Modeling

- A model is a symbolic representation of the real thing (nature).
- A model could be experimental, analytical, or numerical or some combination of the three.
- Solutions of all models are approximations, whether an approximate solution is an acceptable is a decision that is based on additional information, such as: correlation with results from other models, experience, intuition, usefulness.

Errors in FEM

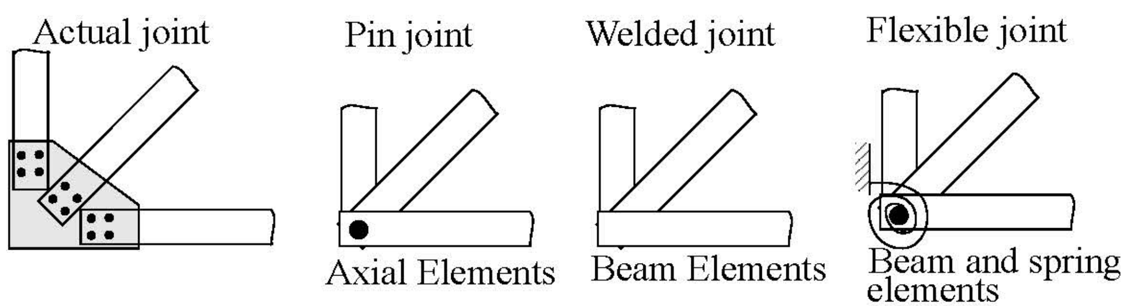
1. Modeling error
2. Discretization error
3. Numerical error

Modeling error

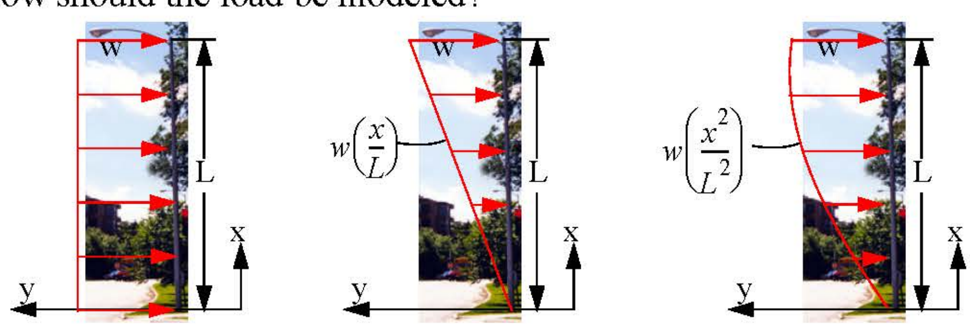
Error that arise from the description of the boundary value problem (BVP): Geometric description, material description, loading, boundary conditions, type of analysis.

- What physical details are important in the BVP description?

  Should a mechanically fastened joint be modeled as a pin joint, welded joint, or a flexible joint.



How should the load be modeled?

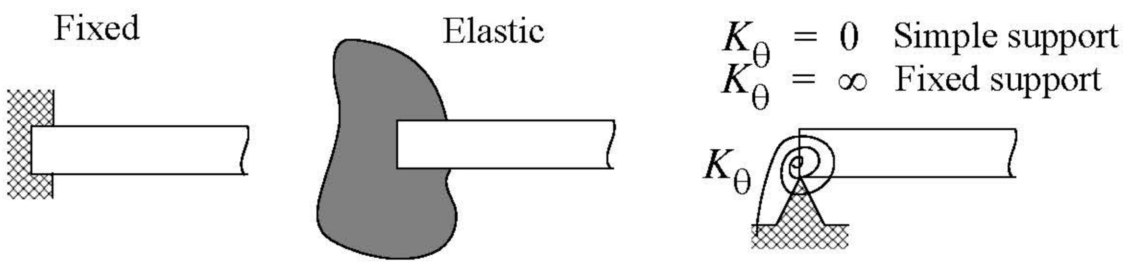


Should the properties of the adhesive be included or ignored in a bonded joint?

Should the material be modeled as isotropic or orthotropic? The coefficients of the differential equation depends upon the coordinate system orientation.

Should the material be modeled as homogeneous or non-homogeneous? The coefficients of the differential equation depends upon coordinates.

How should the support be modeled? i.e., what are the appropriate boundary conditions.



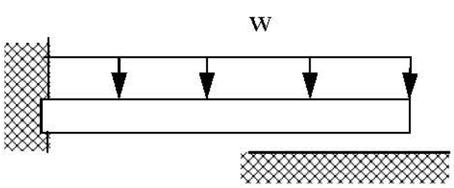
- ## What type of analysis should be conducted?

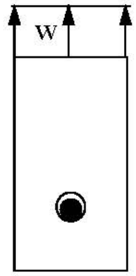
Should you conduct a linear or non-linear analysis?

1. Material non-linearity. The coefficient of differential equations depend upon the forcing function. [Stress and strain are non-linearly related and modulus of elasticity is replaced by tangent modulus that depends upon the load value].
2. Geometric non-linearity. The differential equation is non-linear in u. [Strain and displacement non-linearly related—large deformation or strain]
3. Contact problem. The boundary conditions change with the forcing functions. [The contact length changes with load.

   (i) No friction.

(ii) With friction—need the slip ($F_f = \mu N$) and no slip boundary($F_f < \mu N$).





Should stability analysis be conducted? [Should buckling analysis be conducted?] Stability analysis is related to the second variation of the functional and there are many ways to conduct it.

For time dependent problems should you conduct a dynamic or quasi-static analysis? Should the material be modeled as elastic or viscoelastic?

Discretization errors

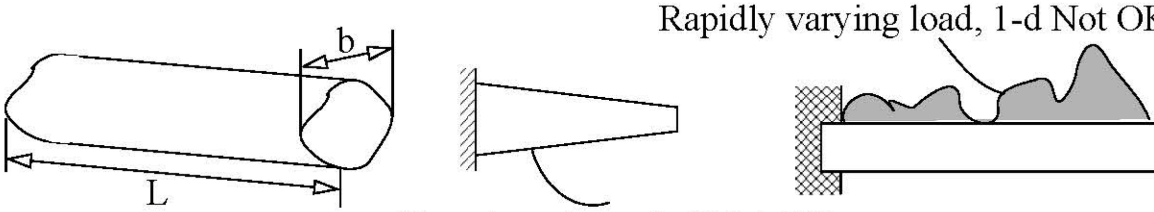
- Errors that arises from creation of the mesh.

Elements in FEM are based on analytical models. All assumptions that are made in the analytical models are applicable to FEM elements.

- **What type of elements should be used?**

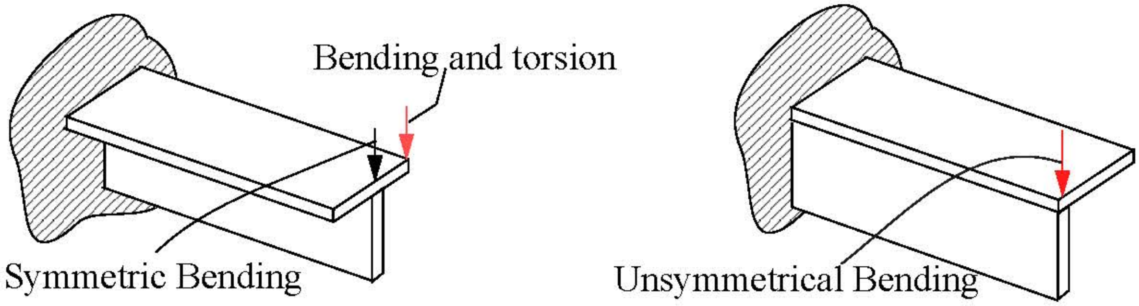
Should 1-d element be used?

Rapidly varying load, 1-d Not OK



$L \gg b$—1-d OK   Very steep taper, 1-d Not OK

Should beam element, which is based on symmetric bending, be used?



- What type of 2-d (plane stress, plane strain) or 3-d element should you use.?
- What mesh density should you use?

  Too fine a mesh results in large computer time that may prevent optimization or parametric studies or non-linear analysis. Too coarse a mesh may result in high inaccuracies. Start with a coarse mesh, study the results and then refine the mesh as needed.
- How accurately should the geometry be modeled?

  Errors from modeling of geometric are generally small. For the same computational effort higher returns in accuracy are obtained in better modeling of displacement--Isoparametric elements are adequate.

Numerical Errors

Errors that arise from finite digit arithmetic and use of numerical methods.

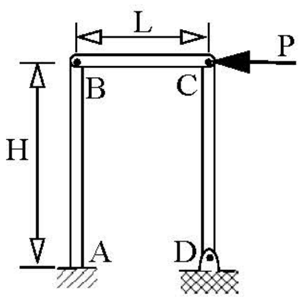
- Integration error

  Few Gauss points leads to numerical instabilities. Large number of Gauss points are computationally expensive and may result in overly stiff elements leading to higher errors.
- Round off error

  The finite digit arithmetic causes these errors, but the growth of round off errors are dictated by several factors. Need to avoid: adding or subtracting very large and very small numbers; dividing by small numbers.

  (i) The manner in which algorithms are written in the computer codes. Non-dimensionalizing the problem will always help.[See Buckingham pi (π) theorem]

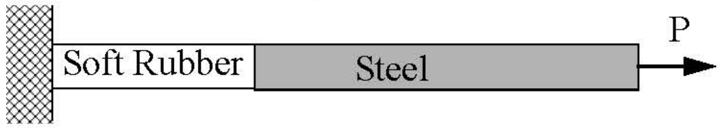
  (ii) Large differences in physical dimensions.



$\frac{H}{L} \to \infty Large \qquad K_{BC} \gg K_{AB} \qquad K_{BC} \gg K_{CD}$

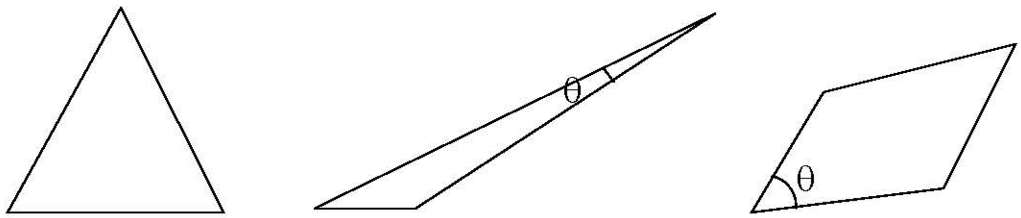
Can BC be modeled as rigid?

(iii) Large differences in stiffness caused my large differences in material properties (or dimensions).



Can steel be modeled as rigid?

(iv) Elements with poor aspect ratio: ratio of largest to smallest dimension in an element. The interior angle θ should not be too large or too small, otherwise The determinant of Jacobian tends towards zero, which leads to poor matrix conditioning.



Good Aspect Ratio Poor Aspect Ratio

- **Matrix conditioning error**

  The coefficients in the matrix of the algebraic equations are affected by the above described numerical errors and it is in the solution of the algebraic equations one realizes something has gone wrong.

Example of poor conditioning:

$$\begin{bmatrix} 1 & -1 \\ -1 & 1.001 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} 2 \\ -3 \end{Bmatrix} \qquad \Rightarrow \qquad \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} -998 \\ -1000 \end{Bmatrix}$$

$$\begin{bmatrix} 1 & -0.999 \\ -0.999 & 1.001 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} 2 \\ -3 \end{Bmatrix} \qquad \Rightarrow \qquad \begin{Bmatrix} u_1 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} -331.777 \\ -334.11 \end{Bmatrix}$$

- Small changes produce large swings in the solution. Thus, a solution may not be correct.
- Matrix conditioning is a measure of diagonal dominance in a matrix.
- Poorly conditioned matrices have a determinant that tends towards zero.

Mesh Refinement

- Regions of large gradients in secondary variables require finer mesh.
- Elements with large jumps in secondary variables across the boundary require finer mesh. The jump may be due to physical reasons such as sudden changes in the material property (interface), geometry, or loading.

- Elements with high strain energy identify the region of the body where mesh should be refined.

  The *h-method* of mesh refinement reduces the size of element.

  The *p-method* of mesh refinement increases the order of polynomial in an element.

  The *r-method* of mesh refinement relocates the position of a node.

  Combinations: hr-method, hp-method, hpr-method

- Generally speaking: the p-method of mesh refinement works well for regions where stresses vary slowly; the hr- method works better suited for regions of large stress gradients.

- Error norms

To develop any mesh refinement scheme one has to have a measure of the error without knowing what should be the actual solution. These measures of errors are called error norms. u(x) is the primary variable and p(x) is the polynomial approximation. u(x) is estimated from the numerical solution.

L_1 norm in 1-d: $e_R = \sum_{i=1}^{N} \int_{x_{i-1}}^{x_i} |u(x) - p(x)| dx$

L_2 norm in 1-d: $e_R = \left[\sum_{i=1}^{N} \int_{x_{i-1}}^{x_i} |u(x) - p(x)|^2 dx\right]^{\frac{1}{2}}$

Energy norms: $e_R = \sum_{e} [I^{(e)}(u) - I^{(e)}(p)]$

The objective of a mesh refinement algorithm is to reduce the error norm in each iteration to meet the user specified value. In solid mechanics and structure if the strain energy is uniform in each element then you have an optimum mesh.

Conclusions

1. There are many reasons for a FEM program to give errors and not work.
2. There are many reasons for a FEM program to work but give wrong results.
3. Don’t blame the software, it is your responsibility to ascertain if you have the right results or not.

Good FEM Practices

1. Do not start by creating a mesh without studying the problem you need to analyze. Can you identify regions where the secondary variable will have gradients or discontinuity? What trends are you expecting in behavior of the primary and secondary variables?
2. Non-dimensionalize your problem. Use Buckingham Pi Theorem to identify the non-dimensionalize groups if you are going to do parametric study or optimization.
3. Create a way of testing your FEM model. A simple test is to assume the primary variables vary linearly. Determine the constants to ensure continuity at transition points. Calculate expressions for secondary variables. Use the analytical solution to calculate the boundary conditions. These conditions will be used to simulate your test on FEM model.
4. Create a crude mesh simulate it with the analytical values and see if the mesh gives good results.
5. Run the crude mesh model with actual loads and boundary conditions. Are the results consistent with what you expected in step 1? Identify the regions of interest. Substructure the problem for purpose of creating meshes of different densities.
6. Create finer mesh and see if your results converge. If your results vary too dramatically then you have a matrix conditioning problem. See what might be causing it.