General Introduction

1.1 Introduction:

The finite element method is a numerical procedure for solution of differential equations. It is ideally suited for implementation on a computer. With the advancements in digital computer, the finite element method has become the method of choice for solving many engineering problems quickly and efficiently. It is extensively used for analysis of structures including air frames and ships, heat transfer and fluid flow, electric motors, heat engines and spacecrafts. It is also used for analyzing the behavior of components of biological systems[1].

The finite element model is created by dividing the structure into finite number of elements. The elements are inter connected by nodes only. The section of elements for modeling the structure depends upon the behavior and geometry of the structure being analyzed, for example frame elements, plane elements, plate elements, or shell elements. The structure can also be modeled by combining different types of elements to approximate different aspects of structural behavior. The modeling pattern, which is generally called mesh for the finite element method, is a very important part of the modeling process. The accuracy of the results obtained from the analysis depends upon the selection of the finite element type and the number of elements of the mesh. The equilibrium equations can easily be solved using digital computers without having to solve large number of partial differential equations by hand. The displacements at each node of the finite element model are obtained by solving the equilibrium equations. Then the stresses and strain can be obtained for each element from the stress- stain and strain- displacement relations[1].

1.2 Problem Statement:

Finite element packages for geometric nonlinear analysis are difficult to obtain and contain a number of limitations. Hence, developing finite element software for geometric nonlinearity will be of great help in research.

Recently, MATLAB had been used by many researches in developing linear finite element analysis using beam elements.

This research is an attempt to extend these programs to cater for geometric nonlinearity.

Solution of many engineering problems is based on linear approximations. In structural analyses, these approximations are represented by consideration that:

- Displacements are small and can be neglected in equilibrium equations,
- The strain is proportional to the stress,
- Loads are conservative, independent on displacements,
- Supports of the structure remain unchanged during loading.

In reality, behavior of structures is nonlinear. Solution of many engineering problems needs abandonment of linear approximations. Moreover, loads may change orientations according to displacements and supports may change during loading. Consequently, structure behaves nonlinearly. If these phenomena are included in a Finite Element Analysis, the set of equilibrium equations becomes nonlinear.

1.3 Objectives:

The objectives of the study are:

1) To learn how to develop geometric nonlinear beam finite element formulation based on Green strains using incremental iterative solutions.
2) To develop a geometrically nonlinear beam finite element program using MATLAB as a modification of linear program.
3) To apply the program for solution of case study problems.
4) To check the result obtained by applying the program using known solutions.

1.4 Methodology of study:

The Methodology of the study was as follows:

1) Carrying out an extensive literature review referring to the following references or information resources**:**
   i. Finite element methods books, journals and research papers.
   ii. Internet web site.
2) Choice of a cantilever beam example as a case study to carry out its analysis using different numbers of elements and load increments, and assuming that the strains are small and the displacements are large.
3) Developing the incremental equilibrium equations for the total Lagrangian Finite Element formulation of small strain, large displacement beams.
4) Developing the explicit formulation of the geometrically nonlinear beam finite element based on Green strains.

5) Modifying a linear beam finite element program and implementing the finite element programs using MATLAB to evaluate the displacements and stresses for the non linear formulations.
6) Application of program and comparison of its results with known solutions.
7) Writing up thesis.

1.5 Outlines of Thesis:

- **Chapter one** presents general introduction, research problem statement, objectives and methodology.
- **Chapter two** includes literature review which provides the definition of beams, Finite Element Method, Applications and Basic Steps in the Finite Element Method, Brief History, and Geometric non-linearity.
- **Chapter three** presents the methods for solution of nonlinear problems, geometric non-linear formulation for thin beams, and the formulation of 2-node straight beam finite elements.
- **Chapter four** contains the computer program developed using MATLAB and presents the necessary flow charts.
- **Chapter five** includes the results obtained for the case studies and the discussion of results.
- **Chapter six** contains of conclusions and recommendations.

Appendices:

- **Appendix (A):** Includes the components of the internal forces for calculation of residuals.
- **Appendix (B):** Includes linear displacement stiffness matrix (K_{0L}).
- **Appendix (C):** Includes MATLAB program sample input data and output results.

Appendix (D): cantilever beam