

Introduction to FINITE ELEMENT METHOD

1-1 Definition

Finite element method is a numerical method that can be used for solving engineering problems.

It is particularly useful for problems involving complex geometries, combined loading and material properties, in which the analytical solutions are not available.

FAQ: Is this method accurate?... Yes, for simple problems and if material properties and loading condition are modeled very close to the actual conditions. No, especially when the problems to be solved are too complex.

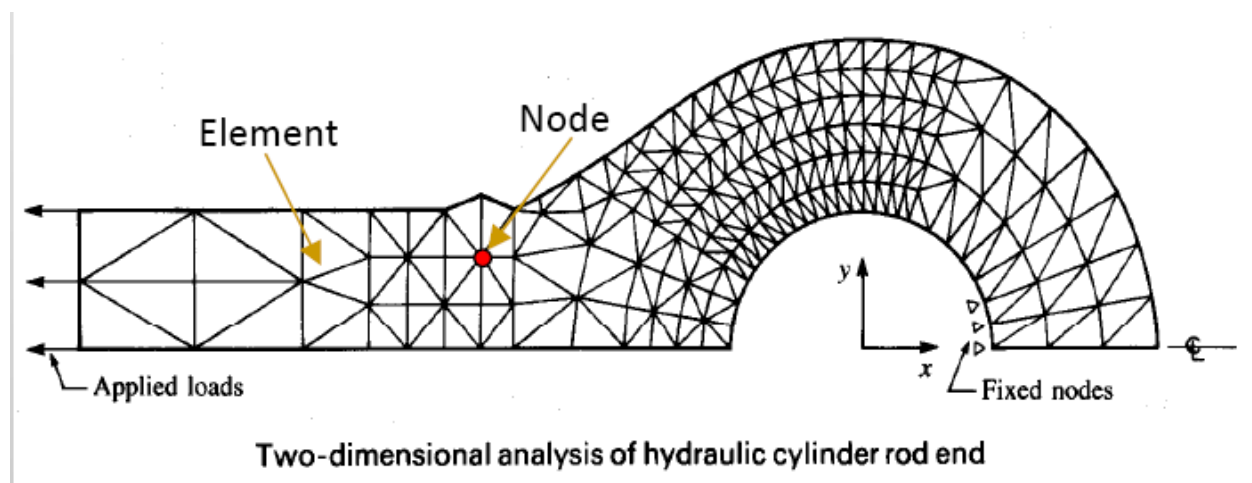
Among the sources of error involved in this method are...

- Geometrical simplification;
- Interpolation function;
- Material properties;
- Round-off, etc.

1-2 Elements and Nodes

The physical body (continuum) is modeled by dividing it into an equivalent assembly of smaller bodies or units, called the **finite elements**.

Elements are interconnected at points common to two or more elements or boundary lines or surfaces, called **nodes** or **nodal points**.



***Elements and nodes are important fundamentals to the finite element method

1-3 Areas of Application

Some application areas of finite element method includes:

- Structural – Static Stress Analysis, Dynamics;
- Heat Transfer – Steady-state, Transient;
- Electromagnetic;
- Fluid Flow;
- Soil Mechanics;
- Acoustics, and
- Many more...

1-4 Advantages of FEM

Advantages of finite element method include:

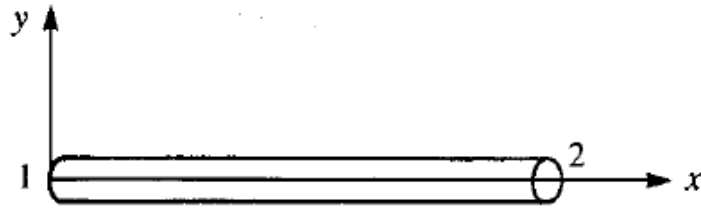
- Handle bodies with complex geometries;
- Accept general loading conditions;
- Bodies different material properties;
- Various support conditions;
- Variable element type and size;
- Easy modification – reanalyze;
- Dynamics – Vibration; Shock loading – drop, crash;
- Nonlinear problems (geometric & material).

1-5 Solution Steps

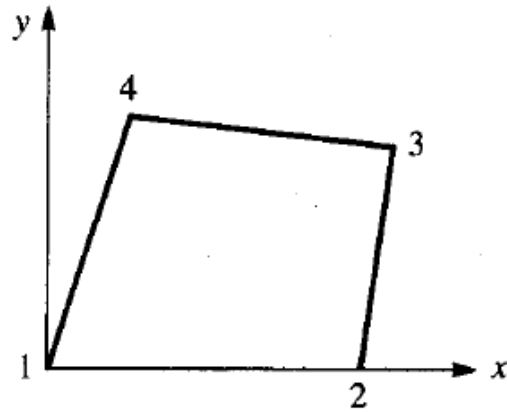
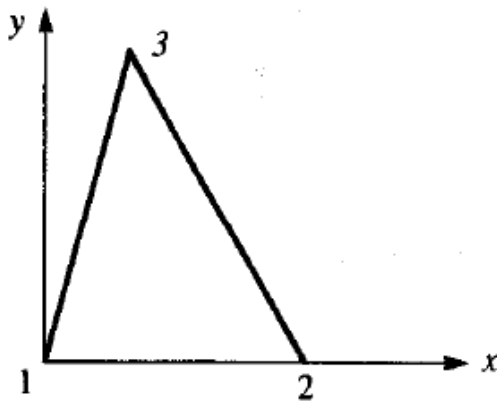
Finite element solution generally follows the following steps:

1. Discretize (subdivide) body and select element type;
2. Select a displacement interpolation function;
3. Define strain-displacement and stress-strain relations;
4. Derive element stiffness matrix and write the system of linear equations (SLEs) for the element;
5. Assemble the global system of equations and introduce the boundary conditions;
6. Solve for the **unknown** nodal values (displacements, temperatures, etc);
7. Evaluate element stresses and reaction at supports;
8. Interpret the results.

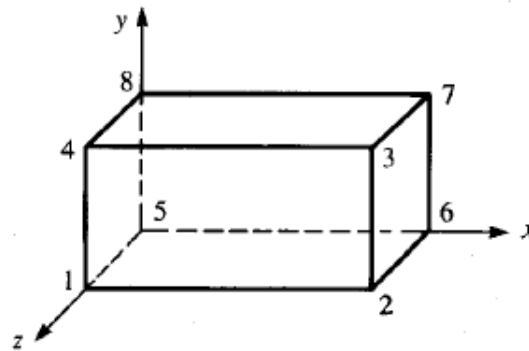
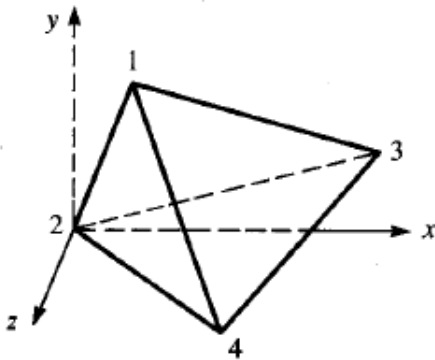
1-6 Types of Elements



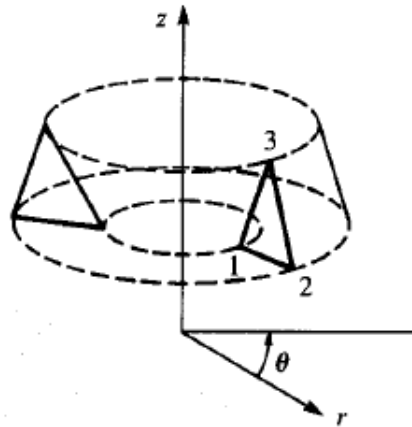
(a) Simple line element



(b) Simple two-dimensional elements

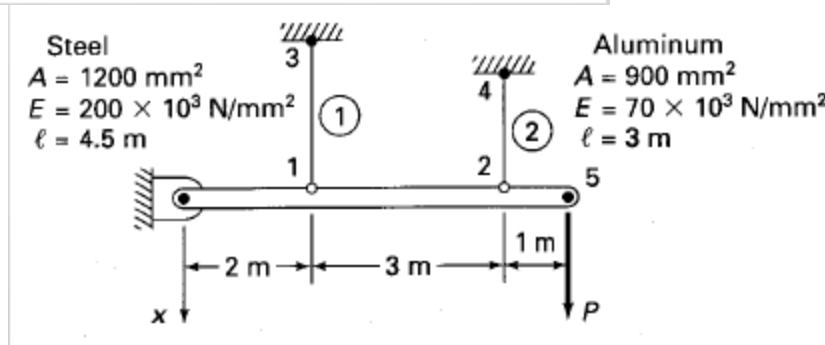
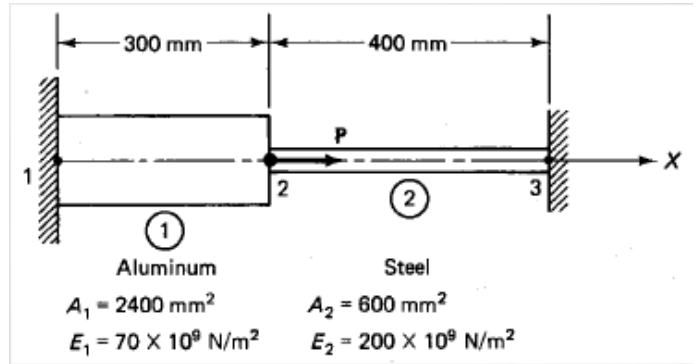
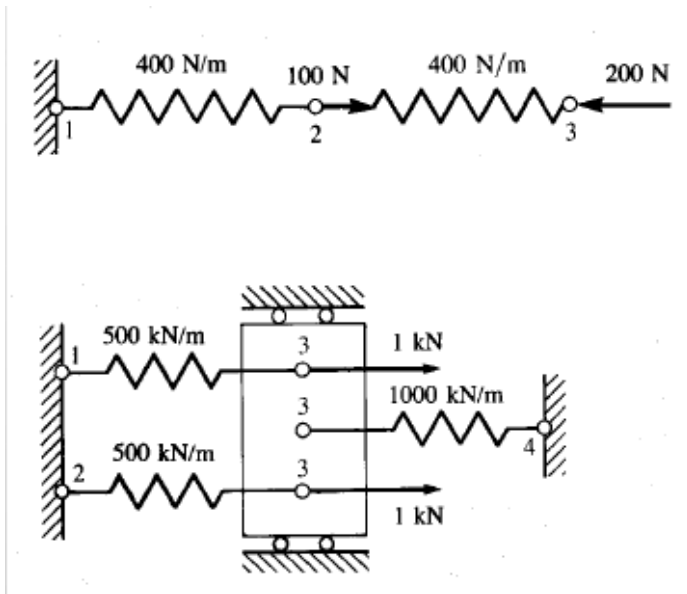


(c) Simple three-dimensional elements

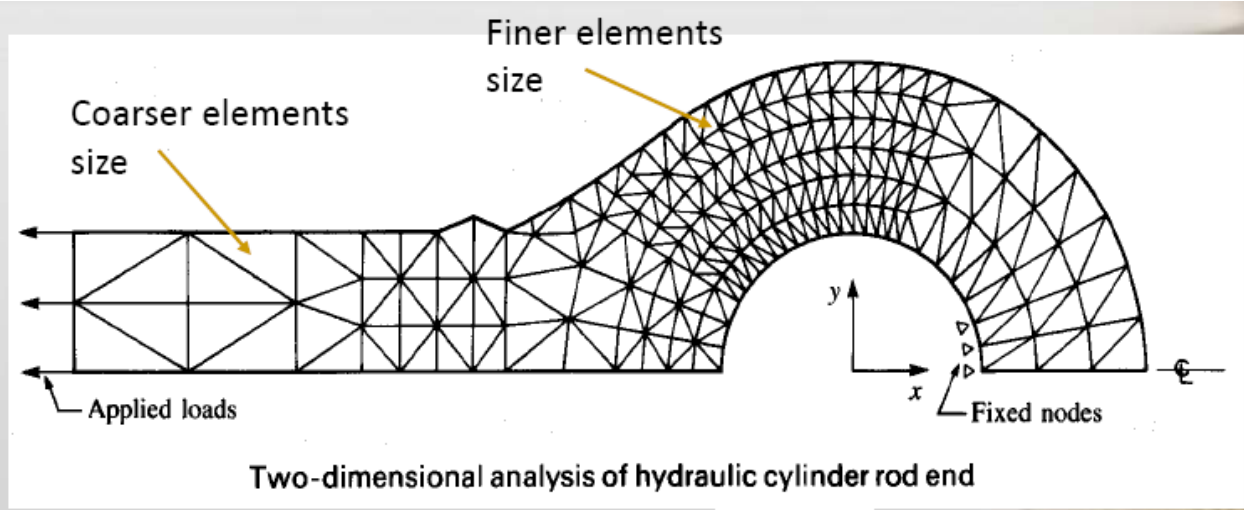
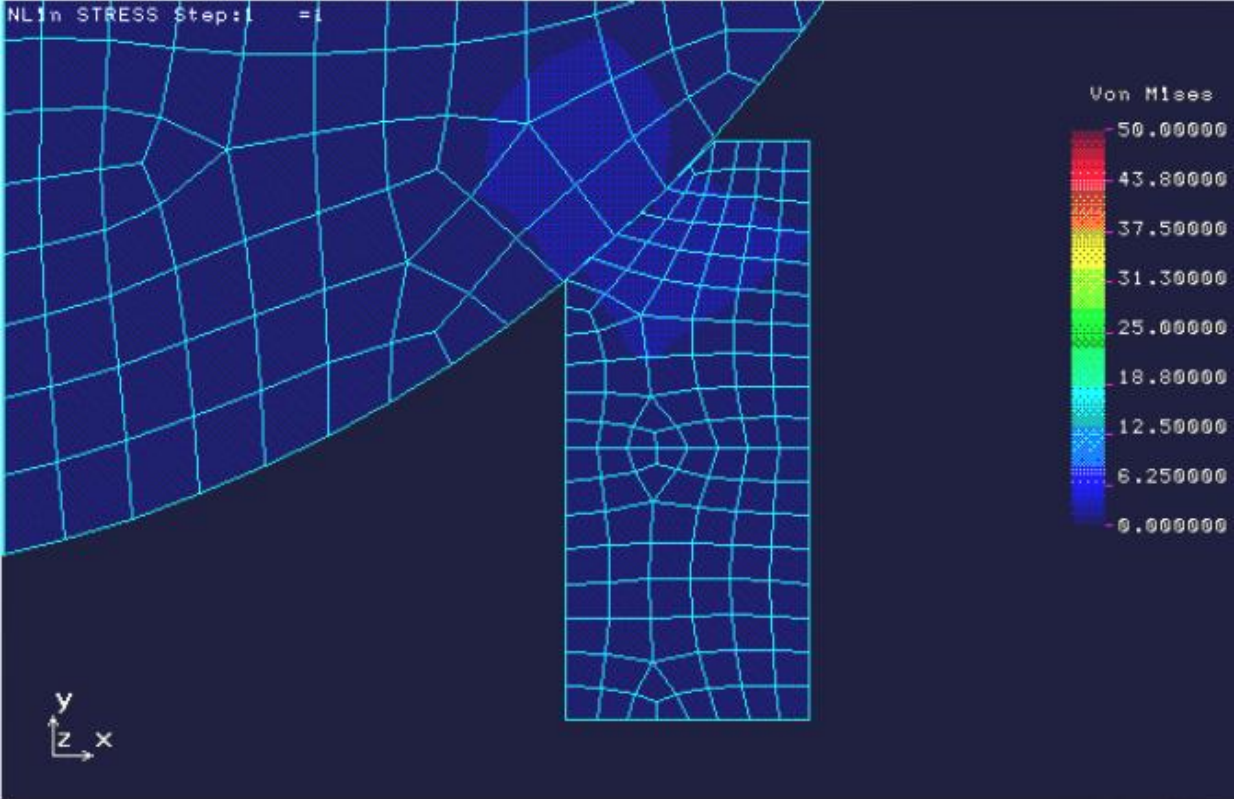


(d) Simple axisymmetric element

1-7 Examples of 1-D Problems



Examples of 2-D Problems



A hydraulic cylinder rod end is modeled using 120 2-D plane-strain triangular elements, with 297 nodes. Symmetry condition is applied to the whole rod end so that only half of the rod end needs to be modeled and analyzed.

1-9 Examples of 3-D Problems

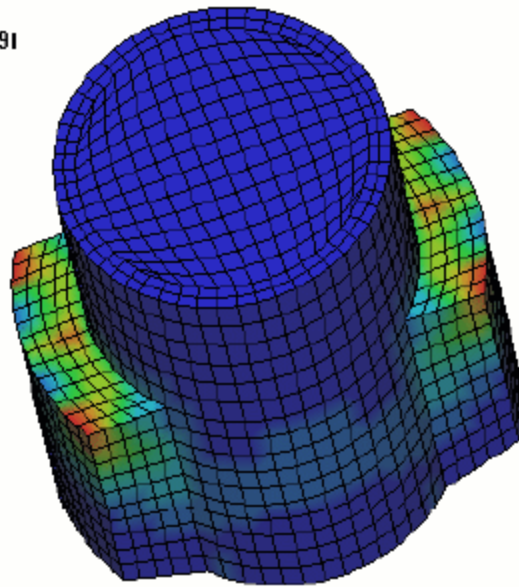
BAT 8.5" ACTION & BOLT 60 KSI CHAMBER P

Time = 1

Contours of Effective Stress (v-m)

min=464.554, at elem# 695

max=181504, at elem# 11691



Fringe Levels

1.815e+005

1.634e+005

1.453e+005

1.272e+005

1.091e+005

9.098e+004

7.288e+004

5.478e+004

3.667e+004

1.857e+004

4.646e+002

