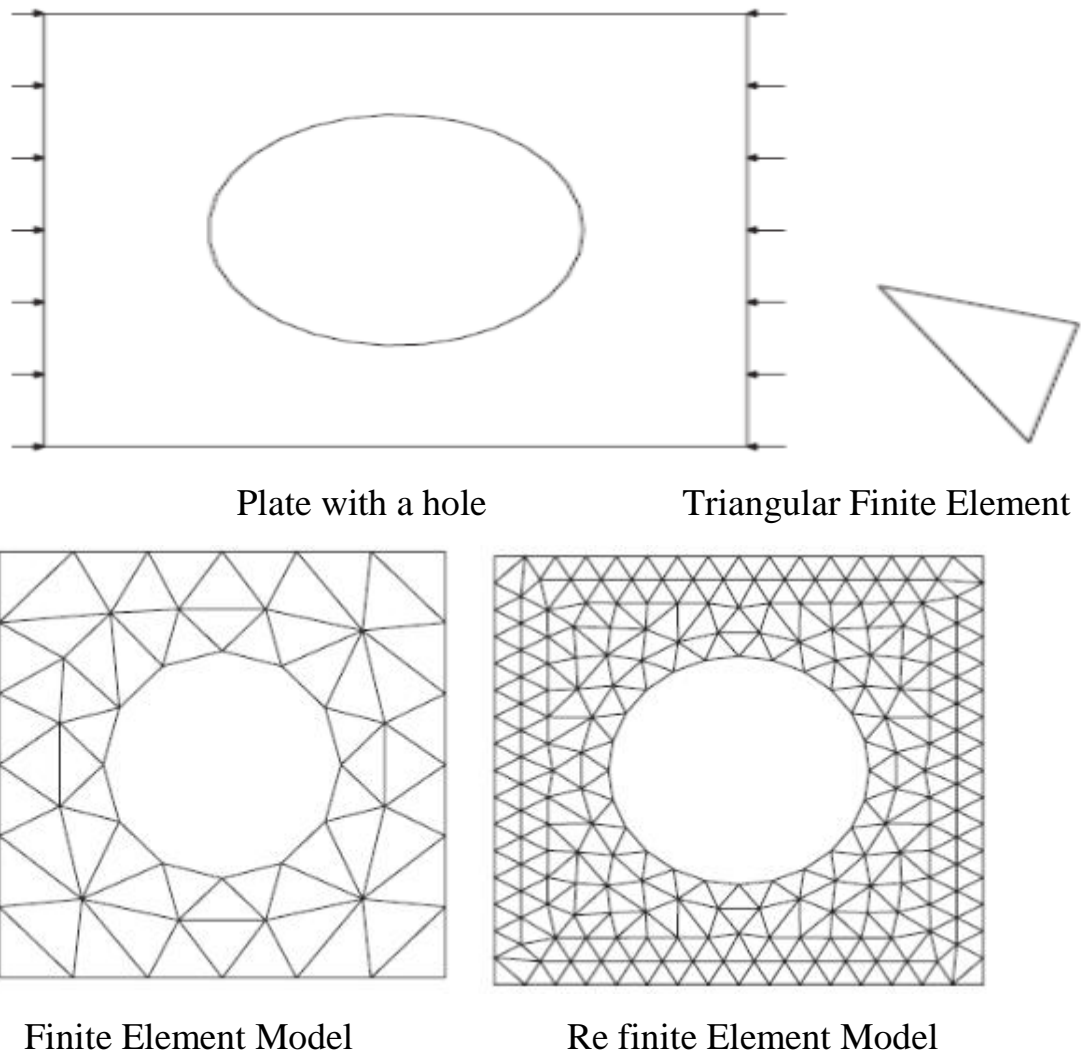


## **Literature Review**

### **2.1 General beams:**

Many physical phenomena in engineering and science can be described in terms of partial differential equations. In general, solving these equations by classical analytical methods for arbitrary shapes is almost impossible. The finite element method (FEM) is a numerical approach by which these partial differential equations can be solved approximately. From an engineering standpoint, the FEM is a method for solving engineering problems such as stress analysis, heat transfer, fluid flow and electromagnetics by computer simulation. Millions of engineers and scientists worldwide use the FEM to predict the behavior of structural, mechanical, thermal, electrical and chemical systems for both design and performance analyses. Its popularity can be gleaned by the fact that over \$1 billion is spent annually in the United States on FEM software and computer time. A 1991 bibliography (Noor, 1991) lists nearly 400 finite element books in English and other languages. A web search (in 2006) for the phrase ‘finite element’ using the Google search engine yielded over 14 million pages of results. Mackerle lists 578 finite element books published between 1967 and 2005. To explain the basic approach of the FEM, consider a plate with a hole as shown in Figure 2.1 for which we wish to find the temperature distribution. It is straightforward to write a heat balance equation for each point in the plate. However, the solution of the resulting partial differential equation for a complicated geometry, such as an engine block, is impossible by classical methods like separation of variables. Numerical methods such as finite difference methods are also quite awkward for arbitrary shapes; software developers have not marketed finite difference programs that can deal with the complicated geometries that are commonplace in engineering. Similarly, stress analysis requires the solution of partial differential equations that are very difficult to solve by analytical methods except for very simple shapes, such as rectangles, and engineering problems seldom have such simple shapes .[3]

The basic idea of FEM is to divide the body into finite elements, often just called elements, connected by nodes, and obtain an approximate solution as shown in Figure 2.1. This is called the finite element mesh and the process of making the mesh is called mesh generation. The FEM provides a systematic methodology by which the solution, in the case of our example, the temperature field, can be determined by a computer program. For linear problems, the solution is determined by solving a system of linear equations; the number of unknowns (which are the nodal temperatures) is equal to the number of nodes. To obtain a reasonably accurate solution, thousands of nodes are usually needed, so computers are essential for solving these equations. Generally, the accuracy of the solution improves as the number of elements (and nodes) increases, but the computer time, and hence the cost, also increases. The finite element program determines the temperature at each node and the heat flow through each element. The results are usually presented as computer visualizations, such as contour plots, although selected results are often output on monitors. This information is then used in the engineering design process. The same basic approach is used in other types of problems. In stress analysis, the field variables are the displacements; in chemical systems, the field variables are material concentrations; and in electromagnetics, the potential field. The same type of mesh is used to represent the geometry of the structure or component and to develop the finite element equations, and for a linear system, the nodal values are obtained by solving large systems (from  $10^3$  to  $10^6$  equations are common today, and in special applications,  $10^9$ ) of linear algebraic equations[3].



*Figure 2.1: Geometry, loads and finite element meshes.*

The preponderance of finite element analyses in engineering design is today still linear FEM. In heat conduction, linearity requires that the conductance be independent of temperature. In stress analysis, linear FEM is applicable only if the material behavior is linear elastic and the displacements are small. In stress analysis, for most analyses of operational loads, linear analysis is adequate as it is usually undesirable to have operational loads that can lead to nonlinear material behavior or large displacements. For the simulation of extreme loads, such as crash loads and drop tests of electronic components, nonlinear analysis is required. [3]

## 2.2 Introduction:

A beam constitutes the simplest way of spanning a gap between two objects. As structural elements, beams are prominent in both civil and

mechanical engineering. They are used as supports for floors in buildings, decks in bridges, wings in aircraft, or axles for cars. [4]

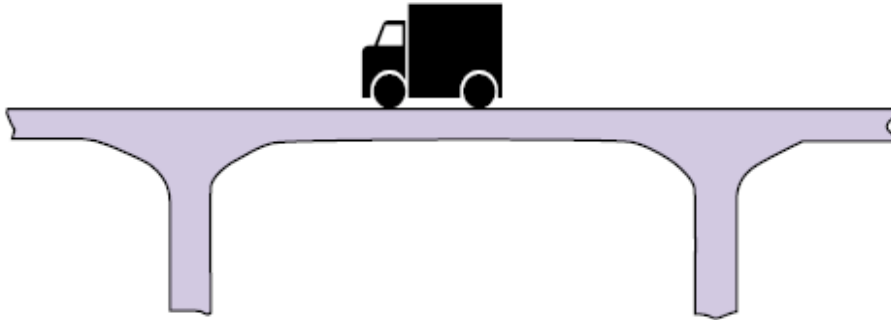


Figure 2.2: A beam is a structural member designed to resist transverse loads.

- The loads on beams may be point loads or uniformly distributed loads. The diagrams below show the way that the point loads and uniform loads are illustrated.

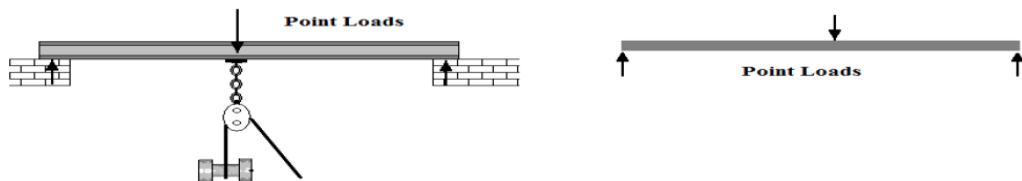


Figure 2.3: Point loads.

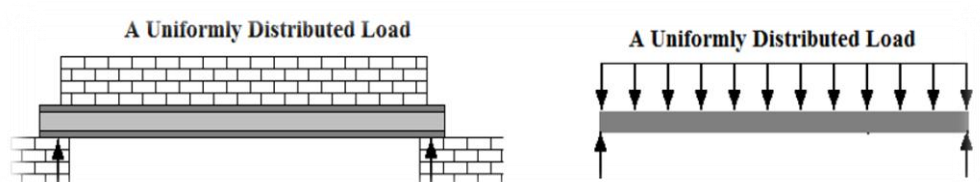


Figure 2.4: Uniformly distributed load.

- The beam can be simply supported or built in:

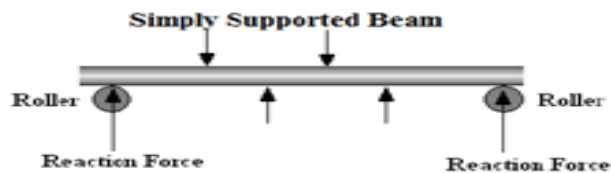


Figure 2.5 (a): A simply supported beam.

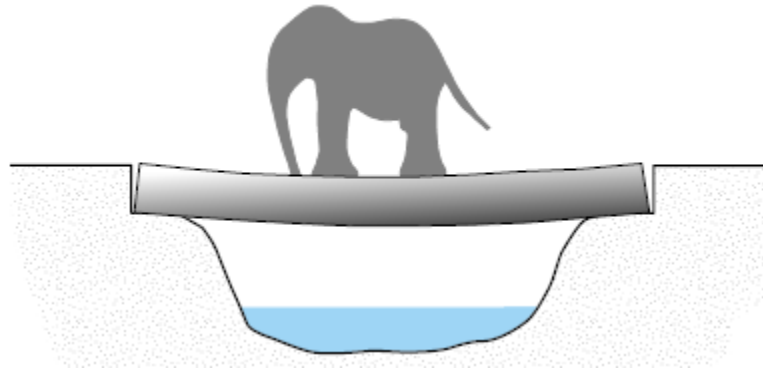


Figure 2.5 (b): A simply supported beam has end supports that preclude transverse displacements but permit end rotations.

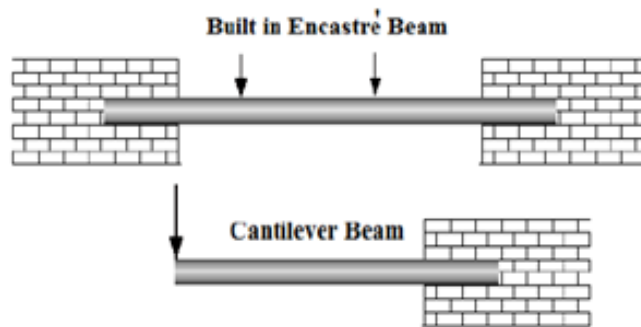


Figure 2.6 (a): Built in beam and cantilever beam.

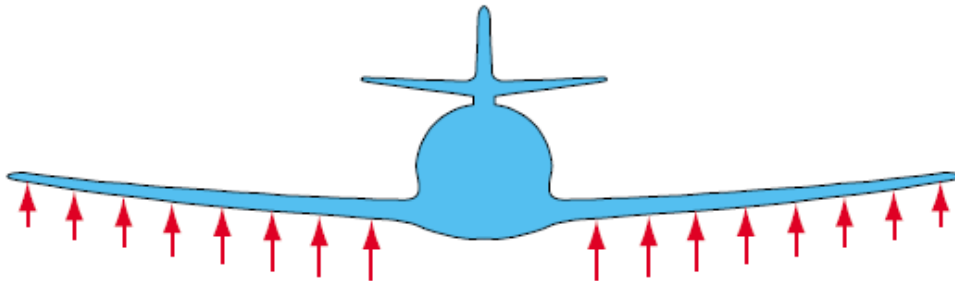


Figure 2.6 (b): A cantilever beam is clamped at one end and free at the other. Airplane wings and stabilizers are examples of this configuration.

### 2.3 Finite Element Method:

In order to analyze engineering system, a mathematical model is developed to describe the system. While developing the mathematical model, some assumptions are made for simplification. Finally the governing mathematical expression is developed to describe the behavior of the system. The mathematical expression usually consists of differential equations and given conditions.

These differential equations are usually very difficult to obtain solutions which explain the behavior of given engineering system. With the advent of high performance computers, it has become possible to solve such differential equations. Various numerical solution techniques have been developed and applied to solve numerous engineering problems in order to find approximate solutions. Especially, the finite element method has been one of the major numerical solution techniques. One of the major advantages of the finite element method is that a general purpose computer program can be developed easily to analyze various kind of problems. In particular, any complex shape of problem domain with prescribed conditions can be handled with ease using the finite element method.

The finite element method requires division of the problem domain into many sub domains and each sub domain is called a finite element. Therefore, the problem domain consists many finite element patches. [4]

## **2.4 Applications of Finite Elements:**

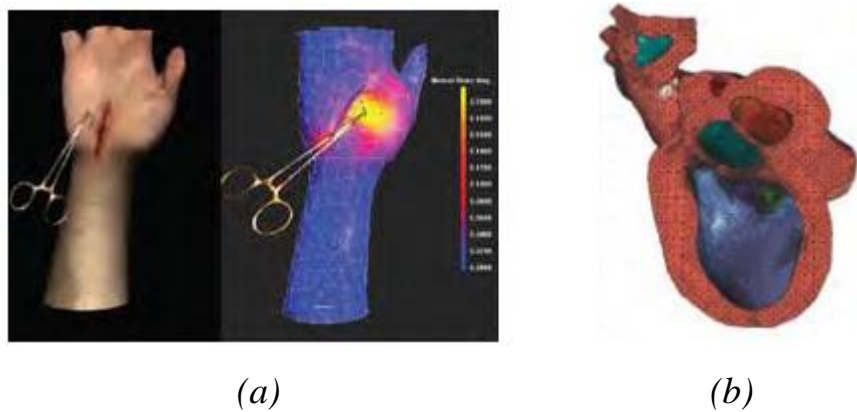
The range of applications of finite elements is too large to list, but to provide an idea of its versatility listed the following:

- 1) Stress and thermal analyses of industrial parts such as electronic chips, electric devices, valves, pipes, pressure vessels, automotive engines and aircraft;
- 2) Seismic analysis of dams, power plants, cities and high-rise buildings;
- 3) Crash analysis of cars, trains and aircraft;
- 4) Fluid flow analysis of coolant ponds, pollutants and contaminants, and air in ventilation systems;
- 5) Electromagnetic analysis of antennas, transistors and aircraft signatures;
- 6) Analysis of surgical procedures such as plastic surgery, jaw reconstruction, correction of scoliosis and many others.

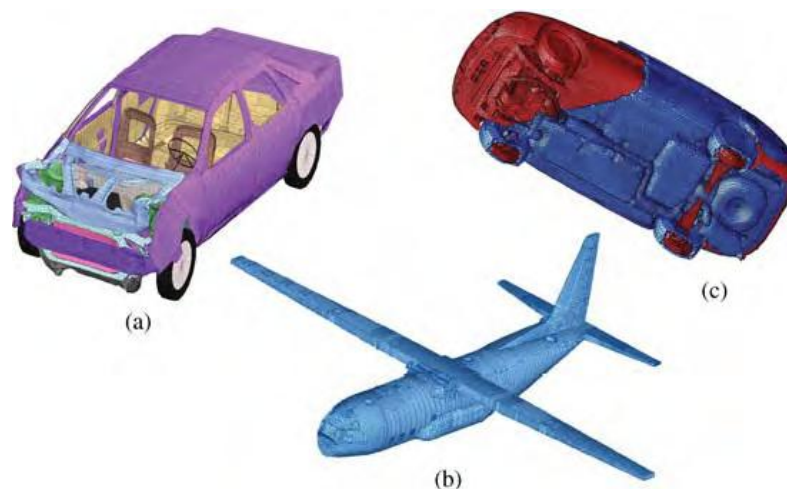
This is a very short list that is just intended to give an idea of the breadth of application areas for the method. New areas of application are constantly

emerging. Thus, in the past few years, the medical community has become very excited with the possibilities of predictive, patient-specific medicine.

One approach in predictive medicine aims to use medical imaging and monitoring data to construct a model of a part of an individual's anatomy and physiology. The model is then used to predict the patient's response to alternative treatments, such as surgical procedures. For example, Figure 2.7 (a) shows a hand wound and a finite element model. The finite element model can be used to plan the surgical procedure to optimize the stitches. Heart models, such as shown in Figure 2.7 (b), are still primarily topics of research, but it is envisaged that they will be used to design valve replacements and many other surgical procedures.



*Figure 2.7: Applications in predictive medicine. (a) Overlying mesh of a hand model near the wound. (b) Cross section of a heart model.*



*Figure 2.8: Application to aircraft design and vehicle crash safety: (a) finite element model of Ford Taurus crash; (b) finite element model of C-130 fuselage, empennage and center wing and (c) flow around a car. [2]*

## **2.5 Brief History:**

The modern development of the finite element method began in the 1940s in the field of structural engineering with the work by Hrennikoff in 1941 and McHenry in 1943, who used a lattice of line (one-dimensional) elements (bars and beams) for the solution of stresses in continuous solids. In a paper published in 1943 but not widely recognized for many years, Courant proposed setting up the solution of stresses in a variation form. Then he introduced piecewise interpolation (or shape) functions over triangular sub regions making up the whole region as a method to obtain approximate numerical solutions. In 1947 Levy developed the flexibility or force method, and in 1953 his work suggested that another method (the stiffness or displacement method) could be a promising alternative for use in analyzing statically redundant aircraft structures. However, his equations were cumbersome to solve by hand, and thus the method became popular only with the advent of the high-speed digital computer.[3]

In 1954 Argyris and Kelsey developed matrix structural analysis methods using energy principles. This development illustrated the important role that energy principles would play in the finite element method.

The first treatment of two-dimensional elements was by Turner et al .in 1956. They derived stiffness matrices for truss elements, beam elements, and two-dimensional triangular and rectangular elements in plane stress and outlined the procedure commonly known as the direct stiffness method for obtaining the total structure stiffness matrix. Along with the development of the high-speed digital computer in the early 1950s, the work of Turner et al. prompted further development of finite element stiffness equations expressed in matrix notation. The phrase finite element was introduced by Clough in 1960s when both triangular and rectangular elements were used for plane stress analysis.

A flat, rectangular-plate bending-element stiffness matrix was developed by Melosh in 1961s. This was followed by development of the curved-shell bending element stiffness matrix for axisymmetric shells and pressure vessels by Grafton and Strome in 1963s.[3]

Extension of the finite element method to three-dimensional problems with the development of a tetrahedral stiffness matrix was done by Martin in 1961, by Gallagher et al. In 1962, and by Melosh in 1963s. Additional three dimensional elements were studied by Argyris in 1964s. The special case of axisymmetric solids was considered by Clough and Rashid and Wilson in 1965s.

Most of the finite element work up to the early 1960s dealt with small strains and small displacements, elastic material behavior, and static loadings. However, large deflection and thermal analysis were considered by Turner et al. in 1960 and material nonlinearities by Gallagher et al. In 1962s, whereas buckling problems were initially treated by Gallagher and Padlog in 1963s. Zienkiewicz et al extended the method to visco-elasticity problems in 1968s.

In 1965s Archer considered dynamic analysis in the development of the consistent-mass matrix, which is applicable to analysis of distributed-mass systems such as bars and beams in structural analysis.

With Melosh's realization in 1963 that the finite element method could be set up in terms of a variational formulation, it began to be used to solve nonstructural applications. Field problems, such as determination of the torsion of a shaft, fluid flow, and heat conduction, were solved by Zienkiewicz and Cheung in 1965, Martin in 1968, and Wilson and Nickel in 1966.

Further extension of the method was made possible by the adaptation of weighted residual methods, first by Szabo and Lee in 1969 to derive the previously known elasticity equations used in structural analysis and then by Zienkiewicz and Parekh in 1970 for transient field problems. It was then recognized that when direct formulations and variational formulations are difficult or not possible to use, the method of weighted residuals may at times be appropriate. For example, in 1977 Lyness et al. applied the method of weighted residuals to the determination of magnetic field.[3]

In 1976 Belytschko considered problems associated with large-displacement nonlinear dynamic behavior, and improved numerical techniques

for solving the resulting systems of equations. For more on these topics, consult the text by Belytschko, Liu, and Moran.

A relatively new field of application of the finite element method is that of bioengineering. This field is still troubled by such difficulties as nonlinear materials, geometric nonlinearities and other complexities still being discovered.

From the early 1950s to the present, enormous advances have been made in the application of the finite element method to solve complicated engineering problems. Engineers, applied mathematicians, and other scientists will undoubtedly continue to develop new applications. For an extensive bibliography on the finite element method, consult the work of Kardestuncer, Clough, or Noor. [3]

## **2.6 Basic Steps of Finite Element Method:**

The basic steps involved in any finite element analysis consist of the following:

### **Preprocessing phase**

- 1) Create and discretize the solution domain into finite elements, that is subdivide the problem into nodes and elements.
- 2) Assume a shape function to represent the physical behavior of an element; that is, an approximate continuous function is assumed to represent the solution of an element.
- 3) Develop equations for an element.
- 4) Assemble the elements to present the entire problem. Construct the global stiffness matrix.
- 5) Apply boundary conditions, initial conditions, and loading.

### **Solution phase**

- 6) Solve a set linear of linear or nonlinear algebraic equations simultaneously to obtain nodal results, such as displacement values at different nodes or temperature values at different nodes in a heat transfer problem.

**Post processing phase**

- 7) Obtain other important information. At this point you may be interested in values of principal stresses, heat fluxes, etc.[6]

**2.7 Geometric Non-linearity:**

For genuine geometric nonlinearity, 'incremental' procedures were originally adopted by Argyris using the (geometric stiffness matrix) in conjunction with an updating of coordinates and possibly, an initial displacement matrix.

The incremental (or forward-Euler) approach can lead to an unquantifiable build-up of errors and, to counter this problem, Newton-Raphson iteration was used by Mallet and Marcal. Zienkiewicz and Oden also recommended a modified Newton-Raphson procedure a special form using the very initial. Elastic stiffness matrix was referred to as the (initial stress) method and much used with material non-linearity. The concept of combining incremental and iterative methods was introduced by Brebbia and Connor. Bath (1996) divided geometric nonlinearities into two groups:

- 1) Geometric nonlinearities with small strains and large displacements and rotations.
- 2) Geometric nonlinearities with large strains, displacements and rotations.

Two approaches may be used to handle geometric nonlinearities

- 1) The total Lagrangian formulation where all the variables are referred to initial configuration and
- 2) The updated formulation where all the variables are referred to the configuration at the beginning of the load step considered.

There are several approaches to perform a nonlinear finite element analysis; most of these divide the total load in steps to track the equilibrium path. The equilibrium point is approached by successive approximations to reduce the unbalanced load (difference between the applied and the internal nodal forces) in each iteration.

Structural nonlinearities can be specified as:

- 1) Geometrical nonlinearities: The effect of large displacements on the overall geometric configuration of the structure.
- 2) Material nonlinearities: Material behavior is nonlinear. Possible material models are, nonlinear elastic, elasto plastic, visco elastic, and visco plastic.
- 3) Boundary nonlinearities, i.e. displacement dependent boundary conditions. The most frequent boundary nonlinearities are encountered in contact problems.

Consequences of nonlinear structural behavior that have to be recognized are:

- 1) The principle of superposition cannot be applied. Thus, for example, the results of several load cases cannot be combined. Results of the nonlinear analysis cannot be scaled.
- 2) Only one load case can be handled at a time.
- 3) The sequence of application of loads (loading history) may be important. Especially, plastic deformations depend on a manner of loading. This is a reason for dividing loads into small increments in nonlinear finite element analysis.
- 4) The structural behavior can be markedly non-proportional to the applied load.
- 5) The initial state of stress (e.g. residual stresses from heat treatment, welding, cold forming etc.) may be important. [1]

Geometric nonlinearity requires a reformulation of the finite element problem.

A.E. Mohamed (1983) [7] presented a thesis on small strain–large rotation theory and finite element formulation of thin curved beams. He used both Green's strain and geometric (engineering) strain measures. He, also, proposed in an appendix a total Lagrangian modified incremental equations for a two–dimensional state of stress based on the geometric strains.

F.M. Adam (2008) [1] presented a thesis on the linear and nonlinear analysis of thin shell structures. The nonlinear formulations are based on the total Lagrangian formulation and using both Green's strain and geometric (engineering) strains. The study recommended using the logarithmic strains as a true measure of strains, to overcome the problems that resulted from the total Lagrangian formulation based on Green's strain, especially for large rotation and large deformation cases.

Jainil N Desai [8] presented a paper on Implementation of a Beam Element in Finite Element Analysis using MATLAB. The element implemented is a two node iso-parametric beam element. It solves for the deflection of the beam according to the boundary conditions and applied loads. He has implemented a MATLAB code to solve a cantilever beam or a simply supported beam with point loads at any location of the beam.