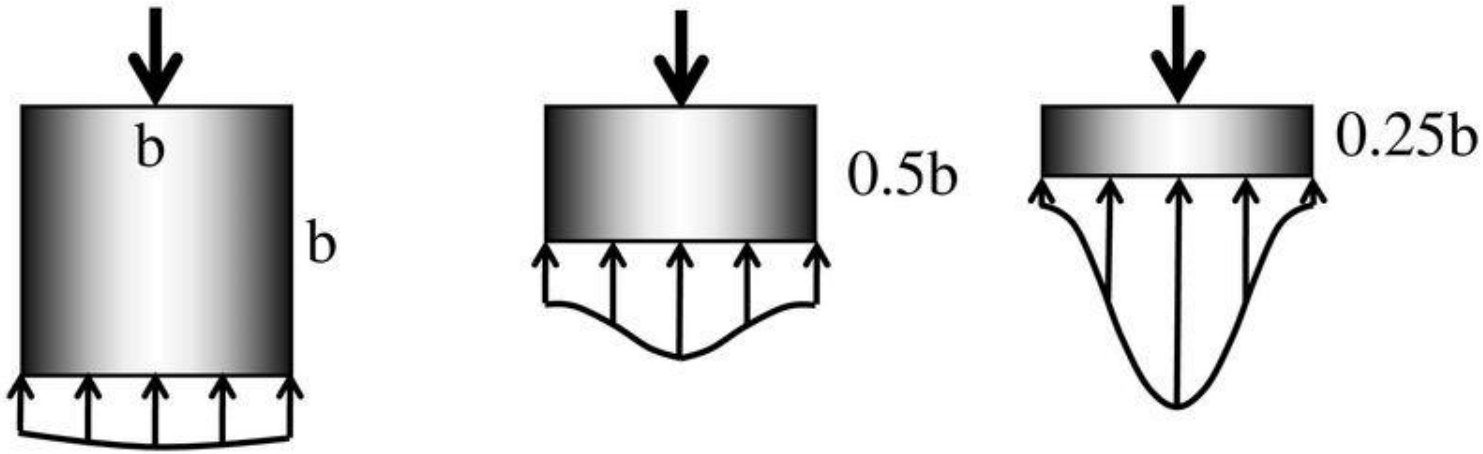


(a) Concentrated force (b) Distributed forces

- Note that the distributed forces are converted to the equivalent nodal forces.
- All applied forces must be converted to the equivalent nodal forces because the RHS of finite element matrix equations is the vector of nodal forces.

PREPROCESSING *cont.*

- St. Venant's principle
  - If the interest region is relatively far from the force location, the stress distribution may be assumed independent of the actual mode of application of the force



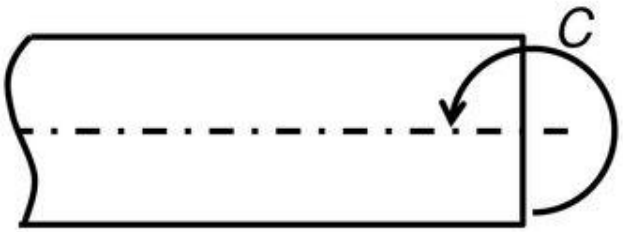
$\sigma_{min} = 0.973\sigma_{ave}$
$\sigma_{max} = 1.027\sigma_{ave}$

$\sigma_{min} = 0.668\sigma_{ave}$
$\sigma_{max} = 1.387\sigma_{ave}$

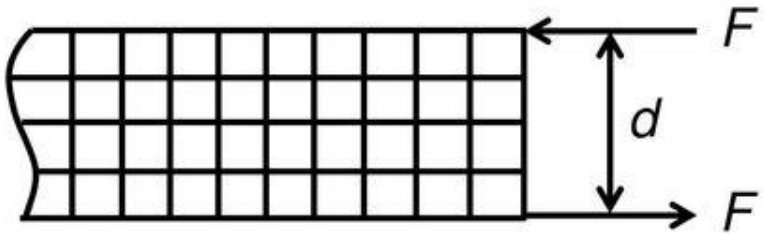
$\sigma_{min} = 0.198\sigma_{ave}$
$\sigma_{max} = 2.575\sigma_{ave}$

PREPROCESSING *cont.*

- Applying a couple to a plane solid

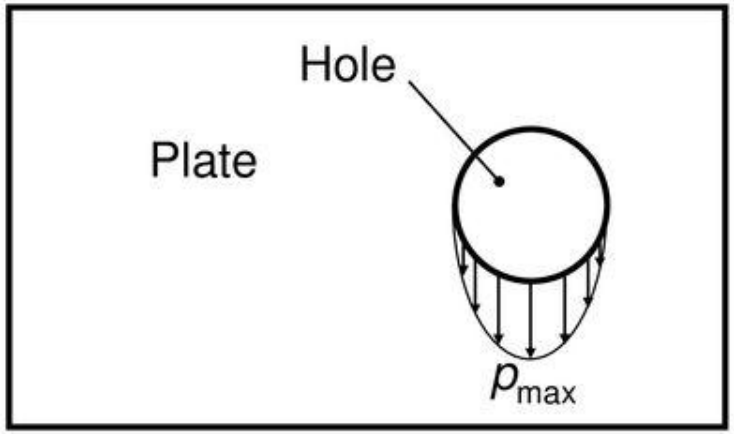


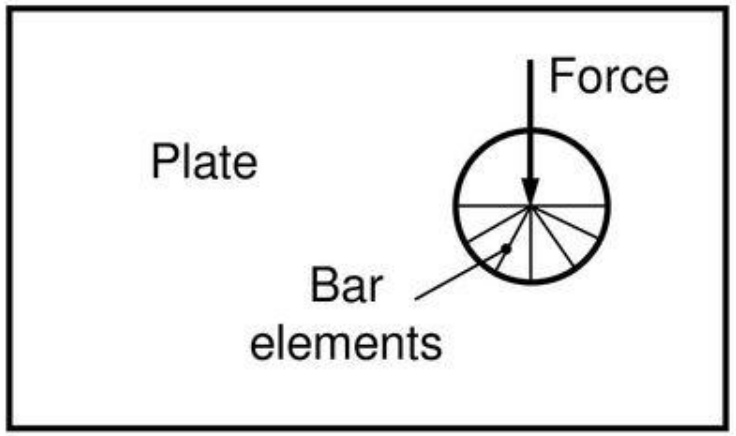
(a) Beam element



(b) Plane solid elements

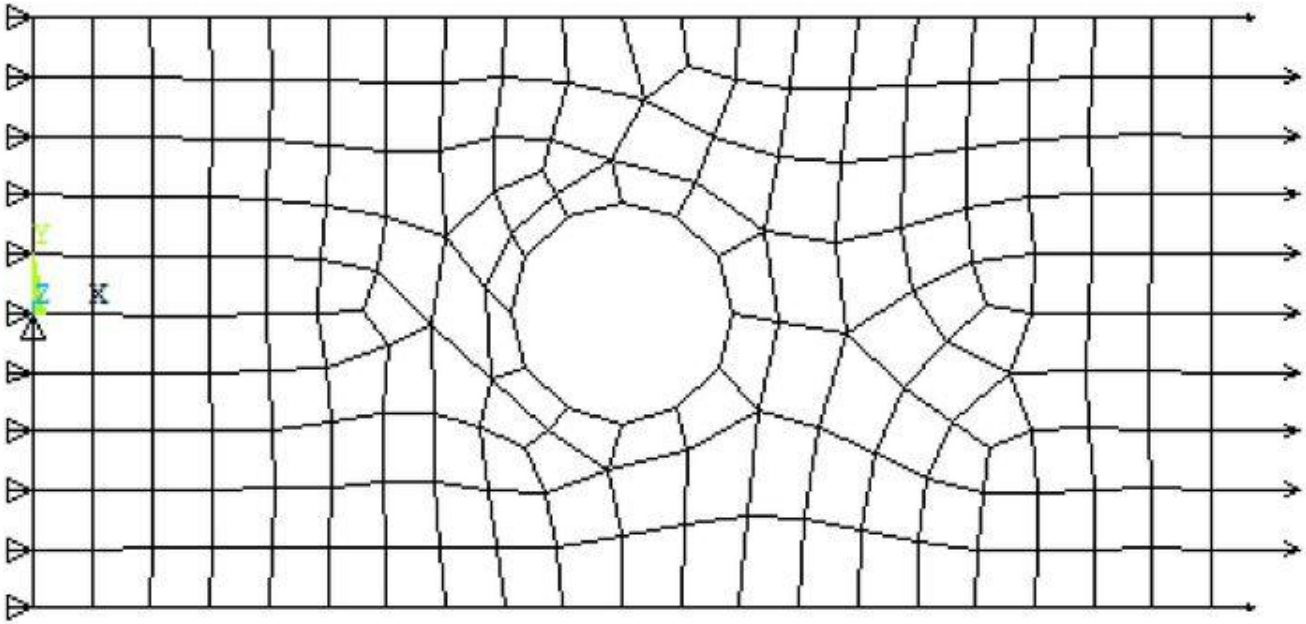
- Applying a force through shaft





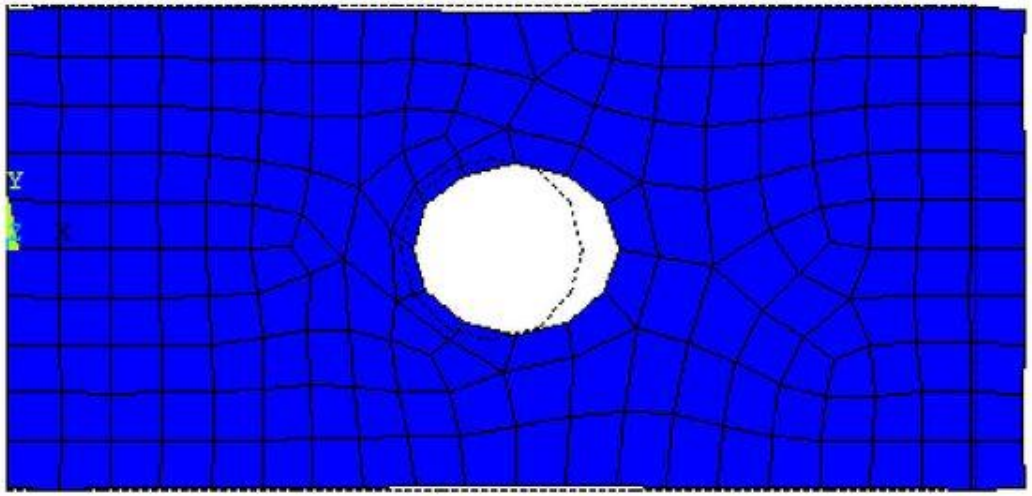
PREPROCESSING *cont.*

- Plate with a hole example
  - All nodes on the left edge are fixed in x-direction
  - node at the center of the left edge is fixed both in x- and y-direction
  - uniform pressure 600 psi , which is equivalent to the 300 lb, is applied on the right edge



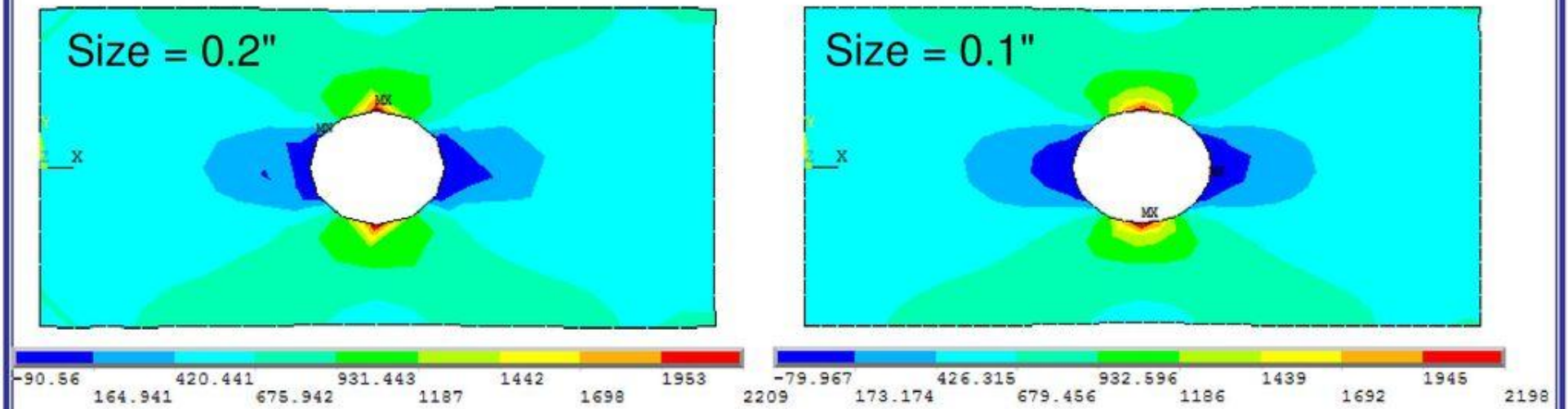
POSTPROCESSING

- Review analysis results and evaluate the performance
  - Engineer must have a capability in interpreting FEA results
  - Requires knowledge and experience in mechanics
  - Engineer can check any discrepancy between the preliminary analysis results and the FEA results
- Deformed shape display
  - Strong tool to understand the mechanism of structural behavior
  - Can verify if the displacement and forces are correctly applied
  - Deformation is often magnified such that it can be visible



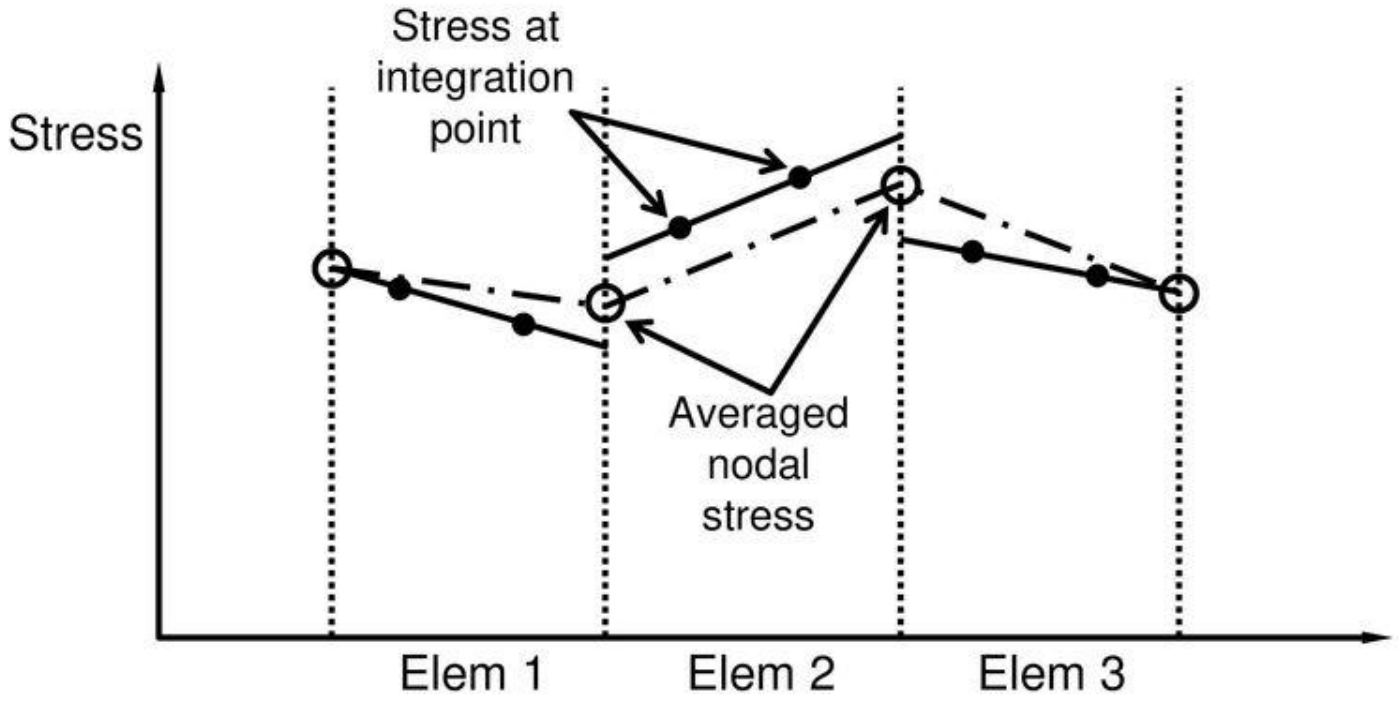
POSTPROCESSING *cont.*

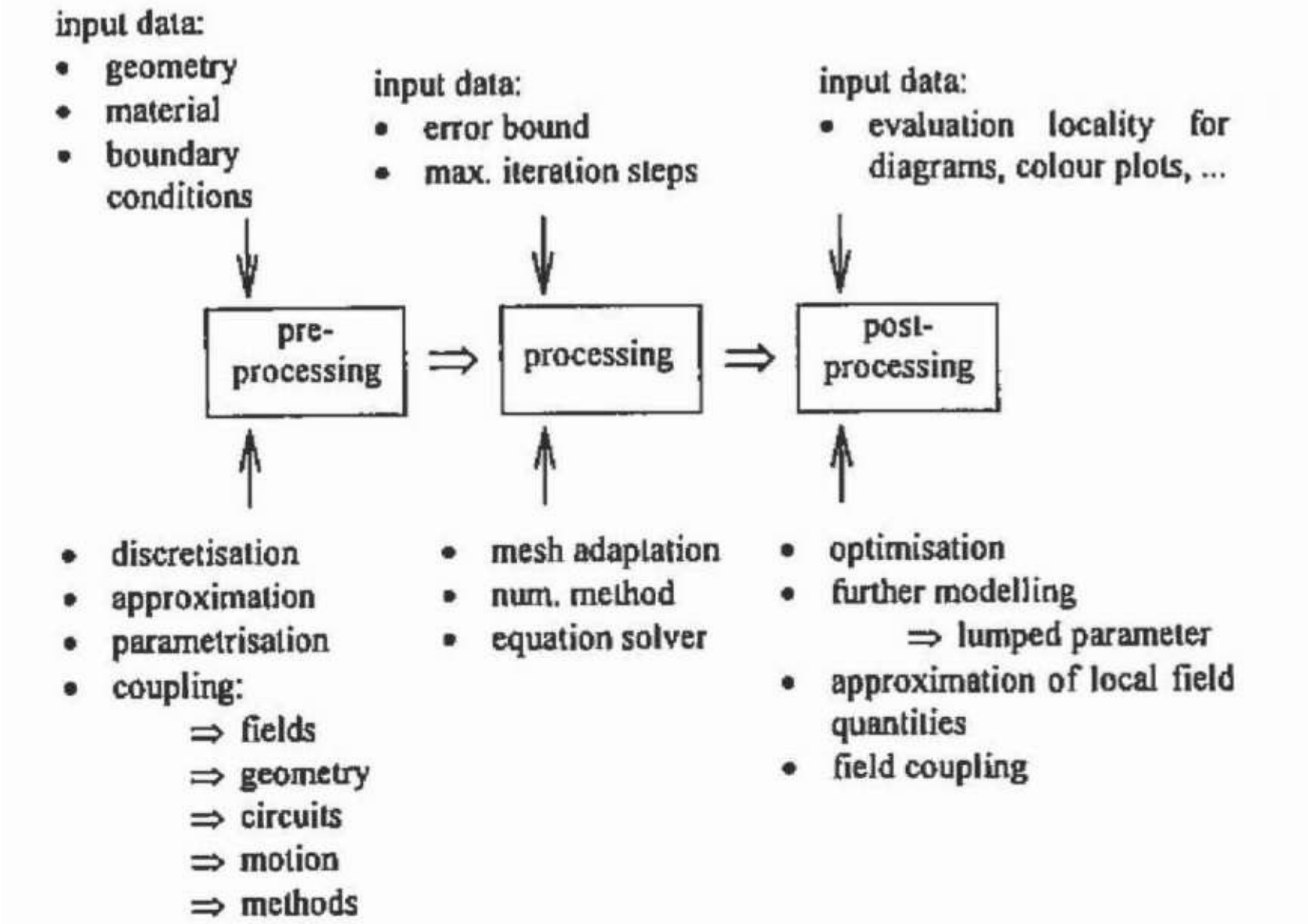
- Contour display
  - Understand the distribution of the stress in the structure and identify the most critical locations
  - Max stress 2,209 psi is 6% higher than that from preliminary analysis results (2,083 psi)
  - Accurate stress values at Gauss integration points are extrapolated to nodes
  - Refined model has 2,198 psi (.5% change from the initial model)



POSTPROCESSING *cont.*

- Stress averaging
  - Contour-plotting algorithms are based on nodal values
  - Stress is discontinuous at nodes
  - Extrapolated stresses are averaged at nodes -> Cause error
  - Difference b/w actual and averages stress values are often used as criterion of accuracy




input data:
• geometry
• material
• boundary conditions
input data:
• error bound
• max. iteration steps
input data:
• evaluation locality for diagrams, colour plots, ...
pre-processing
processing
post-processing
• discretisation
• approximation
• parametrisation
• coupling:
⇒ fields
⇒ geometry
⇒ circuits
⇒ motion
⇒ methods
• mesh adaptation
• num. method
• equation solver
• optimisation
• further modelling
⇒ lumped parameter
• approximation of local field quantities
• field coupling