Modeling Errors and Accuracy

And checking the results!

Sources of Error

Causes of incorrect results:

1. Mistakes (e.g., forgetting a load)
2. Errors
   A. 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)
   B. Discretization Error (i.e., due to piecewise approximation which can be minimized by using higher order shape functions or smaller elements)
   C. Numerical Error (due to limited number of significant digits maintained by computer)

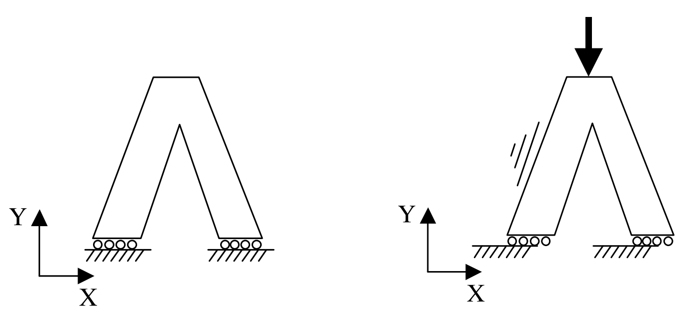
Institute of Technology West Virginia University

1. Mistakes

- Common mistakes that will cause a singular **K** matrix (and therefore no results):
  - **ν = 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.



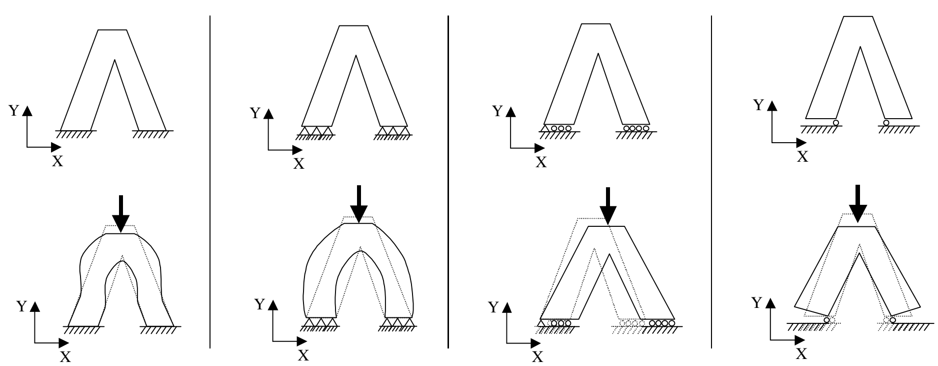
(The stiffness matrix will be singular.)

Common Mistakes

- Mistakes that may go unnoticed:
  - 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)

Common Mistakes

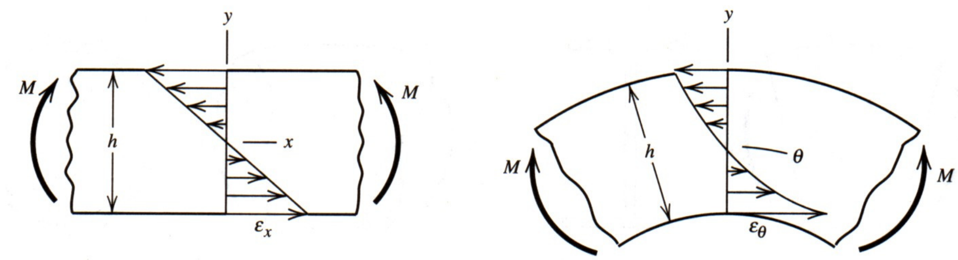
- Effect of wrong support types:



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

2. A. Modeling Error

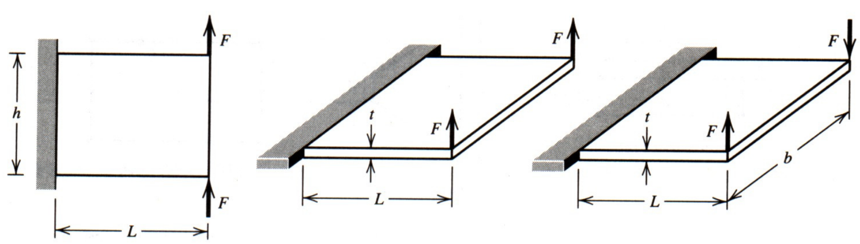
- To do a proper FE analysis, the analyst must understand how the structure is *likely* to behave and how elements are *able* to behave.
- E.g., if the analyst knows the displacement varies linearly, 4-node quad. elements will work, but if they vary quadratically, 8-node quad. elements must be used.



Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

Element Tests

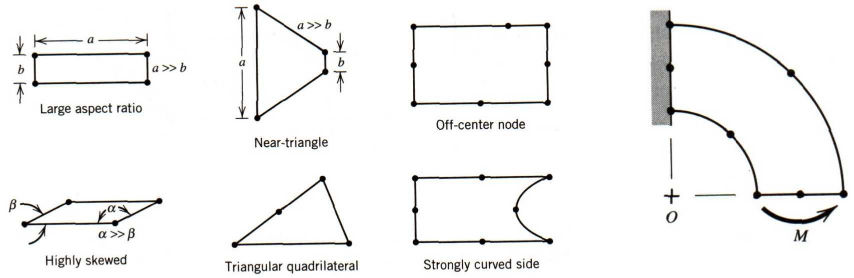
- Use a *patch* test or *single element* test to determine how an element works under different circumstances.
- Study different states of stress and strain.

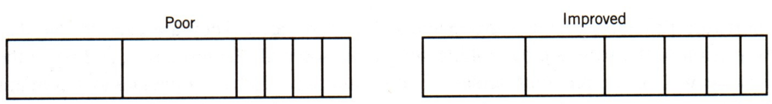


Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

Element Tests

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



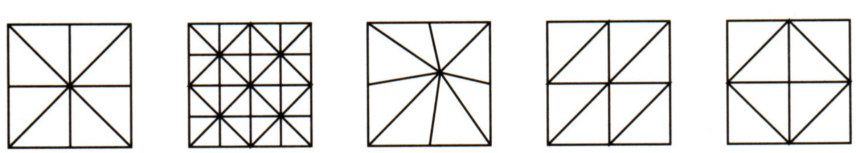


Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

Test Cases

- Established test cases from:
  - research literature
  - National Agency for Finite Element Methods and Standards
  - software documentation

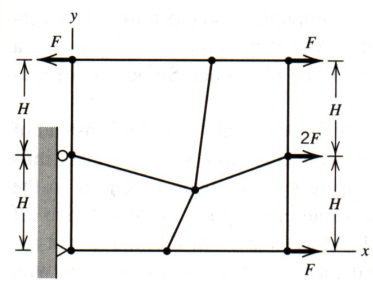
  can be used to check the accuracy of elements and models.
- "Pilot studies" can be used to check software capabilities.

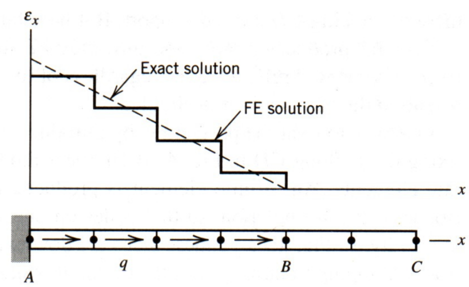


Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

2. B. Discretization Error

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





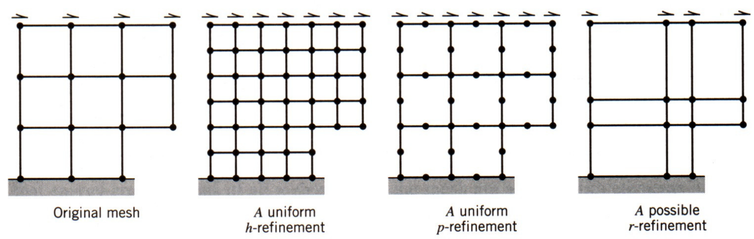
Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

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

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



Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

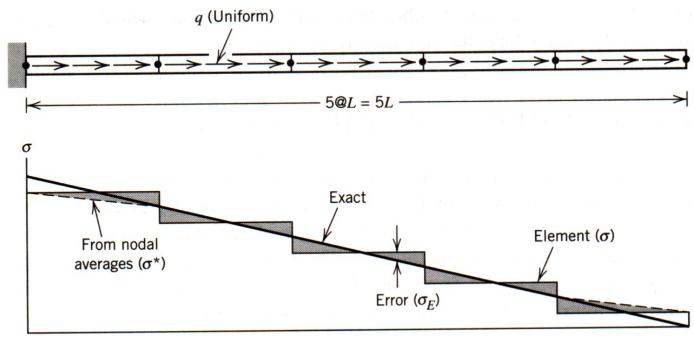
Mesh Refinement

- 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**).

Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

Error Measures

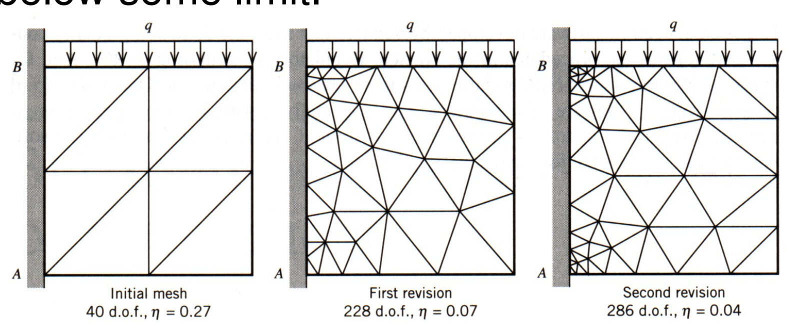
One approach to error estimation is to assume that the nodal averaged stress (σ^*) is correct and the error (σ_E) is given at every point by the difference from the element stress (σ).



Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

Error Measures and Adaptivity

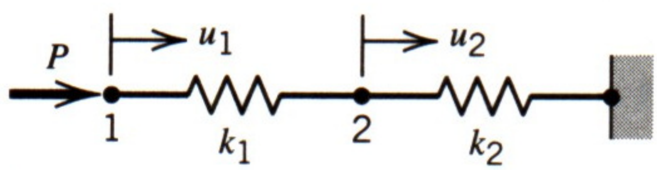
- An Automated Adaptive Solution proceeds by refining the mesh, in elements where the error is large, until the maximum error is below some limit.



Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

2. C. Numerical Error

- Rounding errors will accumulate (more so in large DOF models)
- Adding very small numbers to big numbers is even more problematic (i.e. in "**Stiff Systems**")



$$\mathbf{KD} = \mathbf{R} \quad \text{is} \quad \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 + k_2 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} P \\ 0 \end{Bmatrix}$$

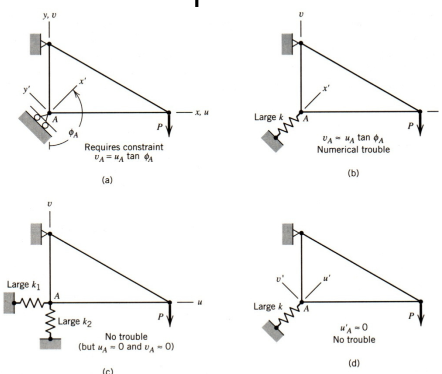
Numerical Error – "Stiff" Systems

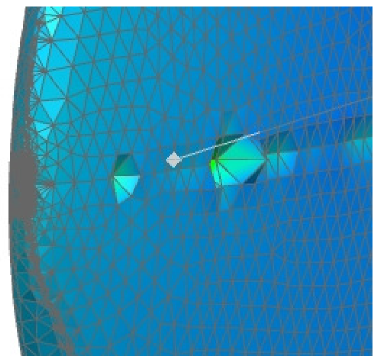
- If $k_1 >> k_2$,

- If $k_2 >> k_1$,

Susceptibility to Ill-Conditioning

- Large cross-coupling stiffness coefficients will cause problems
- Having membrane stiffness ≫ bending stiffness will cause problems.





Institute of Technology West Virginia University

3. 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?)

Institute of Technology West Virginia University

Checking the Model (before solving)

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

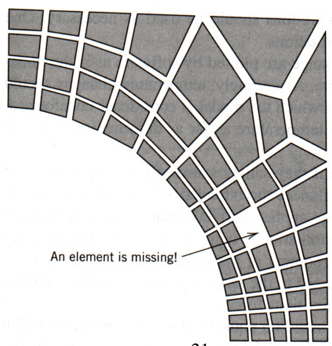
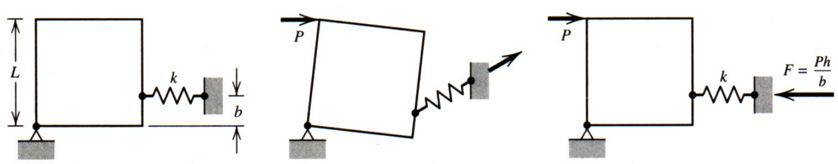


Image from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

Checking the Results (after solving)

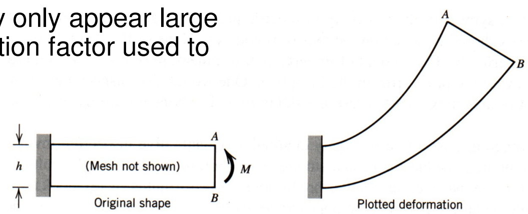
- Results should be checked so that:
  - **Deflections** obey intended **support conditions**.
  - Deflections are symmetric in a symmetric problem.
  - Where a gap closes the parts **do not overlap**.
  - **Support reactions** agree with static calculations.
  - There are no large displacements that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



Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

Checking the Results

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



Images from R. Cook, et al. *Concepts and Applications of Finite Element Analysis*, 1996.

- 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 mid-surface** of beam and shell elements.