# Sources of Error

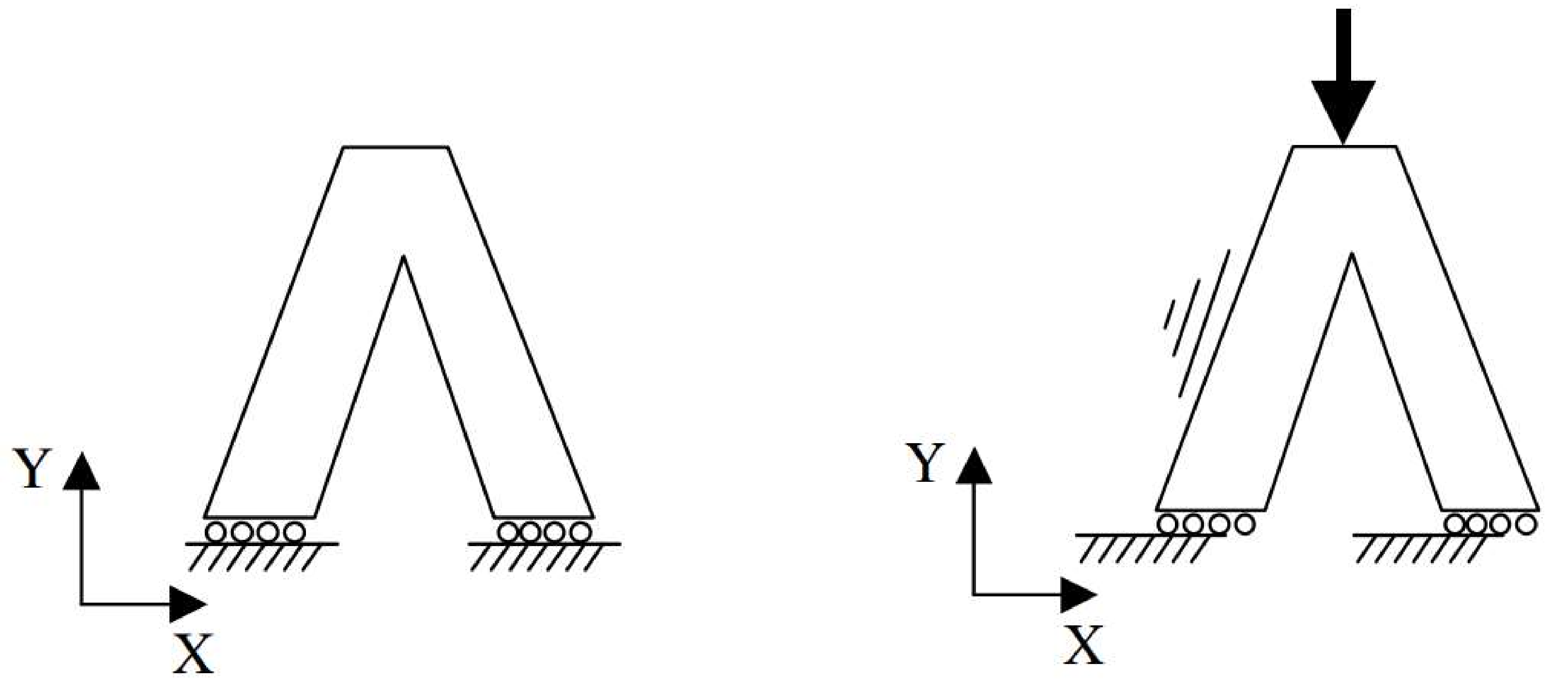
- Causes of incorrect results:
- Mistakes (e.g., forgetting a load)
- Errors
  - Modeling Error (due to simplifying assumptions in mathematical model such as when using beam elements we assume that the cross-sections stay planar and do not change shape)
  - Discretization Error (i.e., due to piecewise approximation which can be minimized by using higher order shape functions or smaller elements)
  - Numerical Error (due to limited number of significant digits maintained by computer)

# Mistakes

- Common mistakes that will cause a singular K matrix (and therefore no results):
  - v = 0.5 in a plain strain, axisymmetric or 3D solid element
  - E = 0 in an element
  - No supports, or insufficient supports
  - Part of the model is a mechanism
  - Large stiffness differences
  - In an element with stress-stiffening, negative stiffening has reduced the stiffness to zero
  - In nonlinear analysis, supports or connections have reached zero stiffness

# Common Mistakes

- Insufficient supports will allow rigid body motion.
- The stiffness matrix will be singular.

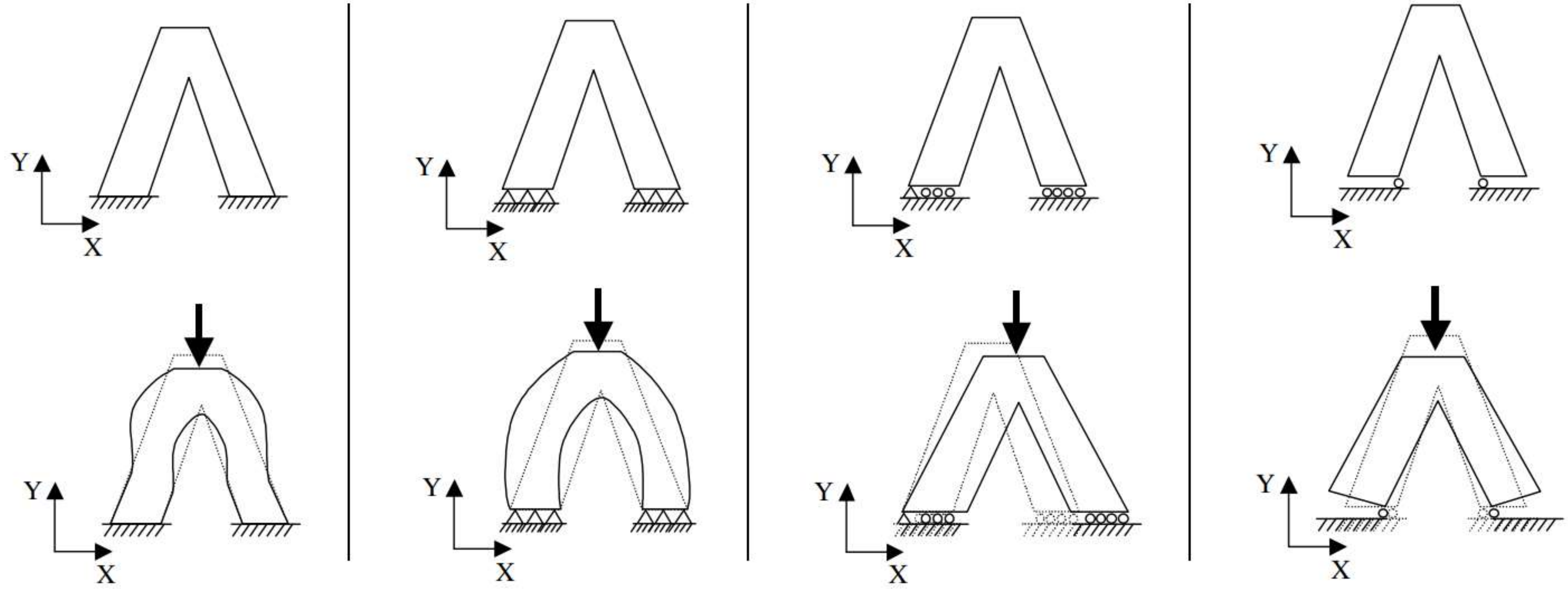


# Common Mistakes

- Incorrect element data (e.g., wrong thickness, beam cross-section, cross-section dimension, beam orientation)
- Supports wrong in location, type or direction
- Loads wrong in location, type, direction or magnitude
- Units mix-up
- A force or mesh defined twice and/or on different duplicated geometry
- Connections not working as intended (e.g. beam element connected to plane element does not transfer moment)

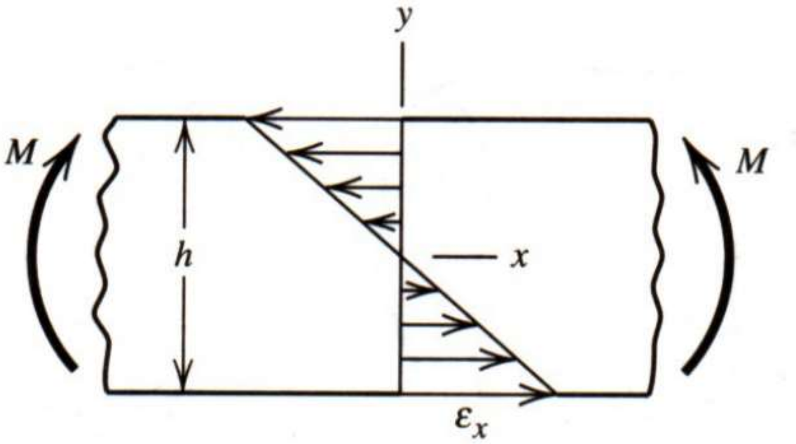
# Effect of wrong support types

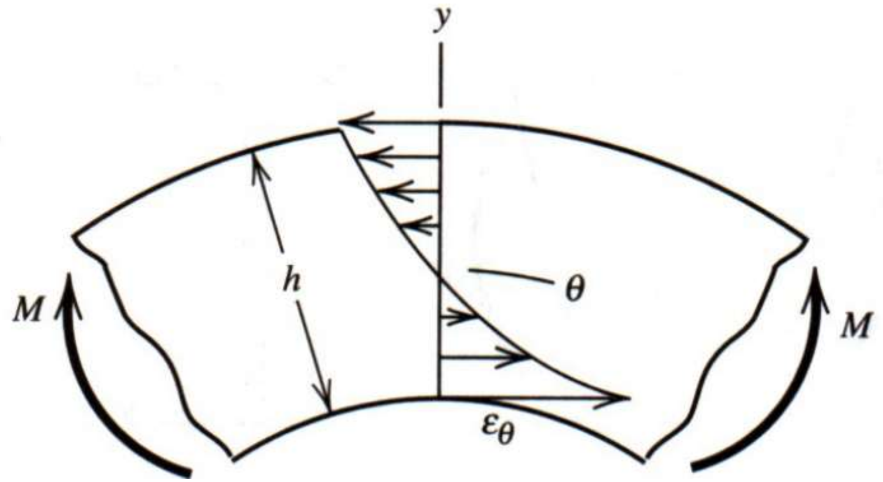
- Each of these will result in different displacements, strains and stresses



# Modeling Error

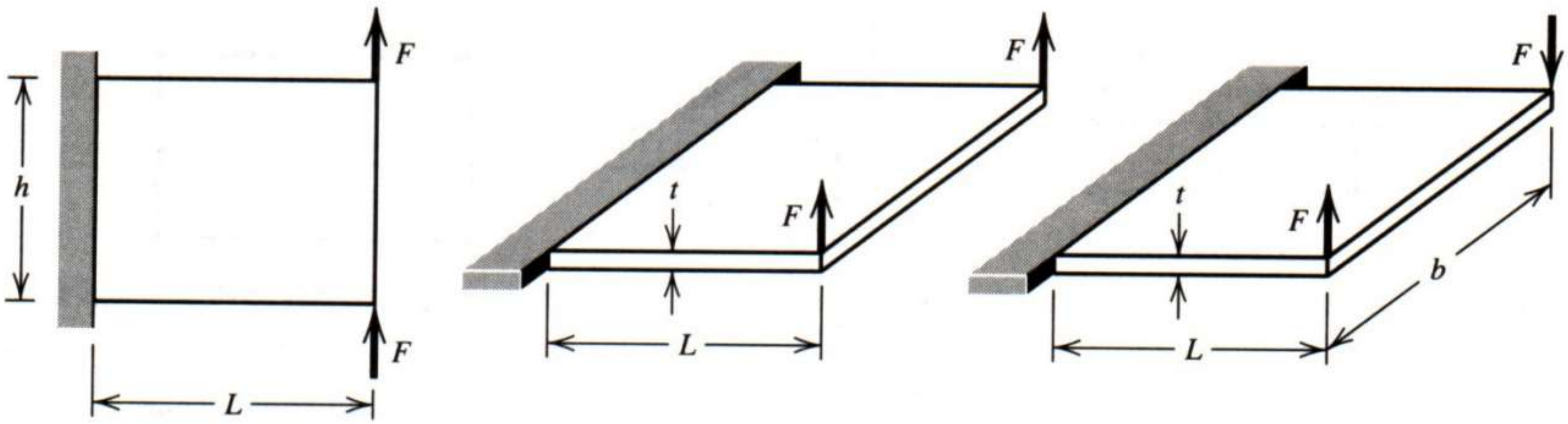
- To do a proper FE analysis, the analyst must understand how the structure is likely to behave and how elements are able to behave.
- E.g., if the analyst knows the displacement varies linearly, 4-node quad. elements will work, but if they vary quadratically, 8-node quad. elements must be used.



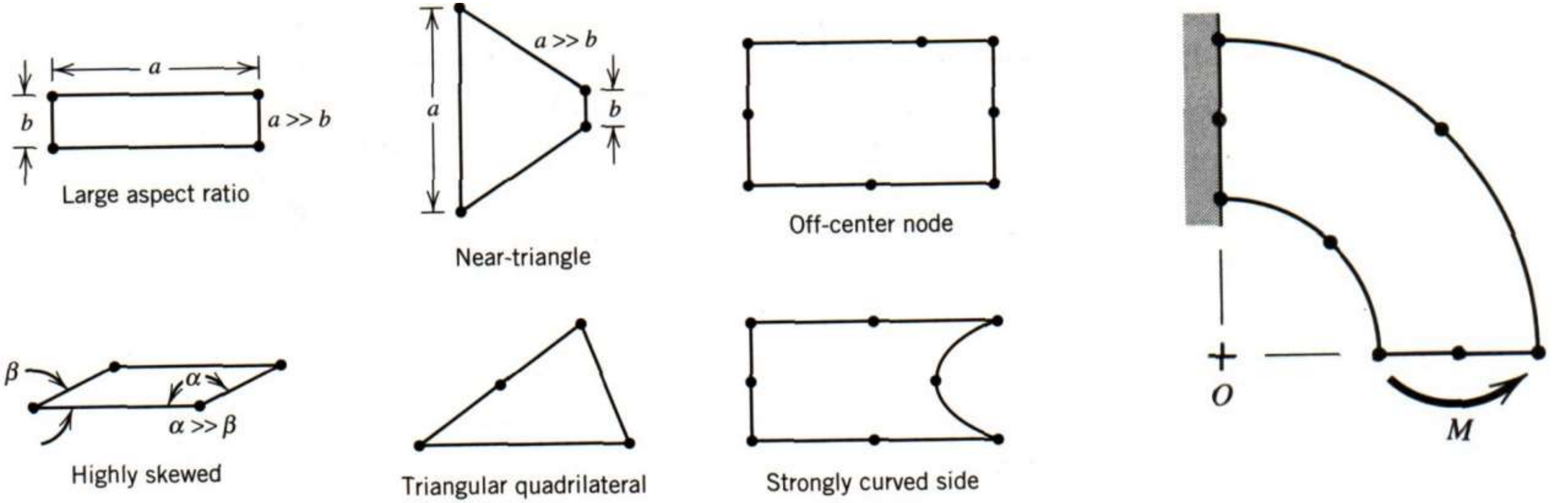
# Element Tests (1)

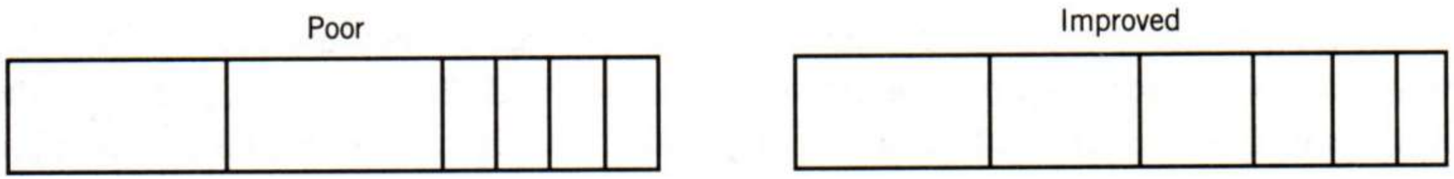
- Use a patch test or single element test to determine how an element works under different circumstances.
- Study different states of stress and strain.



# Element Tests (2)

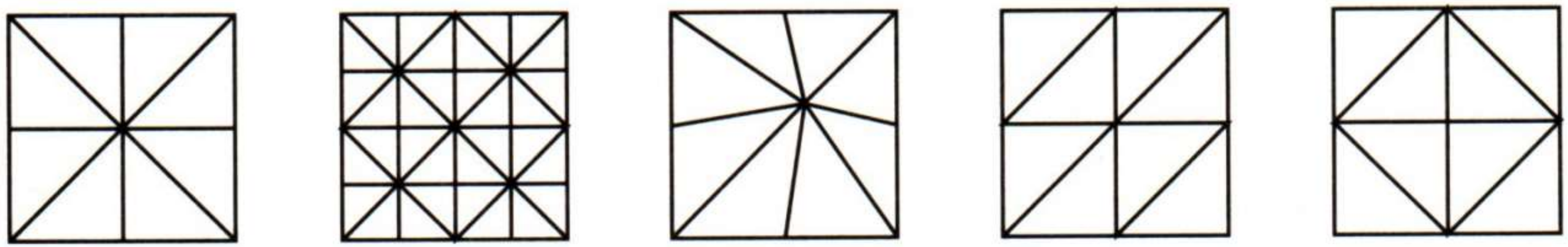
- Study the effects of element distortions and changes in element size.



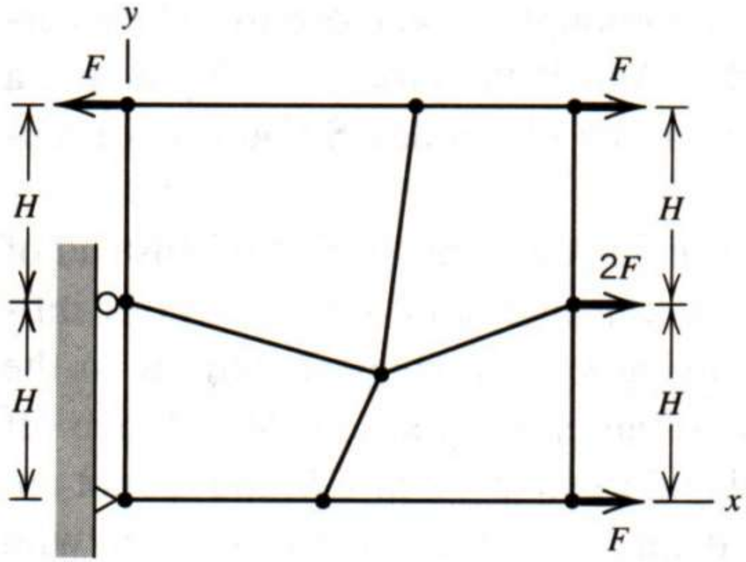
# Test Cases

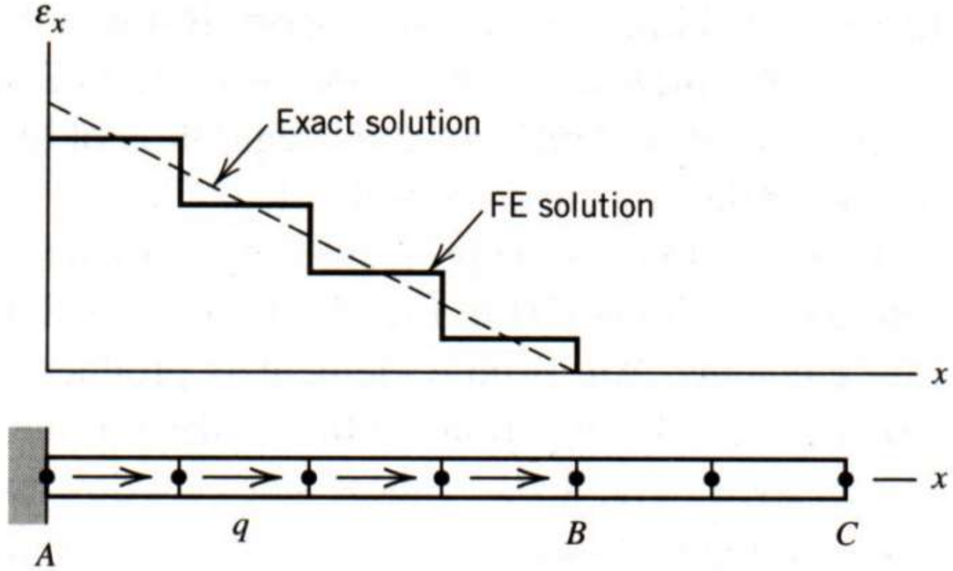
- Established test cases from:
  - research literature
  - National Agency for Finite Element Methods and Standards
  - software documentation
- can be used to check the accuracy of elements and models.
- “Pilot studies” can be used to check software capabilities.



# Discretization Error

- If a mesh is repeatedly refined, will the results converge to a solution?
- Yes, if the elements used pass the “patch” test.



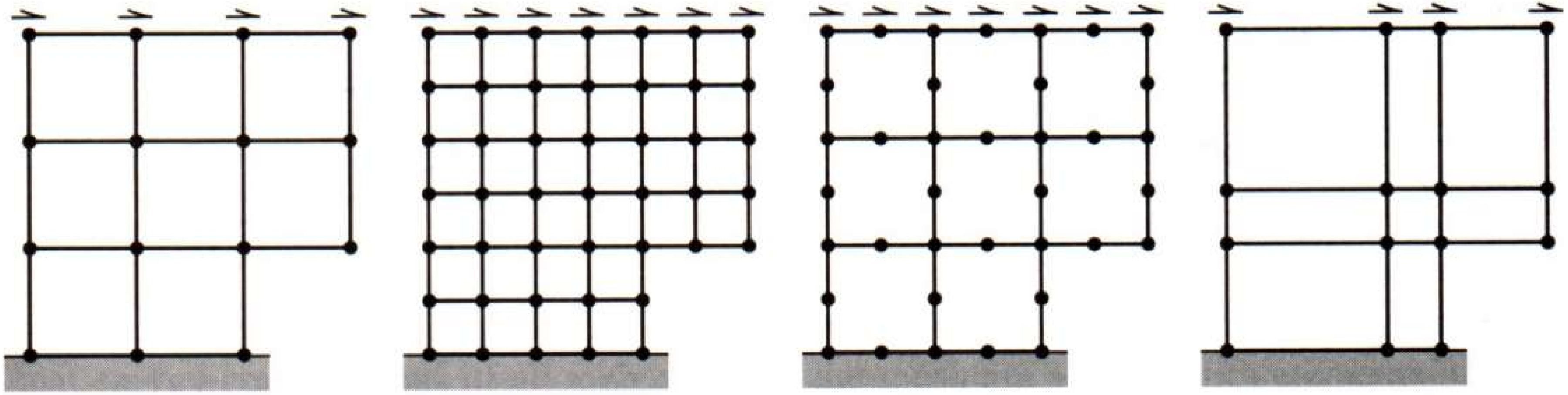



# Convergence Requirements

- In a patch test, the FE model must have:
  - A simple arrangement of elements with one internal node
  - Supports sufficient to stop rigid body motion
  - Work equivalent loads consistent with a constant state of stress (and strain)
- To pass the test, the results must exactly represent the correct constant stress (and strain), within numerical error.

# Mesh Refinement (1)

- There are three ways to refine a mesh:
  - h-refinement (changing the element size)
  - p-refinement (changing to elements with higher order polynomial interpolations)
  - r-refinement (moving nodes)



Original mesh

*A* uniform *h*-refinement

*A* uniform *p*-refinement

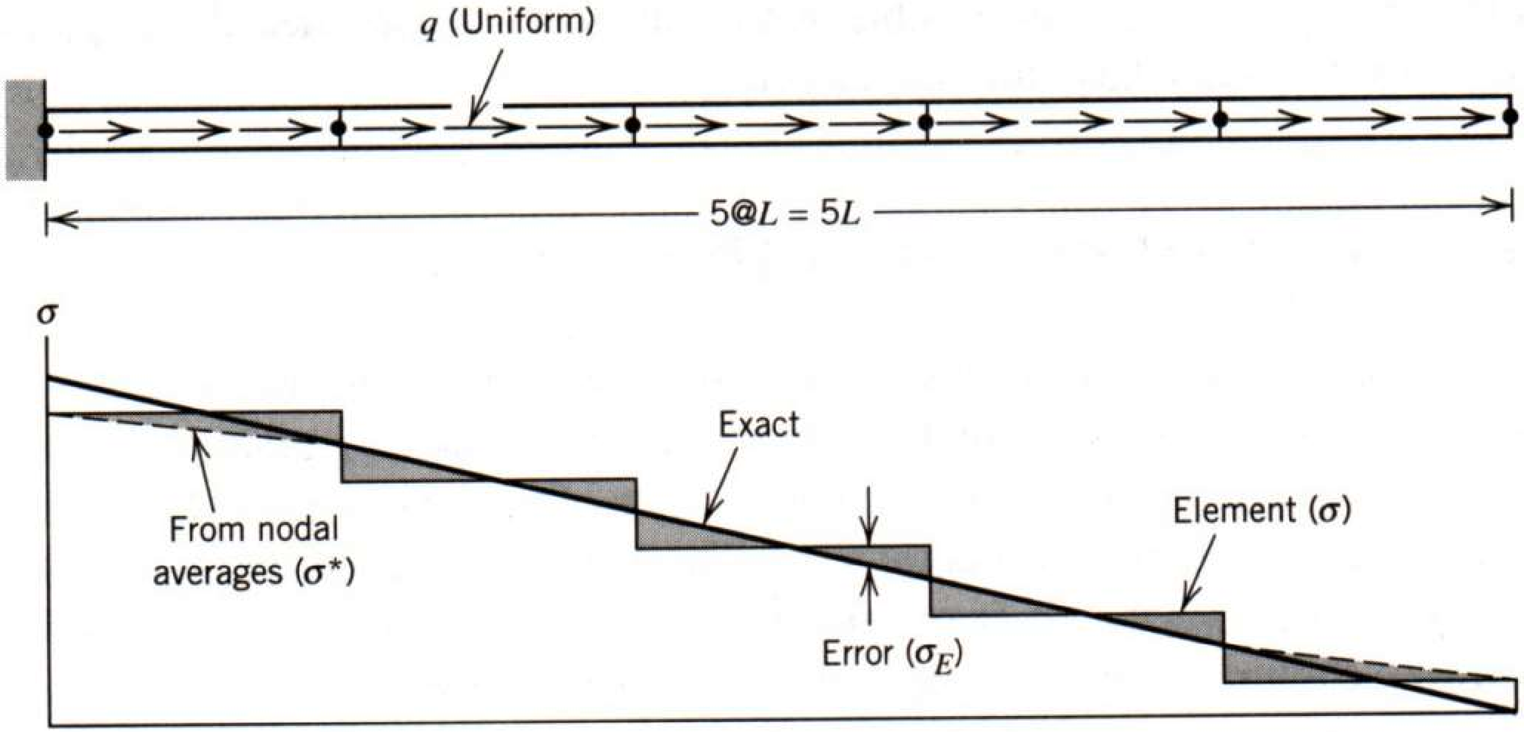
*A* possible *r*-refinement

# Mesh Refinement (2)

- A combination of these methods can also be used.
- The mesh should be refined until convergence is achieved (i.e., the results change very little from the previous refinement).
- Some software automates the refinement process (adaptive meshing)

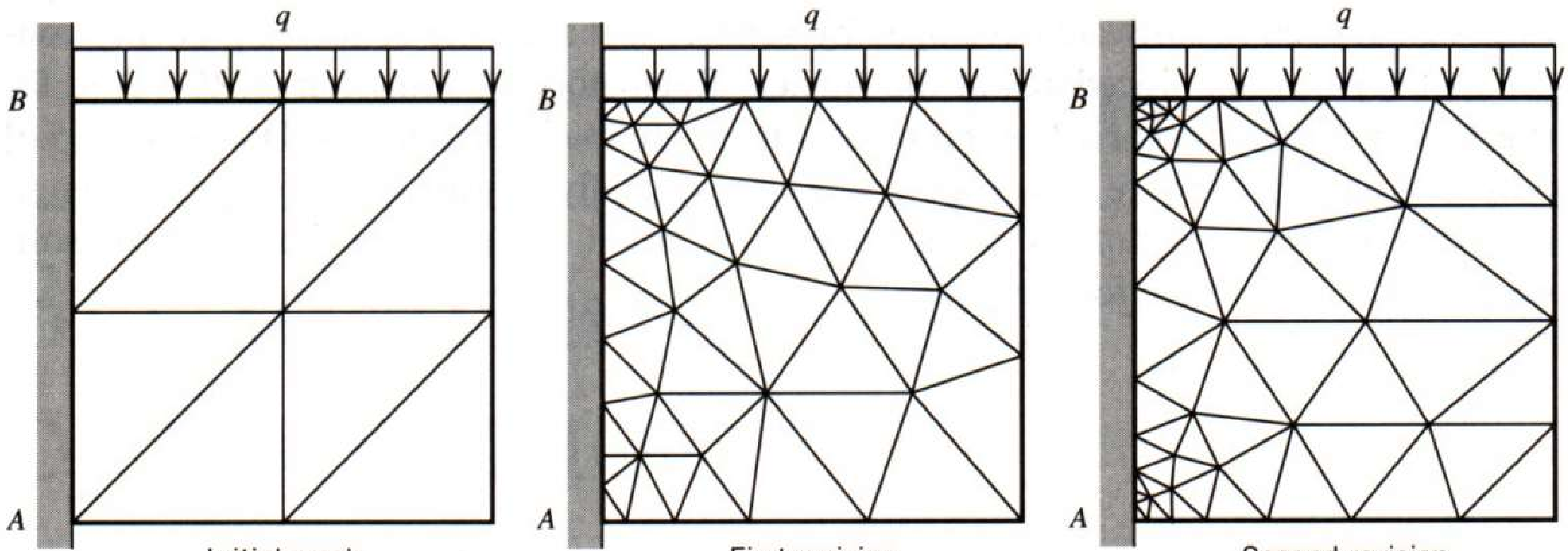
# Error Measures

- One approach to error estimation is to assume that the nodal averaged stress ($\sigma^*$ ) is correct and the error ($\sigma_E$ ) is given at every point by the difference from the element stress ($\sigma$).



# Error Measures and Adaptivity

- An Automated Adaptive Solution proceeds by refining the mesh, in elements where the error is large, until the maximum error is below some limit.




Initial mesh
40 d.o.f., $\eta = 0.27$

First revision
228 d.o.f., $\eta = 0.07$

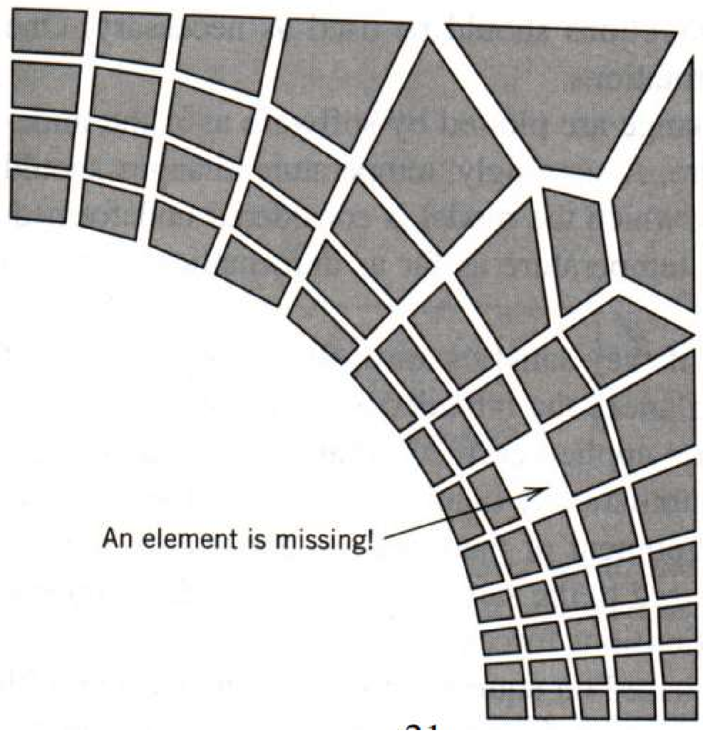
Second revision
286 d.o.f., $\eta = 0.04$

# Checking the Model (before solving)

- Checking done automatically by software
  - Model has mesh and boundary conditions are applied.
  - All mesh and boundary condition properties have been provided.
  - Element aspect ratios and corner angles too small or too large.
  - Element is too warped.
  - Poisson's ratio too large.
  - Curved shell element spans too great an arc.
- Specific checks that can be requested
  - Coincident nodes (Are they supposed to be one node?)

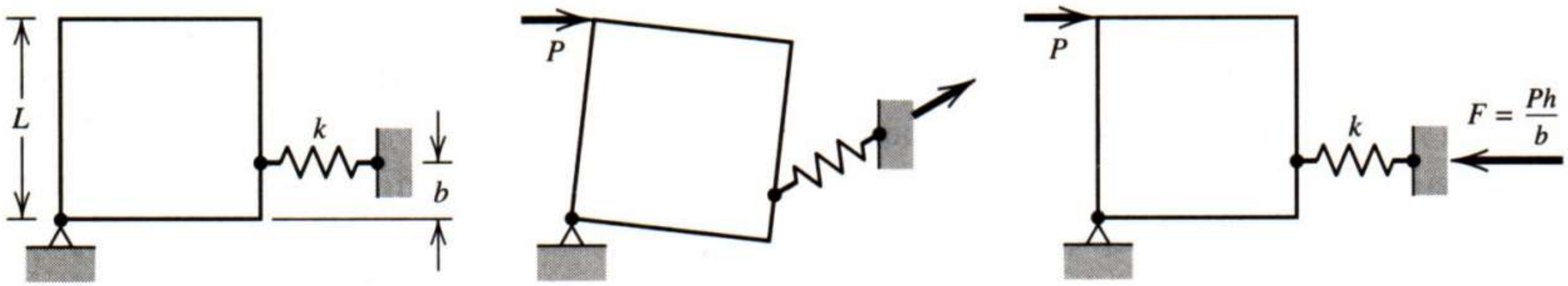
# Checking the Model (before solving)

- Checking done by Analyst
  - Everything meshed properly?
  - All required loads/support conditions applied?
  - Double-check material/shell/beam properties



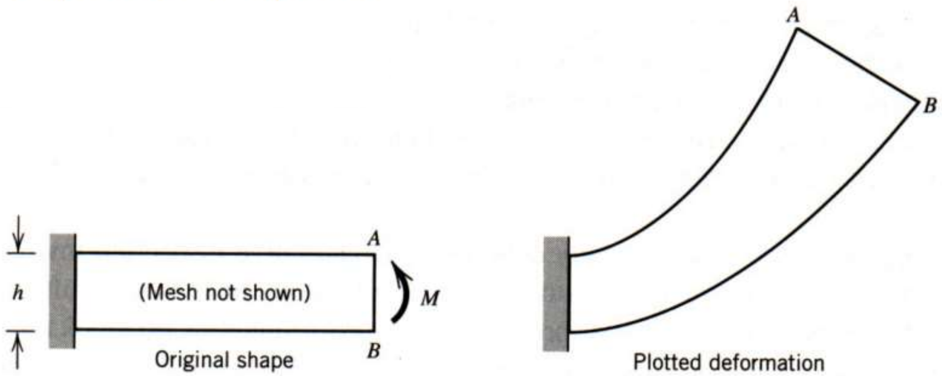
# Checking the Results (after solving)

- Results should be checked so that:
  - Deflections obey intended support conditions.
  - Deflections are symmetric in a symmetric problem.
  - Where a gap closes the parts do not overlap.
  - Support reactions agree with static calculations.
  - There are no large displacements that cause force directions to change (use a nonlinear analysis in this case).



# Checking the Results

- Note that deflections may only appear large because of the exaggeration factor used to display the deflections.




- Stresses should be checked that:
  - Stress contours are normal to planes of symmetry
  - One of the principle stresses should be zero at an unloaded boundary or equal to –p if there is a pressure p loading condition.

# Checking the Results

- When checking stresses note that:
  - Unaveraged stresses should be checked.
  - Confirm that the displayed stress is the one you want to look at (i.e., principle stress vs. shear stress vs. von Mises stress, etc.)
  - Stresses may be in local or global coordinates
  - Stresses may be for the upper, lower or midsurface of beam and shell elements.