Static and Dynamic Nonlinear Analysis of Space Frame

By

Jaymin R. Desai

12MCLC33

DEPARTMENT OF CIVIL ENGINEERING

INSTITUTE OF TECHNOLOGY

NIRMA UNIVERSITY

AHMEDABAD-382481

May-2014

Static and Dynamic Nonlinear Analysis of Space Frame

Major Project

Submitted in Partial Fulfillment of the Requirements for the degree of

MASTER OF TECHNOLOGY

IN

CIVIL ENGINEERING

(Computer Aided Structural Analysis and Design)

By

Jaymin R. Desai

12MCLC33

DEPARTMENT OF CIVIL ENGINEERING

INSTITUTE OF TECHNOLOGY

NIRMA UNIVERSITY

AHMEDABAD-382481

May-2014

Declaration

This is to certify that

a. The major project comprises my original work towards the Degree of Master of Technology in Civil Engineering (Computer Aided Structural Analysis and Design) at Nirma University and has not been submitted elsewhere for a degree.

b. Due acknowledgement has been made in the text to all other material used.

Desai Jaymin R.

Certificate

This is to certify that the Major Project entitled **"Static and Dynamic Nonlinear Analysis of Space Frame"** submitted by **Mr. Jaymin R. Desai (12MCLC33)** towards the partial fulfillment of the requirements for the degree of Master of Technology in Civil Engineering (Computer Aided Structural Analysis and Design) of Nirma University, Ahmedabad, is the record of work carried out by him under my supervision and guidance. In my opinion, the work submitted has reached a level required for being accepted for examination. The results embodied in this major project work, to the best of my knowledge, have not been submitted to any other university or institution for award of any degree or diploma.

Date:

Dr. Paresh V. Patel
Guide,
Professor and Head,
Department of Civil Engineering,
Nirma University,
Ahmedabad.

Dr.K.Kotecha
Director,
Institute of Technology,
Nirma University,
Ahmedabad.

Examiner

Date of Examination

Abstract

Space Frame Structures are widely adopted for important structures like high-rise buildings, industrial structures, oil and gas platforms etc which require higher safety and accuracy in design. To understand actual behavior of structure, it should be analyzed rigorously by considering structural nonlinearities.

The present study is concerned with the analysis of space frame structure. The linear and nonlinear analysis for static and dynamic loading conditions are carried out. Main focus of analysis is on effects of geometric nonlinearity on behavior of space frame structure.

Space frame elements with six degrees of freedom (three translations and three rotations) at each node are considered. Single element per member is assumed. Linear static analysis is performed using direct stiffness method. Natural frequencies and mode shapes are calculated using free vibration analysis. Wilson-Theta method is used for time stepped analysis of space frame and as a result time history plots are developed. These methods are extended for nonlinear analysis. Mostly, space frames are consisting of slender sections which can have large deformations. Due to large deformations, second-order effects are developed which can be included in analysis by updating stiffness matrix. In present study, geometric stiffness matrix which is depending on deformed geometry is considered. Different solution methods for nonlinear equilibrium equations are also discussed in brief. For static nonlinear analysis, combined incremental and iterative method is adopted. Nonlinear response can be evaluated by updating geometric stiffness matrix at the end of each incremental step. To consider large strain and large deformation condition, both geometric stiffness as well as geometry of structure are updated at the end of each incremental step. Unbalanced forces are calculated at the end of each incremental step and are recovered by performing iterations. Same incremental-iteration method is used for time stepped

method. In nonlinear dynamic analysis, incremental forces are depended on time dependent forces. Combination of Wilson-Theta method and Newton-Raphson method is adopted for nonlinear dynamic analysis.

Detail procedures of linear and nonlinear analysis for both static and dynamic conditions are presented. Computer programs are developed based on these procedures. Computer programs are capable of finding linear and nonlinear response. Single bay single storey as well as multi storey frames are analyzed for static and dynamic loading conditions for both linear and nonlinear case. Space frame building with plan dimension of (15m $\times$ 20m) with varying number of storeys from 6 storey to 30 storey with storey height of 5m are analyzed to understand the effects of height on nonlinear behavior of space frame. Loading conditions are kept same for all types of multi-storey frames to compare results. In static loading condition, equivalent static earthquake loads based on IS:1893-2002 are considered for multi storey frames and for dynamic case, time dependent lateral forces are applied at top storey of each frame. Results of linear analysis are compared with ETABS results for validation. Nonlinear analysis results are compared with finite element analysis results to validate procedure of nonlinear analysis. Load displacement curves and time history curves are plotted based on results obtained from developed programs.

Acknowledgement

I would like to express my immense gratitude to my guide **Dr. Paresh V. Patel**, Head of Civil Engineering Department, Institute of Technology, Nirma University, Ahmedabad for his valuable guidance and continual encouragement throughout my major project work. His constant support and interest in subject equipped me with a great understanding of different aspects of the major project work. His extreme supervision and direction right from beginning motivate me to complete this work.

My sincere thanks to **Dr. Sharad P. Purohit**, Professor, Civil Engineering Department and **Dr. Urmil V. Dave**, Professor, Civil Engineering Department for their kind suggestions and motivational words throughout the major project work.

A special thanks to **Dr K Kotecha**, Hon'ble Director, Institute of Technology, Nirma University, Ahmedabad for providing required resources for my project and healthy research environment.

I would like to thank all my friends for their everlasting support and encouragement in all possible ways throughout the major project work.

Most importantly deepest appreciation and thanks to Almighty and my family for their unending love, affection and personal sacrifices during the whole tenure of my study at Nirma University.

- Desai Jaymin R.
12MCLC33

Abbreviations, Notations and Nomenclature

DOFs . . . Degrees of Freedom

$[S_j]$ . . . Global Joint Stiffness Matrix

$\{D\}$ . . . Displacement Vector

$\{F\}$ . . . External Force Vector

$[S_m]$ . . . Member Stiffness Matrix

A . . . cross sectional area of element

E . . . modulus of elasticity

μ . . . poisson's ratio

I_x . . . moment of inertial about X axis

I_y . . . moment of inertial about Y axis

I_z . . . moment of inertial about Z axis

G $= \frac{E}{2(1+\mu)}$ . . . shear modulus

L . . . length of element

(x_j, y_j, z_j) . . . coordinate of j-node or j-joint

(x_k, y_k, z_k) . . . coordinate of k-node or k-joint

$C_x = \frac{x_k - x_j}{L}$ . . . direction cosine for x axis

$C_y = \frac{y_k - y_j}{L}$ . . . direction cosine for y axis

$C_z = \frac{z_k - z_j}{L}$ . . . direction cosine for z axis

$C_{xz} = \sqrt{C_x^2 + C_z^2}$

$[R_t]$ . . . Rotation matrix

α, β, γ . . . Angle of rotation of member axes with respect to global axes

$\{A_j\}$ . . . Joint Load vector

$\{D_s\}$ . . . Displacement with respect to global coordinate system

$\{D_m\}$ . . . Displacement with respect to local coordinate system

$\{A_m\}$ . . . Member end actions

$\{A_{gg}\}$ . . . Induced action vector with respect to global coordinate system

$[M]$ . . . Mass matrix

$[C]$ .... Damping matrix
$[K]$ .... Stiffness matrix
$\{\ddot{u}\}$ .... Acceleration Vector
$\{\dot{u}\}$ .... Velocity Vector
$\{u\}$ .... Displacement Vector
$\{\ddot{X}_g\}$ .... Ground acceleration
$\bar{m}$ .... Mass per unit length
$I_{\bar{m}} = \bar{m}I_x/A$ .... Polar mass moment of inertial
N_1,N_2,N_3,N_4 .... Displacement Function
ω_i .... Natural Frequency of i^{th} mode
ξ .... Damping ratio
Δt .... Time increment
$\{\Delta F\}$ .... Incremental Load Vector
$\tau = \theta\Delta t$ .... Extended time increment
$[K_e]$ .... Elastic Stiffness Matrix
$[K_g]$ .... Geometric Stiffness Matrix
$[K_t] = [K_e] + [K_g]$ .... Tangent Stiffness Matrix
$\{g\}$ .... Unbalanced Force Vector
$\{dd\}$ .... Displacement vector corresponding to unbalanced force
x, y and z .... Coordinates of nodes
$\{D_x\}$, $\{D_y\}$ and $\{D_z\}$ .... Displacement Vector in the direction of X,Y and Z
λ .... Load Factor
NLA .... Non-Linear Analysis

Contents

List of Tables

List of Figures

Chapter 1

Introduction

1.1 General

Space frames are most generalized structure that can represent different class of structures. Space frames are widely used as supporting structure for low-rise building, industrial buildings, oil and gas platforms, high-rise buildings etc. Space frame is more popular because of its simplicity in construction. It also possesses typical and speedy construction procedure and hence highly recommended by engineers. It also provides flexibility to assemble other architecturally attractive structures.

From structural engineer point of view, space frame is the three dimensional structure consisting of beams and columns which are so arranged that forces are transferred in three dimensional manner. Also space frame elements are long in comparison to their cross-sections. Every frame has many joints which can be points of intersection of the elements, points of support and free end of elements.

Analysis of structure is prerequisite for designing any structure. Structural analysis is used to determine internal forces, stresses, displacements and strains for given loading conditions. Based on the analysis results, structural engineers are able to design structures for different load combinations to withstand under worst conditions. Structural analysis is also useful to understand the behavior of space frame in terms of load-deformation curves for different loading conditions. These are useful to check whether the proposed structural design is efficient or not.

Mostly space frames are statically indeterminate so that it is very difficult to analyze it through hand calculations. Also it is difficult to understand general behavior of space frame as it has many elements which are in space so that specific load path can not be predicted. Space frame behaves differently for different conditions like for low rise building it is behaving like shear frame whereas for high rise building it is behaving as cantilever frame. To predict accurate behavior of space frame, appropriate analysis techniques should be adopted. There are numbers of analysis techniques which involve different nonlinearities and load behavior. For example if frame is subjected to large deformation then geometric nonlinear analysis is helpful. The choice of analysis depends on the specific requirements. All these analysis problems can be solved by use of computers. Introduction of efficient computer based analysis techniques reduce time and efforts for analysis of complex space frame structures.

1.2 Types of Structural Analysis

Structural analysis is mainly concerned with finding out the internal forces, deformations and behavior of frame elements when subjected to external forces. Various analysis types are available which depend on the types of loads, stress-strain relationship of material and induced second-order effects. Types of analysis are

a. **Static Analysis**

In static analysis, effects of steady loading conditions on a structure are calculated. Generally all loads except dead load on structure are time dependent. But when the variation of loads are less or in other word, changes in magnitude and direction of loads are sufficiently slow, induced inertia forces can be neglected. Hence, Static analysis is simplified analysis procedure. Static analysis used to determine displacements, stresses and strains induced due to external loads which do not induce inertia and damping effects.

Loads like externally applied forces, gravity forces, imposed displacements, temperature effects etc. are included in static analysis.

b. **Dynamic Analysis**

Dynamic Analysis is a type of analysis which covers the behavior of structures subjected to dynamic loading which have high acceleration. Changes in magnitude or direction of loads are faster compared to natural frequency of structure. Loads like wind load, blast load, earthquake load are dynamic in nature. In dynamic analysis inertia forces, developed due to the applied dynamic load, are also considered. Damping effects in structure can also be included in dynamic analysis.

Structural response in terms of displacement, velocity and acceleration are calculated in dynamic analysis. Different methods are available for time-history analysis as well as modal analysis.

c. **Linear analysis**

Every structure possesses some nonlinearity but to understand its behavior quickly, linear analysis is adopted. There are some basic considerations for linear analysis

- Equations of equilibrium are based on the initial geometry of structure and remain same after load application.
- Material properties do not change.
- Stiffness of structure remains same throughout the analysis
- Structure returns to its original form when loads are removed
- Deformation and strain in every member are small.
- Material has infinite strength and infinitesimal deformation.

Linear analysis is widely used by professionals but it has some limitations. When any of these conditions is not satisfied than nonlinear analysis is adopted.

d. **Nonlinear analysis**

When equilibrium equations for any structure are derived based on its deformed geometry and actual material properties than realistic behavior can be traced. This type of analysis is called nonlinear analysis. Nonlinearity in terms of material, boundary conditions and geometry can be included and it requires some mathematical modeling. Nonlinear analysis is not popular because it is complex and time consuming.

1.3 Need of Nonlinear Analysis

Linear elastic analysis is adopted by design engineers to predict member forces, stresses and displacements because it is simple and one stepped analysis method for given loads. It does not require any load increment or iterations. For different load cases, principle of superposition can be applied for linear analysis. But it is unable to predict exact solutions as all structures have nonlinear behavior before reaching to ultimate conditions or limit state conditions. Hence, modern codes include some nonlinear procedures. By considering nonlinear behavior, one can achieve realistic strength against failure conditions and actual factor of safety. Nonlinear analysis is also highly recommended when structure subjected to abnormal loading conditions like blast condition and during progressive collapse etc. For special structures like power plants, gas and oil plants, it is required to understand nonlinear performance of structure to achieve desire safety levels. To develop new high strength materials for areas like aero-space engineering, tall structures and mechanical engineering, nonlinear analysis is adopted.

Different nonlinear Analysis for structural engineering field are given below:

- **Strength analysis**
  Strength analysis is performed to estimate the load, the structure can support before global failure occurs.
- **Stability analysis**
  It is used to find critical or limit conditions which are used for designing purpose.
- **Service configuration analysis**
  This type of analysis is used to find equilibrium at the service stage or fabrication stage for slender structure like cable stayed bridge.

- **Reserve strength analysis**
  Reserve strength analysis is important to find load carrying capacity beyond limit states to assess safety under worst conditions.
- **Progressive failure analysis**
  It is the combination of stability and strength analysis in which progressive failure is considered.

For structural engineering, these analysis techniques are useful to understand causes of failure of structure, safety and serviceability assessment of existing structure which may have cracks, corrosion or aging conditions. It is also required to utilize full strength of materials so that more efficient design can be proposed.

For research purpose, nonlinear analysis is also used to propose new material models. For developing new guidelines for design, it is required. Nonlinear analysis requires large computations which are very difficult for manual calculation. Development in computational capacity of modern computers and advanced procedure, make nonlinear analysis popular for professionals.

1.4 Impact of Nonlinear Analysis

Compared to linear analysis, following conditions should be considered before adopting nonlinear analysis.

- The principle of superposition cannot be applied.
- Results of several load cases can not be scaled for combination of actions which can be done in linear analysis.

- At a time one load case can be handled.
- Initial stress and strain levels are important for overall response of structure.
- Sequence of applied load is important to predict actual response.

1.5 Types of nonlinearities

As discussed in previous sections, every structure possesses nonlinearities. The sources of nonlinearities for structural engineering problems are

(1) Geometric Nonlinearity

(2) Material Nonlinearity

(3) Contact Nonlinearity

Geometric Nonlinearity

When geometry of structure is changing as it deforms, the relation between force and displacement does not remain linear. This type of nonlinearity is known as Geometric Nonlinearity. In precise way, geometric nonlinearity cause additional actions which are due to large deformations and change in position of loads. It is associated with second order effects like P-delta effects. Second order effects can be calculated by updating stiffness of the structure which is depended on its deformed geometry. This can be understood with example of frame. In Fig1.1(a), deflected lumped mass model of frame structure is shown. Due to deflection, position of gravity load changes and additional forces are induced in structures which is in form of horizontal forces between two consecutive storeys as shown in Fig1.1(b). This is one of the examples of geometric nonlinearity.

Effect of one degree of freedom on other degree of freedoms are considered in geometric nonlinearity using geometric stiffness.

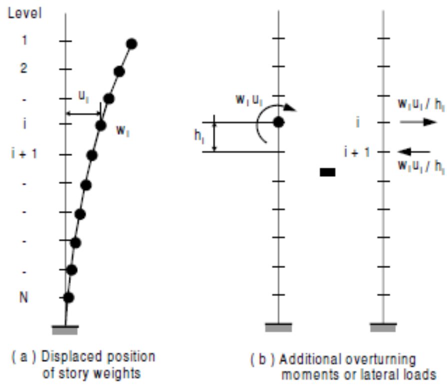

Figure 1.1: Load direction before and after deflection [16]

Geometric nonlinearity is included in space frame analysis when sections of members are too slender or frame has large displacement.

Material Nonlinearity

Structural material has nonlinear stress-strain relationship after beyond elastic limit and hence displacement and forces are affected. Nonlinear constitutive relationship between stress and strain affects stiffness of structure. Many structural materials have large capacity to deform in plastic range and to utilize full capacity of material, material nonlinearity should be considered in analysis. Other variables like prestress, temperature, time, moisture etc can be included in material nonlinearity. Plasticity, creep and visco-elasticity are effects of material nonlinearity.

Different mathematical models are available for different material types which can be used to consider effects of material nonlinearity. Fig 1.2 shows nonlinear stress-strain relationship for concrete suggested by IRC 112 - 2011 for nonlinear structural analysis.

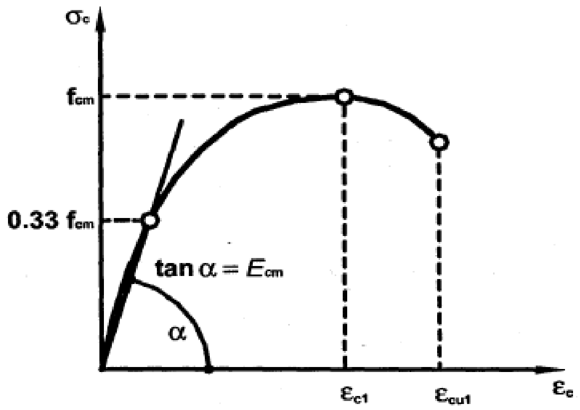

Figure 1.2: Nonlinear Stress-Strain relationship for concrete

Contact Nonlinearity

Contact nonlinearity is also known as boundary condition nonlinearity. When boundary conditions change with respect to displacement magnitude than this conditions should be considered in analysis. Fig 1.3 is example of contact nonlinearity. Under the effect of loading condition when cantilever beam deflects by **d** units, its boundary condition changes.

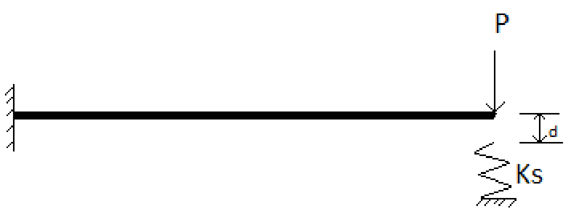

Figure 1.3: Example of Contact Nonlinearity

1.6 Nonlinear analysis of Space Frames

As discussed earlier, space frames are widely used for important structures which require accuracy in design process. For that actual behavior of space frame is required to find out for worst cases. Space frame elements have small cross-section compare to their length so that space frame structure is highly affected to second-order effects. There are wide range of high strength structural materials that are used for space frame construction are available. It is always advisable to use full capacity of materials. So both geometric nonlinearity and material nonlinearity are considered in analysis of space frame. Many times based on the demand, various combinations of nonlinearities are included for nonlinear analysis. Basic combinations of nonlinearities are listed below: [16]

(1) **First order elastic analysis**
This is linear analysis in which both geometric and material nonlinearities are not considered. This is simplest form of structural analysis.

(2) **Second order elastic analysis**
In this analysis, only geometric nonlinearity is considered to trace second-order effects like P-delta analysis, buckling analysis.

(3) **First order elasto-plastic analysis**
This analysis only includes material nonlinearities. Space frame having small deformation where second-order effects are too less so that geometric nonlinearity can be neglected.

(4) **Second order elasto-plastic analysis**
It is most extreme condition where both geometric and material nonlinearities are considered in analysis.

Above mentioned methods are useful to predict ultimate load carrying capacity as well as static or dynamic response of space frame for required nonlinear conditions.

1.7 Objectives of the study

Nonlinear analysis results gives accurate stresses, strains, displacements and internal forces for space frame. Space frame is most common type type of structure which requires safe and economical design. As discussed earlier in section 1.3, different nonlinear analysis are applicable to space frame. These analysis types may be static or dynamic. To understand behavior space frame, rigorous analysis procedure is required which may includes all type of nonlinear analysis discussed in section 1.5.

Looking to above need the objectives of present study are:

- To study linear static and dynamic behavior of space frame and develop computer program.
- To study the effects of nonlinearities on space frame and understand procedure to include them in analysis procedure.
- To obtain nonlinear equilibrium equations for static and dynamic loading conditions.
- To establish detail solution procedure for nonlinear static and dynamic equilibrium equations.
- To develop computer program for static and dynamic nonlinear analysis for space frame.

1.8 Scope of Work

To achieve above objectives the scope of work is decided as follows:

- Linear static analysis of space frame using direct stiffness method.
- Free vibration analysis to obtain dynamic properties of space frame.
- Developing computer program for time history analysis of space frame using Wilson-Theta method.
- Develop method to include nonlinear effects in static and dynamic analysis techniques.
- Formulation of nonlinear iterative analysis procedure for both static and dynamic analysis.
- Developing computer program for nonlinear static and dynamic analysis of space frame in which geometric nonlinearity can be incorporated.
- Analysis of various multi storey space frames for same plan dimensions and storey height for static and dynamic loading conditions for both linear and nonlinear case using developed computer program.

1.9 Organization of report

The contents of major project are presented in various chapters as follows.

Chapter 1 gives an overview about space frames. It introduces types of nonlinearities and different types of analysis techniques. Applications of nonlinear analysis are also mentioned. Based on need of the study, objectives and scope of work are defined.

Literature review related to nonlinear analysis is presented in **Chapter 2**. It provides information about implementation of geometric and material nonlinearities. It includes review of different papers related to analysis procedure for static and dynamic analysis.

Linear analysis procedures are presented in **Chapter 3**. Computer oriented analysis procedures for static linear, free vibration analysis and forced vibration analysis are discussed. Development of program is also presented in this chapter.

In **Chapter 4**, Results of linear static and dynamic analysis obtained from computer programs are also included and validated with ETABS results.

Chapter 5 gives solution methods for nonlinear equations. Geometric stiffness matrix for space frame element is presented. Detail procedures for nonlinear static and dynamic analysis are also discussed. Computer program development for nonlinear analysis is also discussed.

Implementation of computer program for space frame is discussed in **Chapter 6**. Results of nonlinear static analysis are also validated.

Summary and conclusion of present study is reported in **Chapter 7**. Future scope of work is also included in this chapter.

Chapter 2

Literature Review

2.1 General

A brief review of previous studies on nonlinear analysis of frame structure is presented in this chapter. Literatures also focus on effects of material and geometric nonlinearities to Space frames. Various numerical methods for solving nonlinear equilibrium equations are also reviewed. Various research papers have been refereed to understand theoretical formulation of tangent stiffness matrix for space frame elements. The available literature has been classified in three categories as follows

a. Formulation of tangent stiffness matrix for space frame element

b. Solution techniques for nonlinear equilibrium equations

c. Nonlinear analysis procedure for both geometric and material nonlinearities

2.2 Formulation of Tangent Stiffness Matrix

Yang and McGuire[1] presented geometric stiffness matrix for space frame element. Derivation of stiffness matrix was started from principle of virtual displacements. Derivation of stiffness matrix for three dimensional element was evaluated based on its deformed shape. Incremental equilibrium equation was developed which was based on deform geometry of space frame and updated Lagrangian formulation. Defined geometric stiffness matrix was symmetric and coefficients of matrix were depended on the induced member end actions at the end of previous incremental step. Hence tangent stiffness matrix, summation of elastic stiffness matrix and geometric stiffness matrix, should be modified at the end of each incremental step.

Chang[2] presented higher order stiffness matrix for space frame. By incorporating higher order stiffness matrix, number of iterations were reduced. Author had derived higher order stiffness matrix by using rigid body rotation. When there was any incremental load existing on structure, structure was allowed to undergo small rigid rotation. Due to this rotation, Magnitude of forces remained same but the directions of force changed hence additional actions were induced. Author had included these actions in geometric stiffness matrix for space frame element and hence number of iterations were reduced in force recovery step. Equilibrium equations were derived from energy method. Proposed tangent stiffness matrix had elastic stiffness, geometric stiffness and higher order stiffness matrix. $[K_t] = [K_e] + [K_g] + [K_i]$.

Kassimali and Abbasnia[3] studied geometric nonlinearity for elastic space frame. Methods for large deformation analysis and stability analysis of elastic frame was presented that considered large rotation and displacements at joints. Load-deformation relationship for beam-column element was derived in which effect of axial force on other degrees of freedom were considered. Detail procedure for analyzing space frame was discussed. Change in chord length due to axial strain and flexural bowing were taken into consideration. Due to large deformations, coordinates were updated and based on that transformation matrix was also recalculated. Author also suggested to divide each member into small elements to reduce errors in computations.

Teh and Clarke[4] presented symmetry of tangent stiffness matrix. Symmetric tangent stiffness matrix was change to asymmetric matrix due to introduction of higher order stiffness matrix. These matrices were derived from Lagrangian formulations. Asymmetry of stiffness matrix increases problems in matrix analysis method. Authors had derived symmetric correction matrix for space frame element by considering equilibrium at node which could be used in direct stiffness method. This method can be used with modified Newton-Raphson method to predict buckling of 3-D frame.

2.3 Solution techniques for nonlinear equilibrium equations

Chajes and Churchill[5] presented different methods to plot nonlinear load-deformation curve. Geometric nonlinearity for elastic structure was considered. Detail procedure for solution methods like linear incremental method (Fig 2.1(a)), nonlinear incremental method (Fig 2.1(b)) and direct method (Fig 2.1(c)) were established. Governing equilibrium equations were derived and steps for solving them were described for all methods. Main purpose of these methods was to find approximately exact behavior of frame. Tangent stiffness matrices and Secant stiffness matrices were found using energy principle for incremental method and direct method respectively.

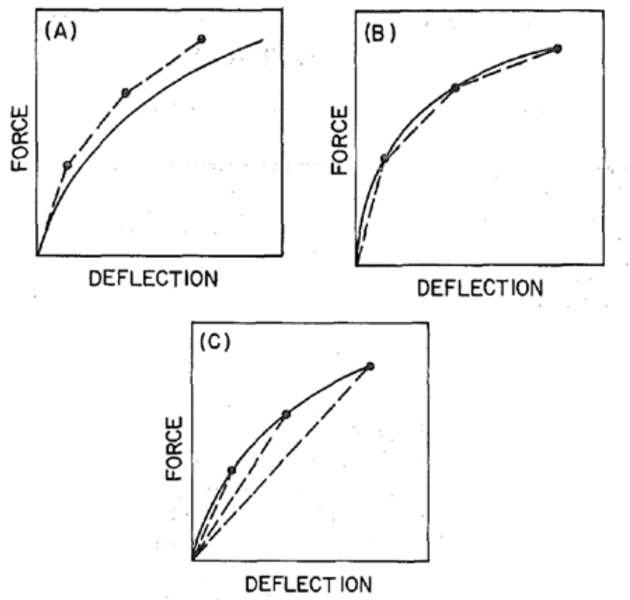

Figure 2.1: (A) linear incremental method, (B) Nonlinear incremental method, (C) Direct Method

Yang et al.[6] studied incremental method for nonlinear analysis of space frame with assumption of large deformation conditions. Incremental equation of equilibrium was derived based on updated geometry of element using updated Lagrangian formulation. Derived geometric stiffness matrix was asymmetric which was canceled out by considering equilibrium condition for deformed conditions. Incremental equation $([k_e] + [k_g])\Delta u = \Delta F$ was solved using incremental iterative procedure. Geometry of space frame and transformation matrix for each member were updated after each incremental step.

Suk et al.[7] presented effective solution technique that reduced time of computations for geometric dynamic nonlinear analysis. Predictor method was developed to reduce time and efforts. Presented method was capable to replace modified Newton-Raphson method. Algorithm of iteration scheme for dynamic nonlinear analysis was summarized. Undamped dynamic analysis was done using Newmark Beta method. At each step, before stating iteration, prediction related to displacement was done and then nonlinear iteration was started from predicted point so that required iterations were decreased. Results suggested that predictor iteration method was able to reduce 40-60 % time of computation. This was verified with dynamic analysis results of cantilever beam and rotating plate.

2.4 Nonlinear Analysis of Space Frame

Thai and Kim[8] studied steel space frames including both geometric and material nonlinearities. Fibre beam-column element was used. Geometric nonlinearities were included using stability functions obtained from the stability solution of beam-column. To consider material nonlinearities, stress-strain relation was adopted using uniaxial stress-strain relations of fibers on the cross-sections. One element for beam and column was proposed. Combined incremental and iterative method was used for nonlinear analysis procedure. Effect of axial force and transverse displacement (P-delta effect) was predicted for space frame. Flexural buckling of column for various end cases like free at top and fixed at bottom, hinged at both ends, fixed beam and space frames were analyzed. Above results were verified with the commercially available software SAP2000. 20 storey steel space frame was analyzed and validated with the available literatures.

Nguyen and Kim[9] studied steel space frame with semi-rigid connections. Nonlinear elastic dynamic analysis was done by considering only geometric nonlinearity. Beam-column element was proposed for beam and column members. Tangent stiffness matrix was calculated using stability function and geometric stiffness matrix. Combination of Newmark numerical integration method and Newton-Raphson method was used to find dynamic properties of space frame. Semi-Rigid Connections were modeled by spring. Three translational and three rotational springs were considered at each joint. Zero length members were used for nodal springs. Programming was done in FORTRAN language and was verified by taking various literature problems as well as software results. Various dynamic loading conditions were included in program. Examples of various space frames subjected to impact load, time dependent force and earthquake excitation were analyzed.

Abbasnia and Kassimali[10] presented method for large deformation elastic-plastic analysis of space frame. Large deformation was considered for elastic system and then extended it for elastic-plastic effects. Beam-column approach was adopted for frame members which consider change in geometry and its effect on stiffness. Material was assumed as elastic-plastic and yielding was considered at the end of member. It implied that plastic hinges would develop at the end of the member. Member was remain elastic between two plastic hinges developed at the end of each member. To define force-displacement relationship, stability function was used. Four structures were analyzed in which both nonlinearities were taken into consideration.

Liew et al.[11], developed improved analysis technique for plastic hinge analysis of space frame. Beam-column element was used for frame members and beam-column formulation was done by stability interpolation function. Material nonlinearity was modeled using concentrated plastic hinge method. Plastic hinges between two ends of member were allowed to developed. Incremental method with incremental load control have been used. Elastic-plastic tangent stiffness matrix was used in analysis process. Method can be used to predict buckling load of column, lateral torsional buckling. Author suggested that accuracy of analysis would be improved by considering more number of element for beam and column of space frame. Six storey frame (Fig 2.2) was analyzed and results are shown in Fig 2.3

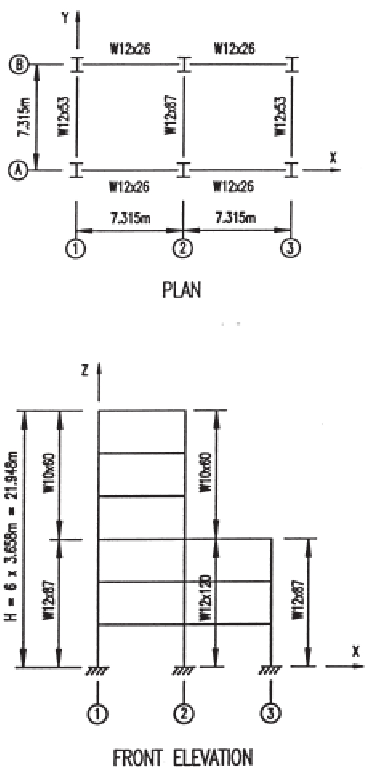

Figure 2.2: Space frame Detail

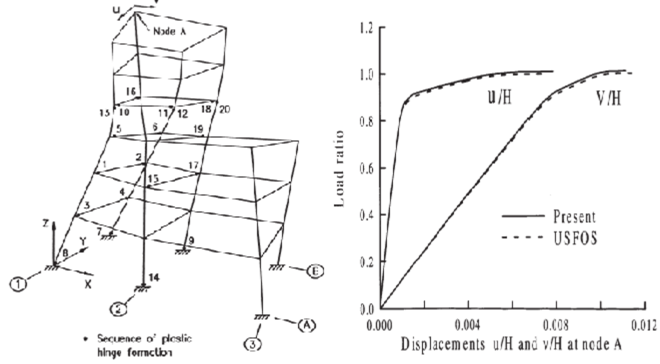

Figure 2.3: Hinge Location and Load-displacement curve

Jiang et al.[12], presented spread of plasticity analysis of space frame. Material nonlinearity was modeled through Von Mises yield criteria. Experimental and benchmark solutions were used to verify proposed method. Analyzed frame detail was shown in Fig 2.2. Analysis requires discretization of member into number of elements. Author also proposed maximum number of element as nine. Method proposed is highly computational intensive but it can be used to predict lateral torsional buckling.

Chan[13] studied analysis of space frame using secant stiffness matrix. Large deformation static and dynamic analysis were done for various conditions and presented. For nonlinear time history, response for space frame was obtained using combination of Newmark Beta and Newton-Raphson approach. Bernoulli's assumption of normal plane section was adopted and Strains in space frame element were assumed small but displacements and rotations were large. Linear elastic material was taken into consideration. Suggestion was made to stop iterations in static and dynamic nonlinear analysis. When unbalanced force was less than 0.1% of total applied load, iterations were terminated. Technique had been verified using number of examples.

Mansouri and Saffari[14] proposed numerical method to solve nonlinear equation. Proposed method was compared with Newton-Raphson method and proved as more efficient and time saving. Method is known as Two-point iterative method. Computational time was reduced by 40 % as number of iteration could be reduced.

2.5 Summary

Brief review of literature on nonlinear analysis of space frame structures is presented in this chapter. Reviewed papers and books are useful to build the foundation for nonlinear analysis procedure. Literature includes solution techniques to solve nonlinear equations, methods to derive geometric stiffness matrix, static nonlinear analysis of elastic frame using incremental-iteration methods, nonlinear time history analysis method and applications of nonlinear analysis of space frame.

Chapter 3

Static and Dynamic Linear Analysis of Space Frame

3.1 General

As discussed earlier, linear analysis gives response of space frame which may be good approximation for some portion of nonlinear response. Many structures remain in operational or service stage and for those conditions, linear analysis is widely accepted. It is also required to understand linear analysis procedure because nonlinear analysis is nothing but linear sequential analysis or say incremental linear analysis. In this chapter, linear analysis procedure are described for both static and dynamic conditions. Methods to solve multi-degrees of freedom (MDOF) system is also discussed. Computer oriented analysis procedure, Direct stiffness method is used for linear static analysis.

3.2 Space Frame Element

Behavior of space frame can be found out by finding response of each members of space frame. In typical analysis problem, behavior of each member under certain loading conditions can be described by governing equation. Space frame consists of many member which are connected to each other which is known as node. When behavior of each node is known than only whole response of frame can be found out. To get response at each node, proper degrees of freedom should be considered as per field conditions. Then boundary value problem is to be solved to get response of each node.

Here, for space frame where members are oriented in three dimension so that six degrees of freedom are considered. (Three translational and three rotational DOFs). For each member consisting two nodes, there are total 12 DOFs. Here beam element with 12 DOFs and corresponding actions are shown in Fig. 3.1. Beam having length L has two end **A** and **B**.

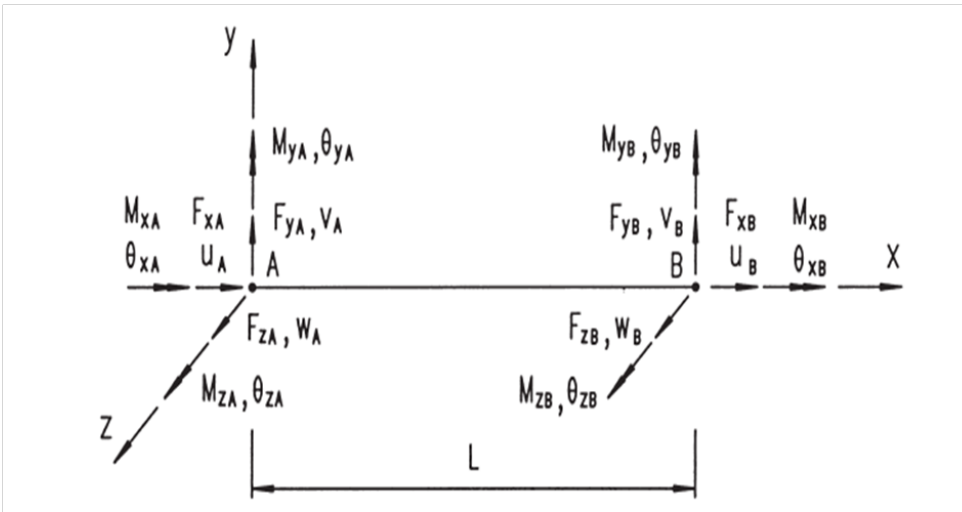

Figure 3.1: Space Frame Element

3.2.1 Stiffness Matrix for Space Frame Element

Direct stiffness method is matrix analysis method. In which response is calculated using equation:

$$[S_j]\{D\} = \{F\} \tag{3.1}$$

Here $[S_j]$ is global joint stiffness matrix. To construct structure stiffness, it is required to find member stiffness matrix $[S_m]$. To find stiffness matrix for prismatic space frame element shown in Fig. 3.1, consider unit deformation for single DOF at a time and find corresponding restraint actions at two ends. This is nothing but joint forces to produce unit deformation for particular DOF. Repeat this procedure for all DOFs. This will form stiffness matrix which is symmetric square matrix.

$$[S_m] = \begin{bmatrix}
\frac{EA_x}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{EA_x}{L} & 0 & 0 & 0 & 0 & 0 \\
 & \frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} & 0 & -\frac{12EI_z}{L^3} & 0 & 0 & 0 & \frac{6EI_z}{L^2} \\
 & & \frac{12EI_y}{L^3} & 0 & -\frac{6EI_y}{L^2} & 0 & 0 & 0 & -\frac{12EI_y}{L^3} & 0 & -\frac{6EI_y}{L^2} & 0 \\
 & & & \frac{GI_x}{L} & 0 & 0 & 0 & 0 & 0 & -\frac{GI_x}{L} & 0 & 0 \\
 & & & & \frac{4EI_y}{L} & 0 & 0 & 0 & \frac{6EI_y}{L^2} & 0 & \frac{2EI_y}{L} & 0 \\
 & & & & & \frac{4EI_z}{L} & 0 & -\frac{6EI_z}{L^2} & 0 & 0 & 0 & \frac{2EI_z}{L} \\
 & & & & & & \frac{EA_x}{L} & 0 & 0 & 0 & 0 & 0 \\
 & & sym & & & & & \frac{12EI_z}{L^3} & 0 & 0 & 0 & -\frac{6EI_z}{L^2} \\
 & & & & & & & & \frac{12EI_y}{L^3} & 0 & \frac{6EI_y}{L^2} & 0 \\
 & & & & & & & & & \frac{GI_x}{L} & 0 & 0 \\
 & & & & & & & & & & \frac{4EI_y}{L} & 0 \\
 & & & & & & & & & & & \frac{4EI_z}{L}
\end{bmatrix} \tag{3.2}$$

3.3 Linear Static Elastic Analysis

In this section detail procedure for static analysis of space frame is described. Material is assumed as elastic. Main aim of linear analysis is to find deformations at each node of space frame and find relevant stresses in each member. Procedure for finding response of space frame is shown below.

3.3.1 Detail Steps for Static Linear Analysis

a. **Identification of Structural Data**

- In first step of analysis, all structural parameters like elastic modulus of elasticity (E) , poisson's ratio (μ), modulus of rigidity (G) and density are defined. Define geometry of space frame in which provide information like coordinates of all nodes (x,y,z) of space frame, assumed section properties (I_x, I_y, I_z, A) of members and their locations. Also define support conditions and their locations. These will be useful for boundary-value problems.

- All nodes are identified by index number. Each member has two joints and they are denoted as j-end and k-end which is nothing but the index number assigned to particular nodes. Corresponding degrees of freedom are denoted by identifiers. This can be understood by Fig. 3.2. These are useful for assembling load vector and overall stiffness matrix.

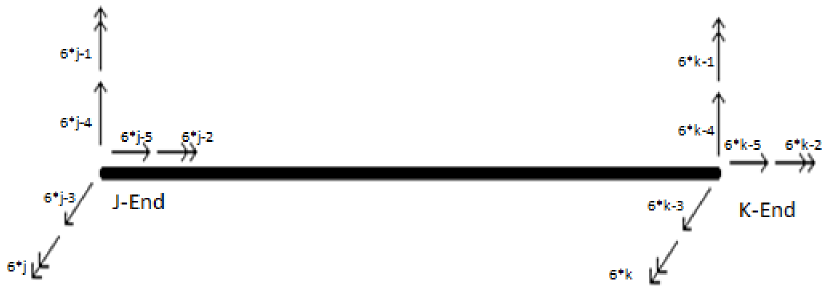

Figure 3.2: Identifier for DOFs at J-end and K-end

b. **Formation of Global Stiffness Matrix**

- To calculate joint stiffness matrix of frame, first find out member stiffness matrix of all members along their local axis using Eq. 3.2 which is depended on the sectional properties of member and material details.

- Find coordinate transformation matrix for all member to convert stiffness matrix from local coordinate system to global coordinate system. It is also known as Rotation Matrix $[R_t]$. Rotation matrix for all space frame member except vertical member is as shown in Eq 3.3

  As shown in Fig. 3.3, Space frame element is considered in space and rotations of α, β and γ are applied to member axes X_m, Y_m and Z_m respectively. Rotation matrix for same is derived to convert member axes to structural axes. X_s, Y_s and Z_s

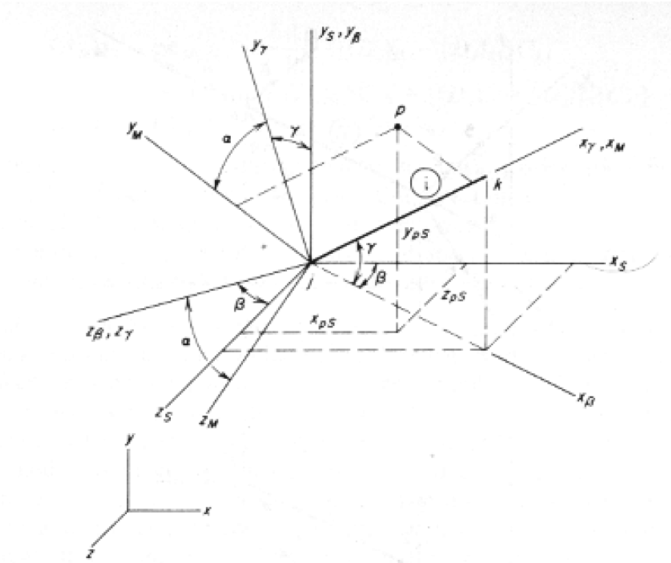

Figure 3.3: Rotation of axes for space frame element

$$[R] = \begin{bmatrix} C_x & C_y & C_z \\ \frac{-C_xC_ycos\alpha - C_zsin\alpha}{C_{xz}} & C_{xz}cos\alpha & \frac{-C_yC_zcos\alpha + C_xsin\alpha}{C_{xz}} \\ \frac{C_xC_ysin\alpha - C_zcos\alpha}{C_{xz}} & -C_{xz}sin\alpha & \frac{C_yC_zsin\alpha + C_xcos\alpha}{C_{xz}} \end{bmatrix} \quad (3.3)$$

For vertical members,

$$[R] = \begin{bmatrix} 0 & C_y & 0 \\ -C_ycos\alpha & 0 & sin\alpha \\ C_ysin\alpha & 0 & cos\alpha \end{bmatrix} \quad (3.4)$$

Now total Rotation matrix $[R_t]$ is written as,

$$[R_t]_{12\times12} = \begin{bmatrix} R_{3\times3} & 0 & 0 & 0 \\ 0 & R_{3\times3} & 0 & 0 \\ 0 & 0 & R_{3\times3} & 0 \\ 0 & 0 & 0 & R_{3\times3} \end{bmatrix} \tag{3.5}$$

Member stiffness matrix with respect to global coordinate system is

$$[S_{ms}]_{12\times12} = [R_t]^T[S_m][R_t] \tag{3.6}$$

- After finding member stiffness matrix in global coordinate system, assemble them to a overall stiffness matrix. In which joint stiffness of members meeting at common node are added.

$$[S_j] = \sum[S_{ms}] \tag{3.7}$$

- Rearrange overall stiffness matrix by incorporating the boundary conditions which includes support conditions.

- This rearranged matrix is always a banded matrix. Hence it is convenient to use banded matrix operation instead of square matrix to solve equilibrium equations.

c. **Formation of Load Vector**

- Define loading conditions for space frame members like concentrated load, uniformly distributed load etc. Find load vector for each members based on provided information for local coordinates. For joint loads, add them directly to load vector based on their locations and directions. $\{A_c\}$

- Covert load vector with respect to local coordinate system to global coordinate system using equation.

$$\{A_{ms}\} = [R_t]^T \{A_c\} \tag{3.8}$$

- Assemble global load vector by adding member load vectors according to their locations.

$$\{A_j\} = \sum \{A_{ms}\} \tag{3.9}$$

d. **Calculation of Results**

- After Constructing the global stiffness and global load vector, equilibrium equation is solved. To reduce time and efforts, solution of equation is calculated using modified Cholesky method. In this method, banded stiffness matrix is factorized into upper triangle and then back substitution is applied to find unknown displacements.

$$[S_j]\{D_s\} = \{A_j\} \tag{3.10}$$

- At the end, displacement response at each free nodes is known. Based on that, member end actions for each member can be calculated by Eq. 3.11.

$$\{A_m\} = [S_m]\{D_m\} \tag{3.11}$$

where,

$$\{D_m\} = [R_t]\{D_g\} \tag{3.12}$$

3.4 Linear Dynamic Analysis of Space Frame

Response of space frame is affected by inertia forces and damping forces induced when dynamic loads are applied. Response of space frame can be found by solving second order simultaneous differential equations. These can be solved using matrix calculations. Equilibrium equation for dynamic condition which is also known as equation of motion, is shown in Eq. 3.13.

$$[M]\{\ddot{u}\} + [C]\{\dot{u}\} + [K]\{u\} = \{F(t)\} \ \ or \ \ [M]\{\ddot{X}_g\} \tag{3.13}$$

where, $[M]\{\ddot{u}\}$ = Inertial Force $[C]\{\dot{u}\}$ = Damping Force $[K]\{u\}$ = Restoring Force

[M],[C] and [K] are respectively mass, damping and stiffness matrices of space frame; $\{\ddot{u}\}$, $\{\dot{u}\}$ and $\{u\}$ are the acceleration, velocity and displacement. F(t) and [M]$\{\ddot{X}_g\}$ are external force vector and force induced by base excitation respectively.

Before proceeding to dynamic analysis, mass matrix and damping matrix for space frame should be derived. Stiffness matrix is calculated using Eq. 3.2.

3.4.1 Mass Matrix for Space Frame Element

Mass matrix of space frame element as shown in Fig. 3.1 can be derived in two ways

- Lumped Mass Matrix
- Consistent Mass Matrix

Mass of space frame element is distributed along its length. The uniformly distributed mass is denoted as $\bar{m}$.

- **Lumped Mass Matrix**

  Lumped mass matrix for space frame member having uniform section is simply diagonal matrix. In lumped mass matrix, mass is equally distributed at two ends. Coefficients of mass matrix corresponding to translations are equal to one half of total mass of element and for torsional displacements, coefficients are equal to polar mass moment of inertia per unit length $(I_{\bar{m}}) = \bar{m}I_x/A$. Whereas for flexural rotations, coefficients are assumed to be zero.

  Lumped Mass Matrix is shown in Eq. 3.14.

$$[M] = \frac{\bar{m}L}{2}\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & \frac{I_x}{A} & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & & & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ & & sym & & & & 1 & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & 1 & 0 & 0 & 0 & 0 \\ & & & & & & & & 1 & 0 & 0 & 0 \\ & & & & & & & & & \frac{I_x}{A} & 0 & 0 \\ & & & & & & & & & & 0 & 0 \\ & & & & & & & & & & & 0 \end{bmatrix} \tag{3.14}$$

- **Consistent Mass Matrix**

In consistent mass matrix, mass of uniform space frame element is distributed according to its displacement functions. This assumption is widely used in analysis. The derivation of consistent mass matrix is based on some mathematical calculations.

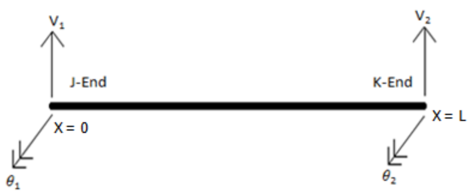

Figure 3.4: Flexural Degrees of freedom and their boundary values

Assume displacement function for flexural behavior of element (Fig. 3.4) is

$$v = c_1 + c_2 y + c_3 y^2 + c_4 y^3 \tag{3.15}$$

Differentiation of Eq. 3.15 gives

$$\theta_y = c_2 + 2c_3 y + 3c_4 y^2 \tag{3.16}$$

Now use boundary conditions for fixed end conditions,

At $x = 0$, $v = v_1$ and $\theta_y = \theta_1$

At $x = L$, $v = v_2$ and $\theta_y = \theta_2$

Put these values in Eq. 3.15 and 3.16 and find value of c_1, c_2, c_3 and c_4. Put all these in Eq. 3.15. This will give displacement function with respect to nodal displacement which is also know as shape function. This can be denoted by,

$$V = \left\{ \begin{array}{cccc} N_1 & N_2 & N_3 & N_4 \end{array} \right\} \left\{ \begin{array}{c} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{array} \right\} \tag{3.17}$$

where,

$N_1(x) = 1 - 3(\frac{x}{L})^2 + 2(\frac{x}{L})^3$

$N_2(x) = x(1 - \frac{x}{L})^2$

$N_3(x) = 3(\frac{x}{L})^2$ - $2(\frac{x}{L})^3$
$N_4(x) = \frac{x^2}{L}(\frac{x}{L}$ - $1)$

By integrating Eq. 3.18, consistent mass matrix for flexural behavior can be calculated,

$$m_{ij} = \int_0^L \bar{m} N_i(x) N_j(x) dx \tag{3.18}$$

Mass matrix for flexural displacements is shown below

$$[M] = \frac{\bar{m}L}{420} \begin{bmatrix} 156 & 22L & 54 & -13L \\ 22L & 4L^2 & 13L & -3L^2 \\ 54 & 13L & 156 & -22L \\ -13L & -3L^2 & -22L & 4L^2 \end{bmatrix} \tag{3.19}$$

Same procedure is applied to flexural behavior with respect to minor axis and same matrix can be found out. For axial and torsional displacement, displacement function is assumed as linear function (refer Fig. 3.5 and 3.6). so that shape function N_1 and N_2 can be derived as

$N_1 = 1$ - $\frac{x}{L}$
and $N_2 = \frac{x}{L}$

and hence using Eq. 3.18, mass matrix for axial displacement and torsional displacement are derived which are shown in Eq. 3.20 and 3.21 respectively.

Figure 3.5: Axial degrees of freedom and their boundary values

Figure 3.6: Torsional degrees of freedom and their boundary values

$$[M]_{axial} = \frac{\bar{m}L}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \tag{3.20}$$

$$[M]_{torsional} = \frac{I_{\bar{m}}L}{6}\begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \tag{3.21}$$

By assembling above Eq. 3.19, 3.20 and 3.21 according to their degrees of freedom, final consistent mass matrix can be formed, Consistent mass matrix for space frame element,

$$[M] = \frac{\bar{m}L}{420} \begin{bmatrix} 140 & 0 & 0 & 0 & 0 & 0 & 70 & 0 & 0 & 0 & 0 & 0 \\ & 156 & 0 & 0 & 0 & 22L & 0 & 54 & 0 & 0 & 0 & -13L \\ & & 156 & 0 & -22L & 0 & 0 & 0 & 54 & 0 & 13L & 0 \\ & & & 140\frac{I_x}{A} & 0 & 0 & 0 & 0 & 0 & 70\frac{I_x}{A} & 0 & 0 \\ & & & & 4L^2 & 0 & 0 & 0 & -13L & 0 & -3L^2 & 0 \\ & & & & & 4L^2 & 0 & 13L & 0 & 0 & 0 & -3L^2 \\ & & sym & & & & 140 & 0 & 0 & 0 & 0 & 0 \\ & & & & & & & 156 & 0 & 0 & 0 & -22L \\ & & & & & & & & 156 & 0 & 22L & 0 \\ & & & & & & & & & 140\frac{I_x}{A} & 0 & 0 \\ & & & & & & & & & & 4L^2 & 0 \\ & & & & & & & & & & & 4L^2 \end{bmatrix} \tag{3.22}$$

3.4.2 Damping Matrix for Space Frame Element

Damping in space frame structure can be approximately calculated by classical damping. In this section, procedure is described to construct classical damping matrix. Rayleigh Damping can be found for space frame having similar damping mechanisms over its height. Classical damping matrix is depended on mass and stiffness of structure.

Classical Damping Matrix

$$[C] = a_0[M] + a_1[K] \tag{3.23}$$

where,

$$a_0 = \xi \frac{2\omega_i \omega_j}{\omega_i + \omega_j} \tag{3.24}$$

$$a_1 = \xi \frac{2}{\omega_i + \omega_j} \tag{3.25}$$

where,

ω_i and ω_j = frequencies of i^{th} and j^{th} mode

ξ = Damping ratio depended on structural material

Classical damping is different for different modes as damping ratio changes. For lower modes of vibration, damping ratio is assumed as constant. For higher modes of vibration, damping ratio should be changed. For space frame analysis, focus is on frame response for lower modes of vibration where damping ratio for concrete material and steel material are 5% and 2% respectively.

3.4.3 Free vibration analysis

Free vibration analysis is useful to find dynamic properties of space frame. Modal frequencies and modal vectors are obtained from free vibration analysis. Equation of motion can be written for undamped free vibration as

$$[M]\{\ddot{u}\} + [K]\{u\} = 0 \tag{3.26}$$

Now assume the solution of Eq. 3.26 as $\{u\} = \{a\}$ sin(ωt - α). Put this in Eq. 3.26 will give

$$-\omega^2[M]\{a\}sin(\omega t - \alpha) + [K]\{a\}sin(\omega t - \alpha) = \{0\} \tag{3.27}$$

rearranging above equation gives,

$$[[K] - \omega^2[M]]\{a\} = \{0\} \tag{3.28}$$

Above equation is true when determinant of matrix $[[K]$ - $\omega^2[M]]$ is equal to zero. The expansion of this equation results in polynomial equation of degree n. Solution of this equation gives ω_i^2 where, i = 1,2,...,n. where n is the number of modes considered. Solution of this equation can be found out by Eigen value method. Eigen values give square of natural frequencies of different modes (ω_i^2) and Eigen vectors give mode shapes of different modes.

3.4.3.1 Analysis Procedure

In this section, analysis steps for free vibration of analysis of space frame is listed.

- **Identification of Structural Data**

  Define all information related to material properties and cross-sectional properties. Define mass per unit length for all members. Also define coordinates of each node of space frame.

- **Construction of Stiffness matrix and Mass Matrix**

  Calculate member stiffness matrices and assemble to overall stiffness matrix. Member mass matrices are also calculated based on lumped mass or consistent mass. Same procedure described in Linear static analysis for assembling stiffness and mass matrix is used.

- **Solution of Characteristic equation det[[M] - ω^2[K]] = 0**

  Natural frequencies and corresponding mode shapes are calculated using inverse iteration method.

3.4.4 Time-History Analysis of Space Frame

Response of space frame in form of displacements, velocity and acceleration is required for designing any frame for its extreme conditions. Time history analysis is useful for designing earthquake resistant structure, machine foundation, oil and gas plant structures. Method is capable of providing response curves at each floor level which can be used in industrial building design where each floor contains different services and has different safety requirements.

For forced vibrations or base excitation conditions, time history plots are developed by time stepped methods. Different numerical integration methods for time stepped analysis are listed below:

a. Newmark Beta Method

b. Wilson Theta Methd

c. Runge-Kutta Method

d. Houbolt Method

All these methods are incremental time stepped methods for solving equation of motion as shown in Eq. 3.13. Among all these methods, Wilson-Theta method is conditionally stable. Only requirement for accurate result is proper selection of value of theta (θ). Wilson-Theta method is acceleration based method. Basic assumption of Wilson-Theta method is that acceleration varies linearly over the time interval from t to (t + $\theta\Delta$t). Compared to Newmark Beta method, linear acceleration variation is extended for time (t + $\theta\Delta$t) instead of (t + Δt).

3.4.4.1 Wilson Theta Method

Incremental equation of motion for step-by-step linear acceleration method with extended time (t$_i$ + τ) where τ = $\theta\Delta$t can be written as, [18]

$$[M]\widehat{\{\Delta\ddot{u}_i\}} + [C]\widehat{\{\Delta\dot{u}_i\}} + [K]\widehat{\{\Delta u_i\}} = \widehat{\{\Delta F_i\}} \tag{3.29}$$

Incremental quantities can be defined for i^{th} time step as,

$$\begin{aligned}
\widehat{\{\Delta\ddot{u}_i\}} &= \ddot{u}(t_i + \tau) - \ddot{u}(t_i) \\
\widehat{\{\Delta\dot{u}_i\}} &= \dot{u}(t_i + \tau) - \dot{u}(t_i) \\
\widehat{\{\Delta u_i\}} &= u(t_i + \tau) - u(t_i) \\
\widehat{\{\Delta F_i\}} &= F(t_i + \tau) - F(t_i)
\end{aligned} \tag{3.30}$$

Assumption for linear variation is shown in Fig. 3.7.
Linear expression for acceleration at time t can be defined as,

$$\ddot{u}(t) = \ddot{u}_i + \frac{\widehat{\Delta\ddot{u}_i}}{\tau}(t - t_i) \tag{3.31}$$

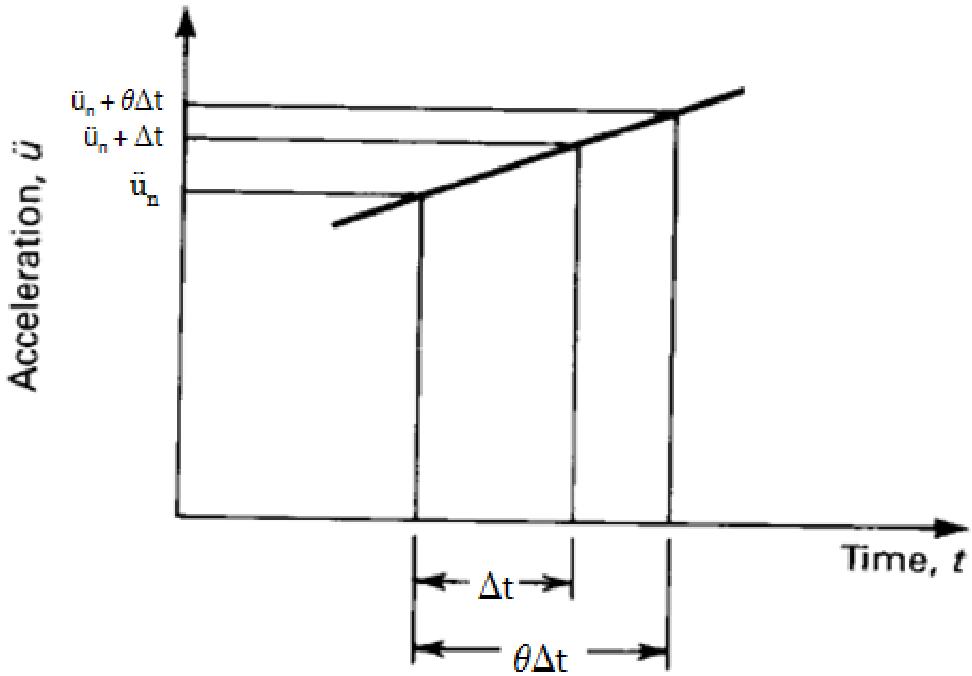

Figure 3.7: Assumption of Linear Variation of acceleration

Integrating Eq. 3.31 between limits t_i and t gives velocity and again integrating Eq. 3.31 gives displacement.

$$\dot{u}(t) = \dot{u}_i + \ddot{u}_i(t - t_i) + \frac{\widehat{\Delta\ddot{u}_i}}{2\tau}(t - t_i)^2 \tag{3.32}$$

$$u(t) = u_i + \dot{u}_i(t - t_i) + \frac{1}{2}\ddot{u}_i(t - t_i)^2 + \frac{\widehat{\Delta\ddot{u}_i}}{6\tau}(t - t_i)^3 \tag{3.33}$$

Putting value of t = $t_i + \tau$ in Eq. 3.32 and 3.33, gives incremental quantities

$$\widehat{\Delta\dot{u}_i} = \ddot{u}_i\tau + \frac{1}{2}\widehat{\Delta\ddot{u}_i}\tau \tag{3.34}$$

and

$$\widehat{\Delta u_i} = \dot{u}_i\tau + \frac{1}{2}\ddot{u}_i\tau^2 + \frac{1}{6}\widehat{\Delta\ddot{u}_i}\tau^2 \tag{3.35}$$

Now make equation in terms of $\widehat{\Delta\ddot{u}_i}$ from equation 3.35 and put it in Eq. 3.34, will give incremental quantities $\widehat{\Delta\ddot{u}_i}$ and $\widehat{\Delta\dot{u}_i}$. Both equations have only one unknown quantity which is incremental displacement ($\widehat{\Delta u_i}$) for time increment $(t_i + \tau)$.

$$\widehat{\Delta\ddot{u}_i} = \frac{6}{\tau^2}\widehat{\Delta u_i} - \frac{6}{\tau}\dot{u}_i - 3\ddot{u}_i \tag{3.36}$$

$$\widehat{\Delta\dot{u}_i} = \frac{3}{\tau}\widehat{\Delta u_i} - 3\dot{u}_i - \frac{\tau}{2}\ddot{u}_i \tag{3.37}$$

Put above two equations in Eq. 3.29, will form linear equations with only unknown $\widehat{\Delta u_i}$. Compare it with equation

$$[\overline{K_i}]\{\widehat{\Delta u_i}\} = \{\overline{\Delta F_i}\} \tag{3.38}$$

where,

$$\overline{K_i} = [K] + \frac{6}{\tau^2}[M] + \frac{3}{\tau}[C] \tag{3.39}$$

and

$$\overline{\Delta F_i} = \widehat{\Delta F_i} + [M](\frac{6}{\tau}\dot{u}_i + 3\ddot{u}_i) + [C](3\dot{u}_i + \frac{\tau}{2}\ddot{u}_i) \tag{3.40}$$

After finding $\widehat{\Delta u_i}$, all other incremental quantities can be calculated for time increment (t + τ). Based on extended acceleration, acceleration increment for (t + Δ t) can be derived. Steps for finding response quantities for space frame are discussed in next sections.

Wilson-Theta Method is conditionally stable if theta (θ) $\geq$ 1.38. Numerical stability is depended on results of direct integration method. For $\theta \geq 1.38$, any value of Δt will give stable results. For accuracy of results, Δt value should be small.

3.4.4.2 Analysis Procedure

a. **Identification of Space frame data and Construction of Stiffness, Mass and Damping matrix**

   Based on the information provided, construct member stiffness matrix, consistent mass matrix and damping matrix as defined in previous sections. Assemble all matrices and rearrange them by incorporating boundary conditions. Make banded matrices for stiffness, mass and damping.

b. **Initialization**

   Read initial vectors of displacements $\{u_0\}$, velocity $\{\dot{u}_0\}$ and forces $\{F_0\}$ at each node of space frame for all degrees of freedom. Calculate initial acceleration based on following equation.

$$[M]\{\ddot{u}_0\} = \{F_0\} - [C]\{\dot{u}_0\} - [K]\{u_0\} \tag{3.41}$$

c. **Define values of θ and Δt**

Select $\theta \geq 1.38$ and value of $\Delta t \leq \frac{T_{min}}{\Pi}$. where, T_{min} = smallest time period of the system considered.
Calculate constant values,

$\tau = \theta \Delta t$, $a_1 = \frac{3}{\tau}$, $a_2 = \frac{6}{\tau}$

$a_3 = \frac{\tau}{2}$ and $a_4 = \frac{6}{\tau^2}$

d. **Formation of effective stiffness matrix**

Form effective stiffness matrix using following equation,

$$\overline{[K]} = [K] + a_4[M] + a_1[C] \tag{3.42}$$

e. **Formation of effective load vector**

Form incremental load vector for i^{th} step, by linear interpolating load for time interval t_i and $(t_i + \tau)$. Here load increment is considered for extended time period. $(\theta \Delta t)$.

$$\{\widehat{\Delta F_i}\} = \{F_{i+1}\} + (\{F_{i+2}\} - \{F_{i+1}\})(\theta - 1) - \{F_i\} \tag{3.43}$$

where,

$F(t)_i = F(i * \Delta t)$ for i^{th} time increment.

Find increment load vector $\overline{\widehat{\Delta F_i}}$,

$$\{\overline{\widehat{\Delta F_i}}\} = \{\widehat{\Delta F_i}\} + (a_2[M] + 3[C])\{\dot{u}_i\} + (3[M] + a_3[C])\{\ddot{u}_i\} \tag{3.44}$$

where,

$\{\dot{u}_i\}$ = velocity vector for i^{th} time step at each free nodes of space frame $\{\ddot{u}_i\}$ = acceleration vector for i^{th} time step at each free nodes of space frame

f. **Solution for incremental displacement vector $\{\widehat{\Delta u_i}\}$ for extended time period**

Solve following equation by modified Cholesky method for $\{\widehat{\Delta u_i}\}$.

$$\overline{[K]}\{\widehat{\Delta u_i}\} = \{\overline{\Delta F_i}\} \tag{3.45}$$

g. **Calculation of incremental acceleration $\{\widehat{\Delta \ddot{u}_i}\}$ for extended time period**

$$\{\widehat{\Delta \ddot{u}_i}\} = a_4\{\widehat{\Delta u_i}\} - a_2\{\dot{u}_i\} - 3\{\ddot{u}_i\} \tag{3.46}$$

h. **Find incremental quantities for increment of Δt**

Linear variation of acceleration is assumed for extended time period. Using this assumption, incremental acceleration is calculated by,

$$\{\Delta\ddot{u}_i\} = \frac{\widehat{\{\Delta\ddot{u}_i\}}}{\theta} \tag{3.47}$$

Other incremental quantities are,

$$\{\Delta\dot{u}_i\} = \{\ddot{u}_i\}\Delta t + \frac{1}{2}\{\Delta\ddot{u}_i\}\Delta t \tag{3.48}$$

$$\{\Delta u_i\} = \{\dot{u}_i\}\Delta t + \frac{1}{2}\{\ddot{u}_i\}\Delta t^2 + \frac{1}{6}\{\Delta\ddot{u}_i\}\Delta t^2 \tag{3.49}$$

i. **Find response quantities at time** (t_{i+1})

$$\begin{aligned} \{u_{i+1}\} &= \{u_i\} + \{\Delta u_i\} \\ \{\dot{u}_{i+1}\} &= \{\dot{u}_i\} + \{\Delta\dot{u}_i\} \\ \{\ddot{u}_{i+1}\} &= \{\ddot{u}_i\} + \{\Delta\ddot{u}_i\} \end{aligned} \tag{3.50}$$

Repeat steps from (e) to (i) for each increments. where value of i = 0,1,2,... Results of analysis gives response of space frame for each time steps. Using those, time history curve can be developed.

3.5 Computer Program Development

Computer programs for linear analysis are developed for fast computations. These programs are also useful to develop nonlinear analysis program. Each analysis program consists of three major steps

- Pre processing
- Processing
- Post Processing

In pre processing step, input files are generated based on user defined data. This is important for designing the actual analysis problem. Same data are used in second step. In processing step, analysis is performed based on developed procedures. After performing whole analysis, desire results are obtained and output file is generated in last step called post processing.

3.5.1 Static Linear Analysis

Computer program is developed based on the procedure discussed in previous sections. In first step of analysis, input file is generated based on user defined data. Input file consists of geometry of space frame, material properties, cross-sectional properties and boundary conditions. Analysis steps are performed in second step. In that section, stiffness matrices for each structural members are calculated and analysis is performed. At the end of analysis, displacements at each node are found out. Based on these displacements, member end actions are calculated and presented in output files. Base reactions are also calculated. Flow chart for developed computer program is shown in Fig. 3.8.

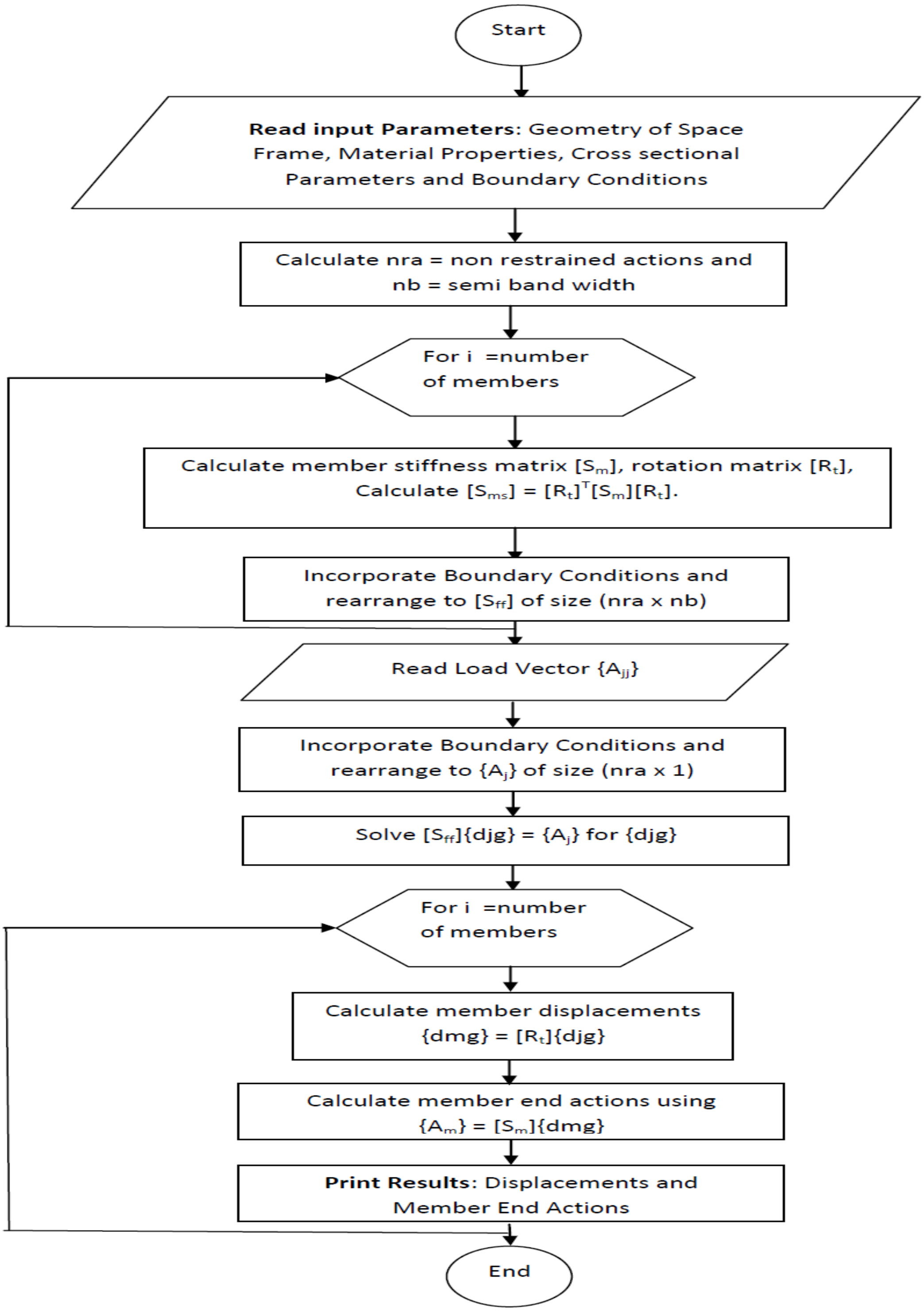

Figure 3.8: Flow Chart : Static Linear Analysis

3.5.2 Dynamic Analysis

Dynamic analysis consists of two part. Programs are developed for both the cases.

- Free Vibration Analysis
- Forced Vibration Analysis

3.5.2.1 Free Vibration Analysis

Free vibration analysis is useful to find out natural frequencies and mode shapes of space frame which will be useful for forced vibration. First step of program is to develop input files which includes geometry of space frame, material properties, cross-sectional properties, mass per unit length for each member and boundary conditions. Based on those data, consistent mass matrices and stiffness matrices are developed in core program. Analysis is performed based on Inverse Iteration Method. Output file is generated in terms of natural frequencies and mode shapes for desire modes. Flow of computer program is shown in Fig 3.9.

3.5.2.2 Forced Vibration Analysis

Forced Vibration Analysis is performed based on Wilson-Theta method. Analysis procedure is discussed in previous sections. Input parameters are generated which is same as for free vibration analysis. In core program, stiffness matrices and consistent mass matrices are calculated. Based on those matrices, Rayleigh damping matrix is calculated. Forcing function is to be developed based on user defined data. Time increment method is used. Responses are calculated for each time increment. As a output, time history responses at desire node for desire degree of freedom are recorded. Output file is generated. Based on those data, Time history plots can be developed. Flow of program is shown in Fig. 3.10 and 3.11.

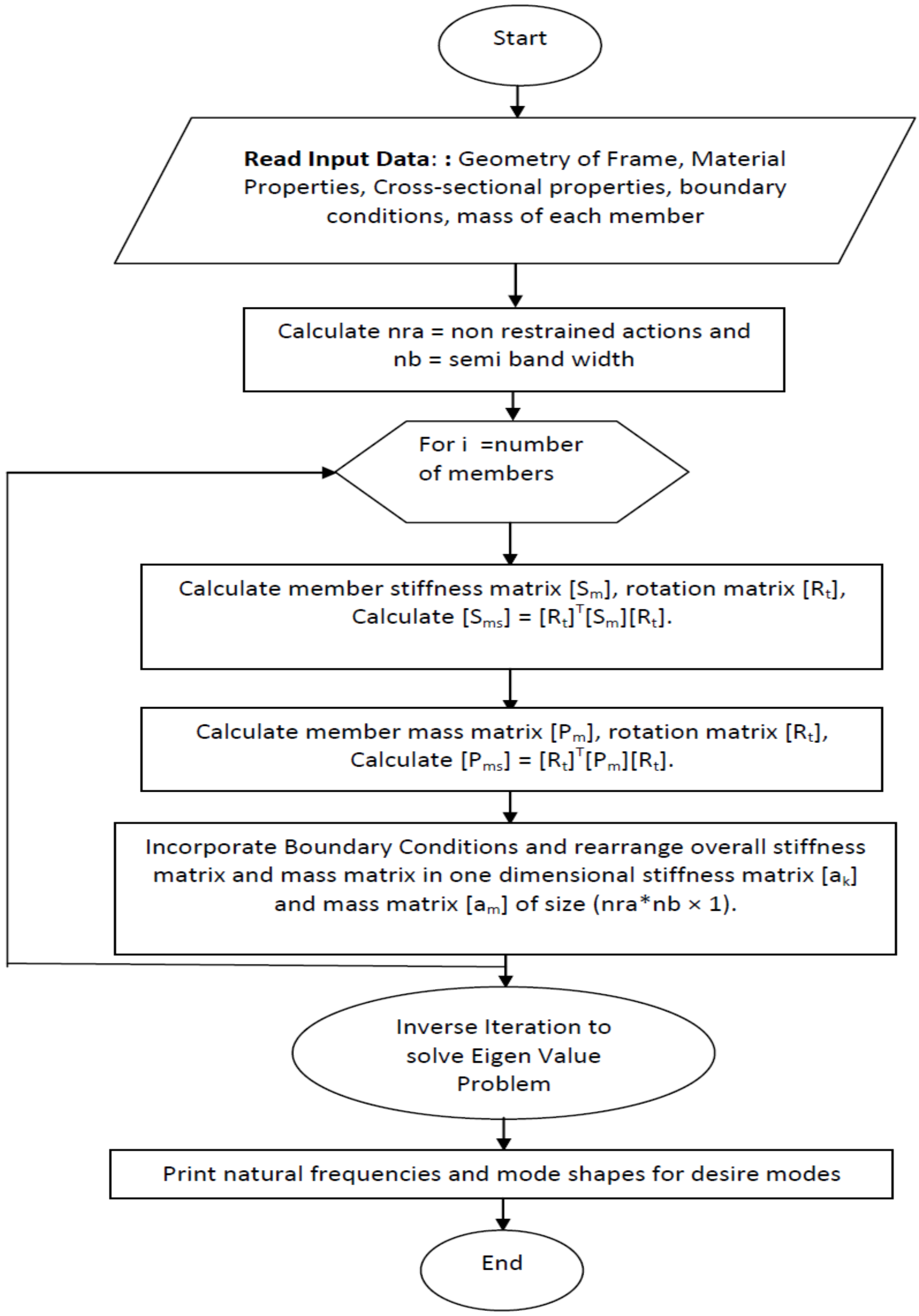

Figure 3.9: Flow Chart : Free Vibration Analysis

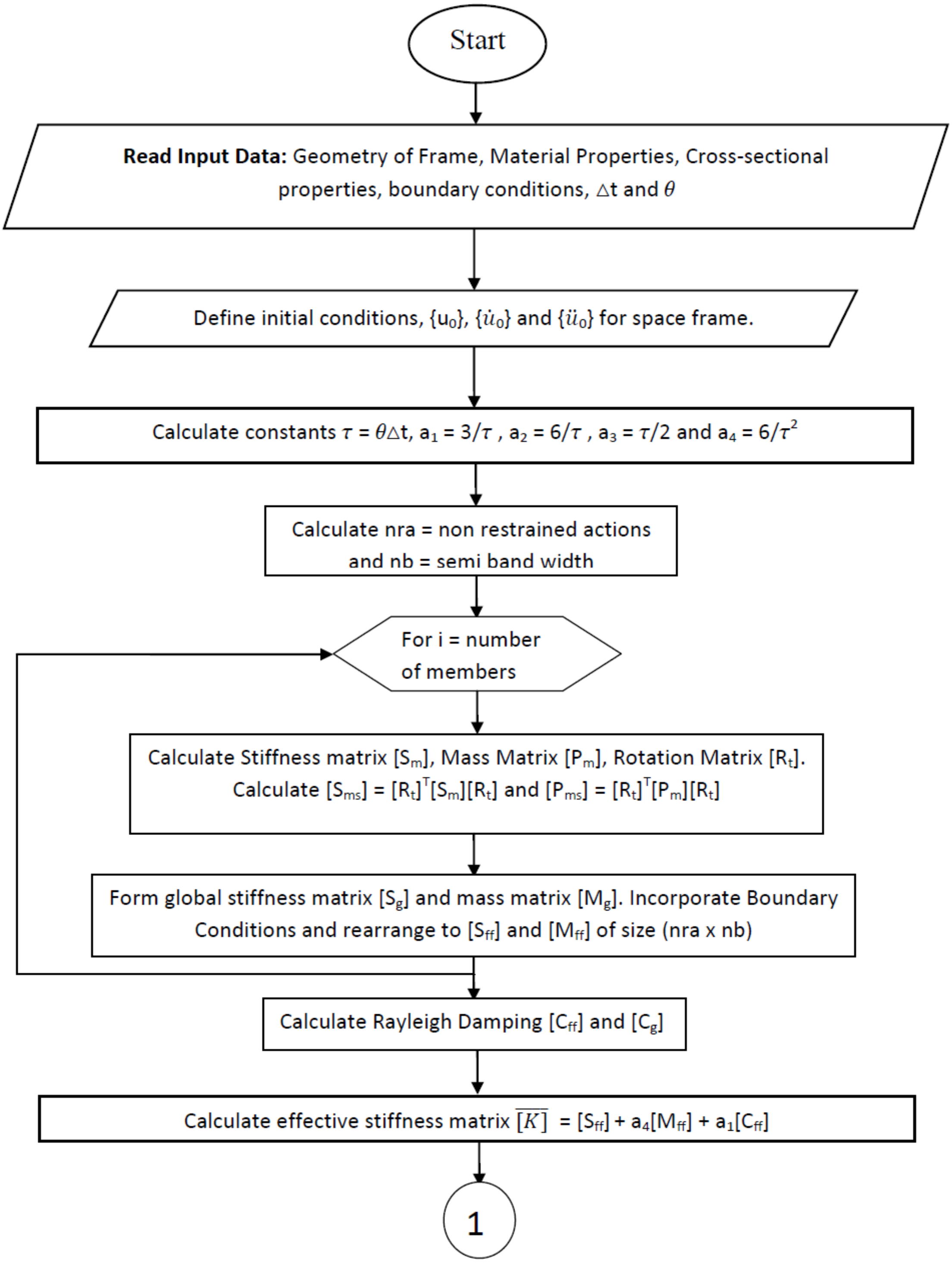

Figure 3.10: Flow Chart : Forced Vibration Analysis

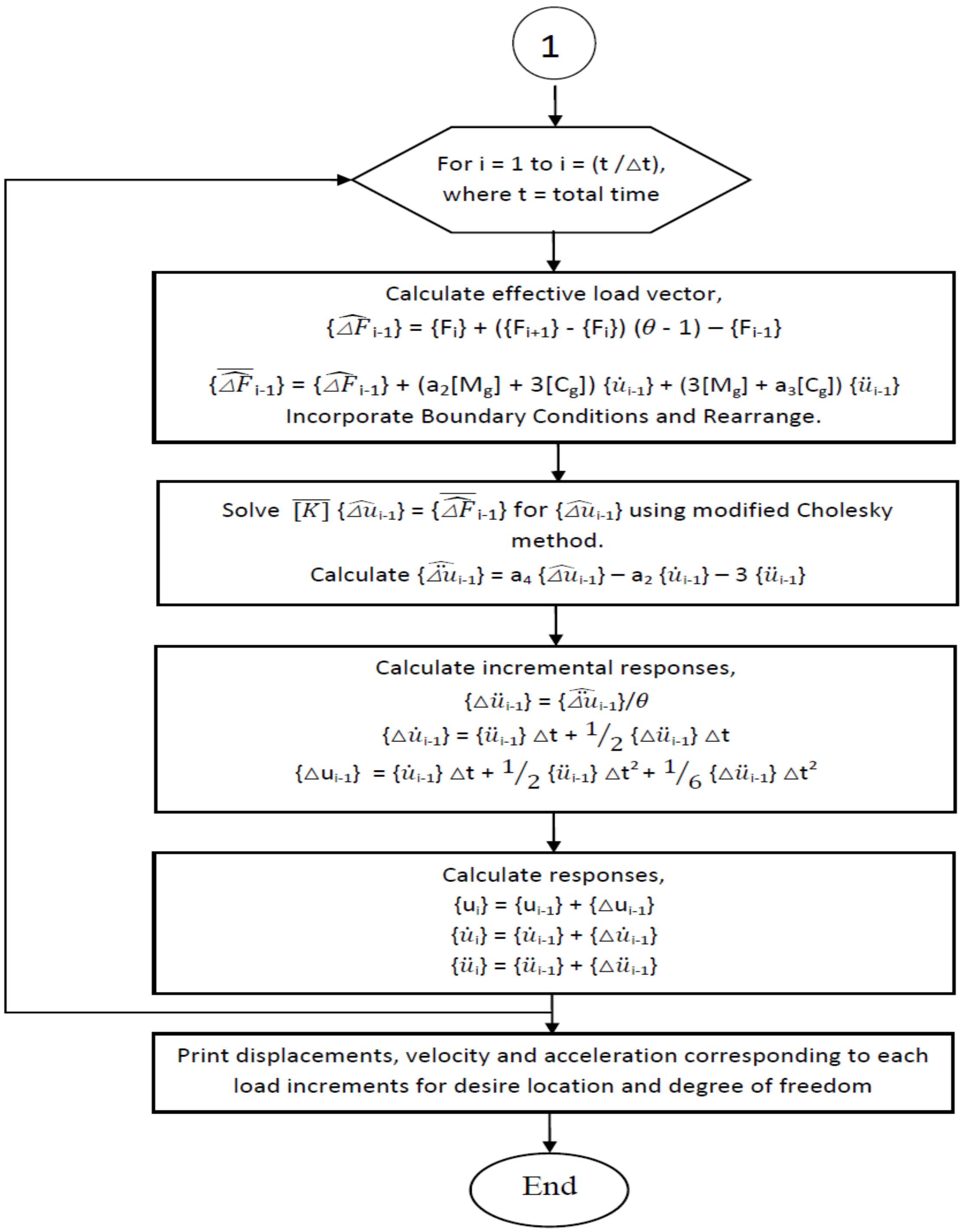

Figure 3.11: Flow Chart : Forced Vibration Analysis (Continued)

3.6 Summary

In this chapter, detail procedures for linear static and dynamic analysis are discussed. Dynamic linear analysis involves free vibration and forced vibration of multi degrees of freedom. Computer programs are developed based on these method. Flow charts of developed computer programs are presented. All these computer oriented methods can be extended for nonlinear analysis by incorporating incremental and iteration methods. Linear analysis methods provide base for formation of nonlinear analysis procedure hence it is required to understand each steps of analysis.

Chapter 4

Linear Analysis : Results and Discussions

4.1 General

Linear analysis results give good approximation of displacements for given loading conditions. Also results of different loading conditions can be easily superimposed. Linear analysis procedure is also useful to established nonlinear analysis steps. In this chapter, linear analysis of space frame for both static and dynamic loading conditions are performed. Developed computer programs are used to analyze space frame which will be used to prepare nonlinear analysis program. Results of linear analysis are presented for various space frames. Results of displacements for static linear, time period of different modes and time history responses plots are presented in this chapter. Validation of results is done by comparing results with ETABS results.

4.2 Results and Discussion

4.2.1 Single bay Single Storey Frame

Single bay single storey frame shown in Fig. 4.1 for static and dynamic loading conditions. Main purpose of analysis is to validate computer oriented analysis procedure.

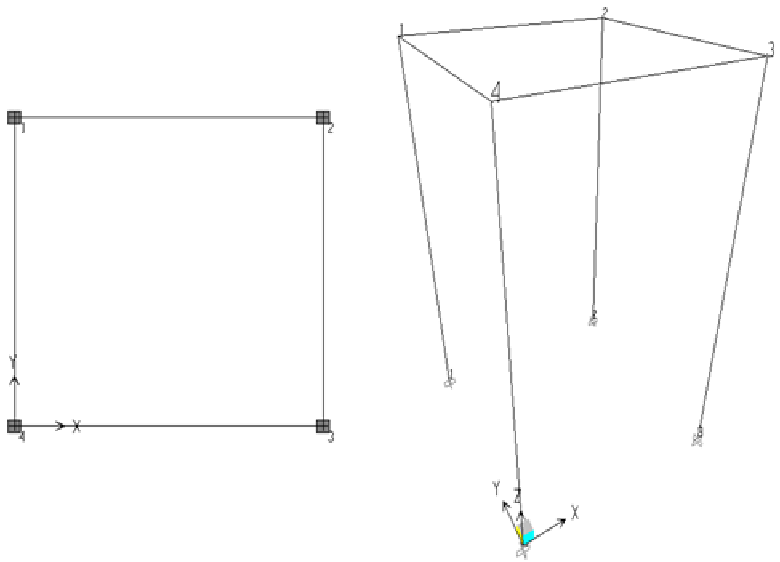

Figure 4.1: Plan and 3-D view of space frame

Space Frame Data

Plan Dimension : 5m×5m

Storey Height : 8m

Beam and Column Size : (200×200) mm

Grade of Concrete : M25

Poisson's Ratio (μ) : 0.2

Support Condition : Fixed

4.2.1.1 Static Analysis Results

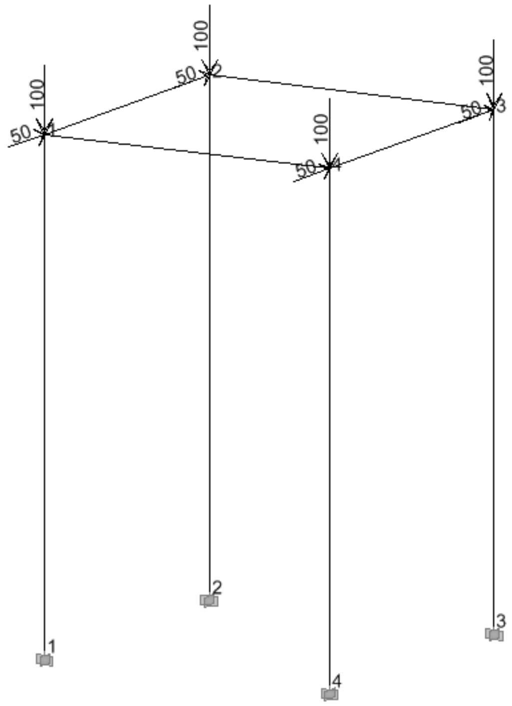

Figure 4.2: Static Load Case

Here self weight of members are neglected. Loads are applied on each joints of first storey. Program and ETABS results in terms of displacement are shown in Table 4.1. Displacement of each node of first storey are same due to symmetric loading.

Table 4.1: Static Linear Analysis Result

Displacements at node 3	Program Results	ETABS Results
$\mathbf{D}_x$	824 mm	823.9 mm

4.2.1.2 Dynamic Analysis

- **Parameters of dynamic analysis**

Damping Ratio ξ = 5 %

For Time History Analysis,

Time dependent Force : F(t) = 50 sin(t) at node 3 for t = 0 to t = 30 s (Fig. 4.3)

Time Increment : t = 0.1 second

$\theta = 1.5$

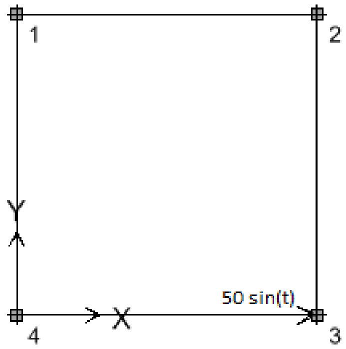

Figure 4.3: Dynamic Load Case

- **Free Vibration Analysis**

Table 4.2: Results of Natural Time Period

	Program Result	**ETABS Result**
Time Period (second)	0.7117	0.764

- **Forced Vibration Analysis**

  Response of space frame is calculated in terms of displacement, velocity and acceleration. Time history plots are prepared in Fig. 4.4 and 4.5 based on Program and ETABS results.

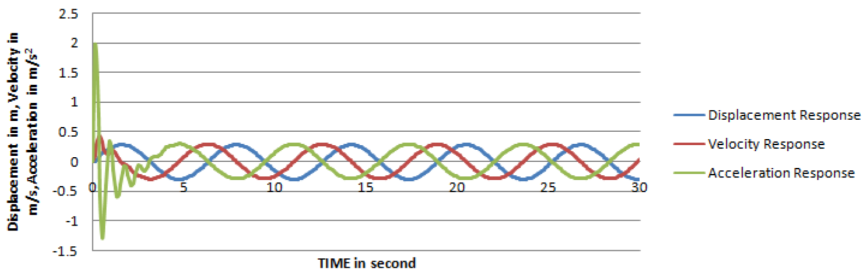

Figure 4.4: Program Result of time history analysis

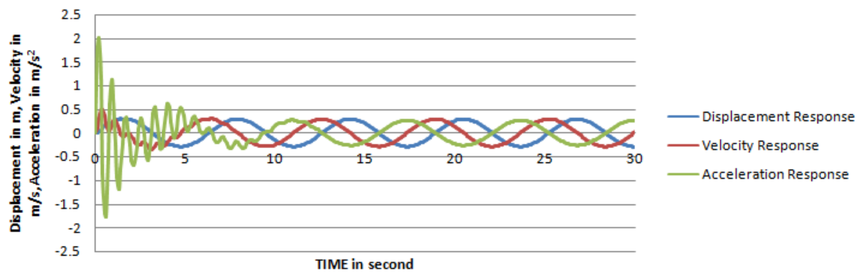

Figure 4.5: ETABS Result of time history analysis

Results of time history analysis obtained from program are presented in terms of time history curve as shown in Fig. 4.4. Same frame is modeled in ETABS and analyzed. Results of program are having good agreement with ETABS results. Also time period value is matched with ETABS results. Maximum responses are presented in Table 4.3 which is also verified with ETABS results.

Table 4.3: Maximum Response of displacement, velocity and acceleration

		Program Results	**ETABS Results**
Displacement (m)	+ ve	0.283 m at t = 1.5 s	0.292 m at t = 1.4 s
	-ve	-0.300 m at t = 4.7 s	-0.294 m at t = 1.4 s
Velocity (m/s)	+ ve	0.424 m/sec at t = 0.4 s	0.490 m/sec at t = 0.4 s
	-ve	-0.296 m/sec at t = 3.1 s	-0.351 m/sec at t = 3.1 s
Acceleration (m/s^2)	+ ve	1.965 m/s^2 at t = 0.2 s	2.008 m/s^2 at t=0.3 s
	-ve	-1.275 m/s^2 at t = 0.5 s	-1.768 m/s^2 at t = 0.6 s

4.2.2 Multi-storey space frames

Various space frames are analyzed using computer program. Results of computer programs are compared with ETABS results. Analysis is performed for both static and dynamic loading conditions. Data of space frames like plan of space frame (Fig. 4.6), storey height, beam dimensions, slab and wall thickness etc, earthquake parameters are kept similar to compare results with each other. Various data like number of storeys, column sizes are varying for each model.

Common data of space frames

- Plan Dimension : (15×20) m
- Storey Height : 5 m
- Beam Size : (300×450) mm
- Zone : V
- Soft soil condition
- I = 1 and R = 5
- Fixed at base

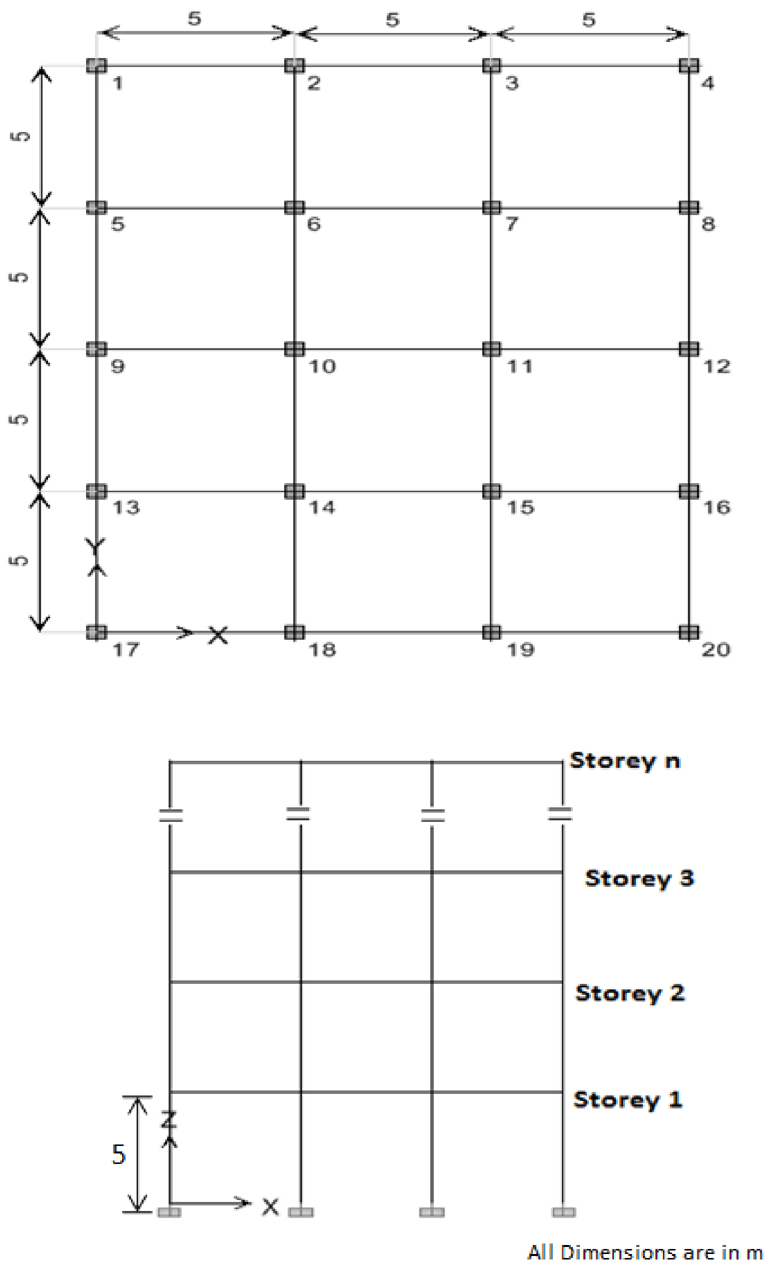

Figure 4.6: Plan and Elevation of multi-storey space frames (Common for all models)

Column sizes and base shear of different models of multi-storey frames are shown in Table 4.4.

Table 4.4: Column sizes and Base shear for Space Frames

Model Name	Column Size	Grade of Concrete	Base Shear (Vbx) (kN)
6 storey	(450×450)	M30	3425 kN
10 storey	(600×600)	M30	2955.2 kN
15 storey	(750×750)	M30	3890.8 kN
20 storey	(1000×1000)	M40	4456.7 kN
25 storey	(1100×1100)	M40	4456.7 kN
30 storey	(1200×1200)	M40	4728.3 kN

Static Load Data

Self weight of each elements are found out and distributed at each nodes of frame. Earthquake in x direction is considered and based on assumed parameters of site conditions, base shear (V_{bx}) is calculated for various frames and value of base shear is shown in Table 4.4.

Lateral loads are distributed to each storey as per IS 1893:2002 and these are applied to each nodes of storey depending on contributory area.

Based on joint loads for gravity and earthquake conditions, load vector is formed which is used in static analysis. Steps for formation of load vector is presented.

Formation of Load Vector

a. Self weight of each elements of frame are calculated at each node of frame for all storeys. To consider worst case, only dead weight of slabs (130 mm thick), walls (230 mm thk.) on all beams, beams and columns are calculated. Live Load = 5 kN/m^2 and Floor Finish = 1 kN/m^2

b. Self weight of each storeys are distributed among all nodes of that storey.

c. Earthquake loads are considered. Base shear (V_{bx}) is calculated for soft soil strata and earthquake zone V.

$V_{bx} = A_h W$, where W = total weight of building and $A_h = \frac{Z}{2} \frac{I}{R} \frac{S_a}{g}$.

d. Based on storey weight, lateral forces are calculated (Q_i).

$Q_i = V_{bx} \frac{W_i H_i^2}{\sum W_i H_i^2}$

e. Lateral force for each storey is distributed among all nodes based on weight ratio of that node. Hence, lateral force is distributed to all nodes.

f. At the end, load vectors are formed based on self weight and lateral load for each nodes. This is used as input parameter for analysis.

Same procedure is followed for each frame model to calculate forces at each node.

4.2.2.1 Static linear Analysis Results

Analysis is performed for static loads are analysis results are compared with ETABS results. Results of displacements in X-direction for node number 20 (Refer Fig. 4.6) of top storey of each model are compared with that of ETABS results and presented in Table 4.5.

Table 4.5: Results of displacement of node 20 of top storey in X direction

Model Name	**Program Result**	**ETABS Result**
6 storey	0.255 m	0.261 m
10 storey	0.279 m	0.288 m
15 storey	0.509 m	0.527 m
20 storey	0.613 m	0.664 m
25 storey	0.812 m	0.815 m
30 storey	1.023 m	1.105 m

. Variation between results of displacements are about 2% - 5% which are in accepted limit.

4.2.2.2 Dynamic linear Analysis Results

1. Free Vibration Analysis Results

Free vibration analysis are performed to find out dynamic properties of space frames which will be used in forced vibration analysis. In free vibration analysis, consistent mass matrix is used. Free vibration results for 6 storey, 10 storey, 15 storey, 20 storey, 25 storey and 30 storey are shown in Tabular form. Results of free vibration are compared with ETABS results.

Table 4.6: Time Periods of different modes of 6 storey Frame

Mode	**Time Period (sec)**	
	Program Results	**ETABS results**
1 - X	1.039	1.013
1-Y	1.017	0.998
1-Torsion	0.943	0.893
2-X	0.387	0.337
2-Y	0.333	0.333
2-Torsion	0.326	0.299

Table 4.7: Time Periods of different modes of 10 storey Frame

Mode	**Time Period (sec)**	
	Program Results	**ETABS results**
1 - X	1.739	2.219
1-Y	1.687	2.152
1-Torsion	1.536	1.882
2-X	0.552	0.705
2-Y	0.537	0.686
2-Torsion	0.495	0.605
3-X	0.481	0.389
3-Y	0.367	0.381
3-Torsion	0.360	0.340

Results of free vibration analysis for 6 storey frame and 10 storey frame are shown in Table 4.6 and 4.7. Results are compared with ETABS results. Program results are differing from ETABS results for lower modes by 20% to 30% whereas for higher modes, results are verified as variation between two results are approximately 3% - 6%. Possible reasons for difference between two results are distribution of mass at various nodes and consideration of DOFs for program and ETABS.

Table 4.8: Time Periods of different modes of 15 storey Frame

Mode	**Time Period (sec)**	
	Program Results	**ETABS results**
1 - X	2.766	3.316
1-Y	2.670	3.203
1-Torsion	2.306	2.657
2-X	0.873	1.048
2-Y	0.847	1.017
2-Torsion	0.748	0.855
3-X	0.555	0.572
3-Y	0.476	0.559
3-Torsion	0.465	0.482

Table 4.9: Time Periods of different modes of 20 storey Frame

Mode	**Time Period (sec)**	
	Program Results	**ETABS results**
1-X	4.060	5.899
1-Y	3.918	5.667
1-Torsion	2.920	3.983
2-X	1.248	1.846
2-Y	1.212	1.785
2-Torsion	0.962	1.291
3-X	0.655	0.992
3-Y	0.641	0.968
3-Torsion	0.637	0.735

Results of free vibration for 15 storey frame and 20 storey frame are shown in Table 4.8 and 4.9. Results of three modes are compared with ETABS results. Lower mode results are differing from ETABS by 16% - 32%. For other modes, results are verified. Possible reasons for difference between two results are distribution of mass at various nodes and consideration of DOFs for program and ETABS.

Table 4.10: Time Periods of different modes of 25 storey Frame

Mode	**Time Period (sec)**	
	Program Results	**ETABS results**
1-X	5.540	6.081
1-Y	5.333	5.855
1-Torsion	3.629	3.823
2-X	1.705	1.872
2-Y	1.653	1.815
2-Torsion	1.214	1.233
3-X	0.895	0.984
3-Y	0.876	0.963
3-Torsion	0.720	0.697

Table 4.11: Time Periods of different modes of 30 storey Frame

Mode	**Time Period (sec)**	
	Program Results	**ETABS results**
1-X	7.210	7.785
1-Y	6.920	7.478
1-Torsion	4.274	4.429
2-X	2.215	2.393
2-Y	2.144	2.316
2-Torsion	1.446	1.437
3-X	1.161	1.255
3-Y	1.135	1.228
3-Torsion	0.867	0.821

Free vibration analysis is performed for 25 storey and 30 storey frames. Results are shown in Table 4.10 and 4.11. Observed variation in results of program and ETABS are about 4% - 8%. Possible reasons for difference between two results are distribution of mass at various nodes and consideration of DOFs for program and ETABS. Results of free vibration are used in forced vibration.

2. Forced Vibration Analysis Results

In forced vibration analysis, damping effects are also included. Response quantities are calculated using Wilson theta method.
Where,

- Forcing function : F(t) = 500 sin(t) for t = 0 to t = 15 second at corner nodes 1,4,17 and 20 of top storey
- $\theta = 1.5$
- Time increment, $\Delta t = 0.05$ second
- ω_i = Natural frequency of first mode in X direction for each model
- $\xi = 5\%$

Response quantities for node 20 of Top Storey of each model are recorded.

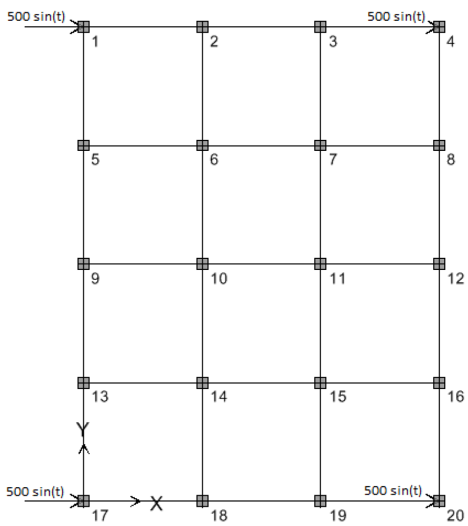

Figure 4.7: Magnitude and direction of time dependent Force

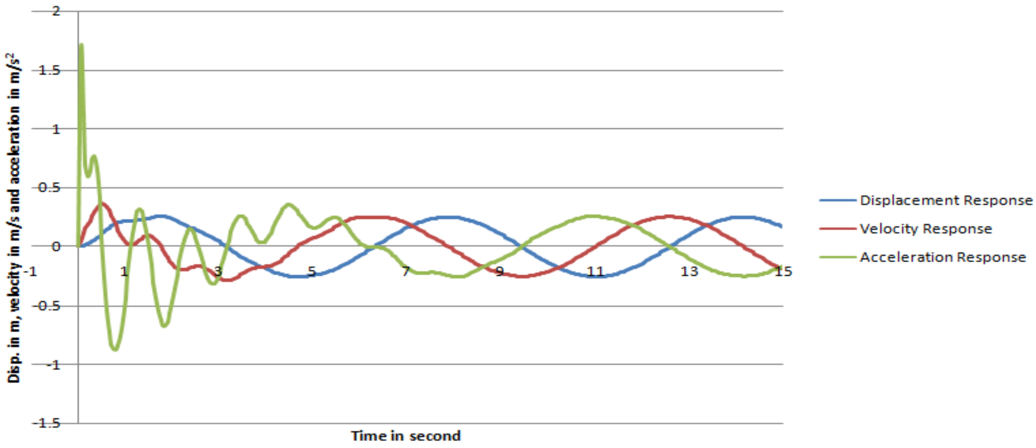

Figure 4.8: Program Results : Time History Analysis of 6 Storey Frame

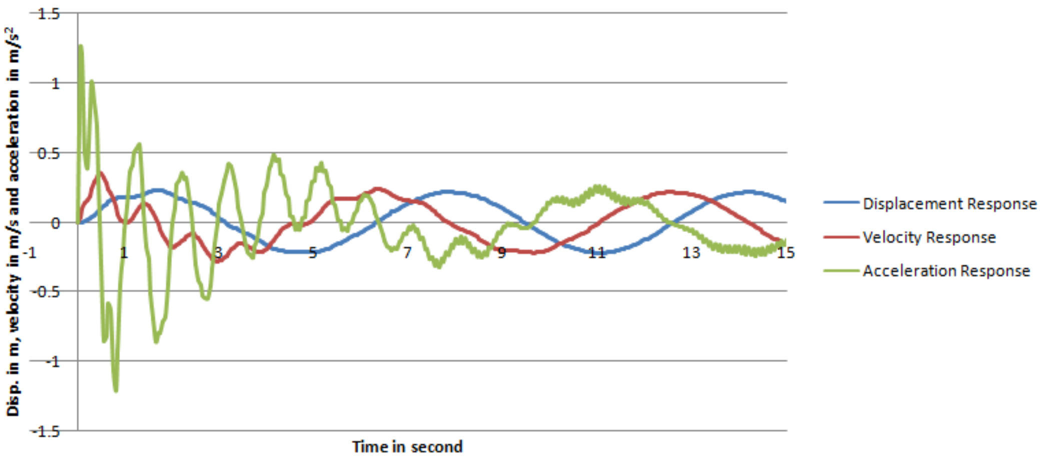

Figure 4.9: ETABS Results : Time History Analysis of 6 Storey Frame

Table 4.12: Maximum Response quantities for 6 Storey Frame

Maximum Responses		**Program Results**	**ETABS Results**
Displacement (m)	"+ ve"	0.254	0.231
	"- ve"	-0.258	-0.218
Velocity (m/s)	"+ ve"	0.365	0.351
	"- ve"	-0.289	-0.278
Acceleration (m/s^2)	"+ ve"	1.675	1.262
	"- ve"	-0.875	-1.206

Dynamic linear analysis is performed for 6 Storey frame subjected to time dependent force using developed computer program. Time history of responses like displacements, velocity and accelerations are calculated and recorded for top node. Results are shown in Fig. 4.8. Same frame is also analyzed in ETABS. Results of ETABS are shown in Fig. 4.9. Results of developed program are in good agreement with ETABS results. Maximum responses are also presented in Table 4.12. Variation between program results and ETABS results is approximately 4% - 9%.

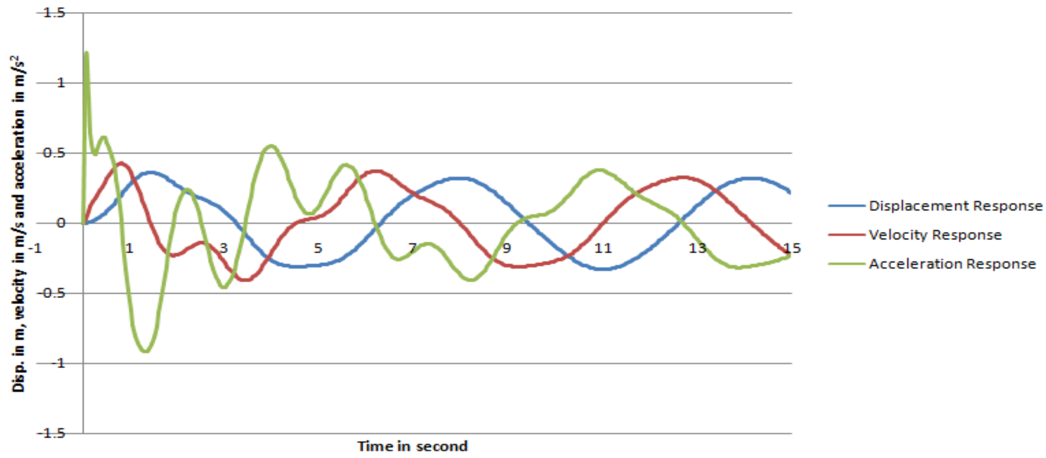

Figure 4.10: Program Results : Time History Analysis of 10 Storey Frame

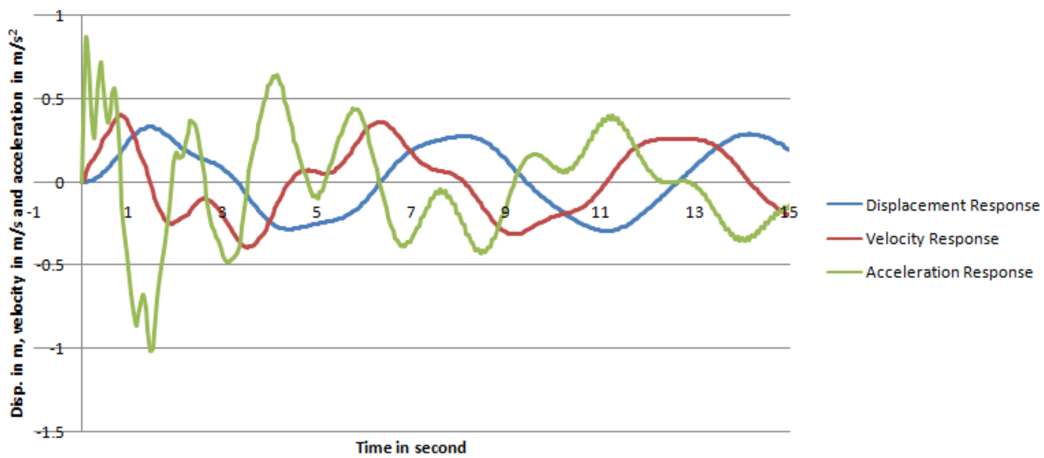

Figure 4.11: ETABS Results : Time History Analysis of 10 Storey Frame

Table 4.13: Maximum Response quantities for 10 Storey Frame

Maximum Responses		**Program Results**	**ETABS Results**
Displacement (m)	"+ ve"	0.360	0.333
	"- ve"	-0.333	-0.292
Velocity (m/s)	"+ ve"	0.425	0.406
	"- ve"	-0.410	-0.389
Acceleration (m/s^2)	"+ ve"	1.180	0.874
	"- ve"	-0.919	-1.013

Developed computer program is used for analysis of 10 storey frame subjected to dynamic force. Results are recorded for node 20 of top storey. Based on those results, time history curves (Fig. 4.10) are plotted. To validate program results, same frame is analyzed in ETABS. Results of ETABS are shown in Fig. 4.11. Maximum response quantities are also compared (4.14). Displacement based on computer program are 10% higher than displacement based on ETABS.

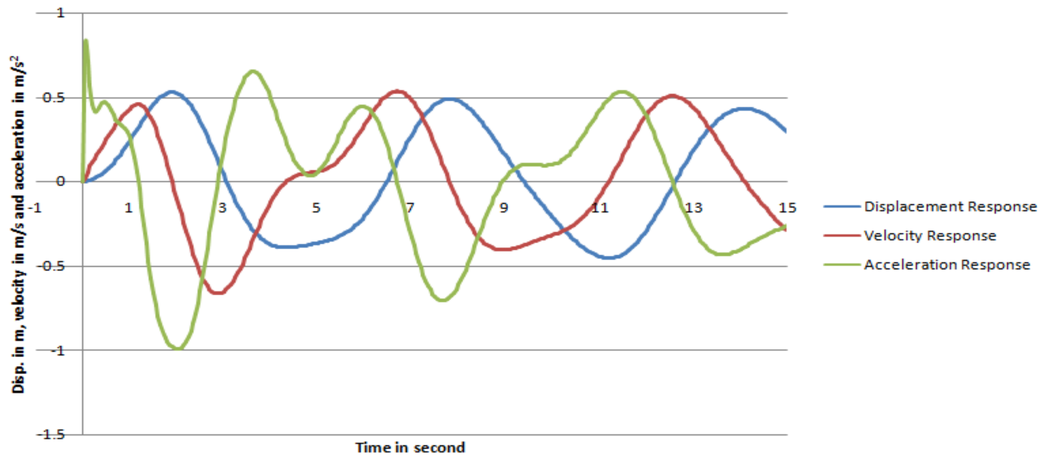

Figure 4.12: Program Results : Time History Analysis of 15 Storey Frame

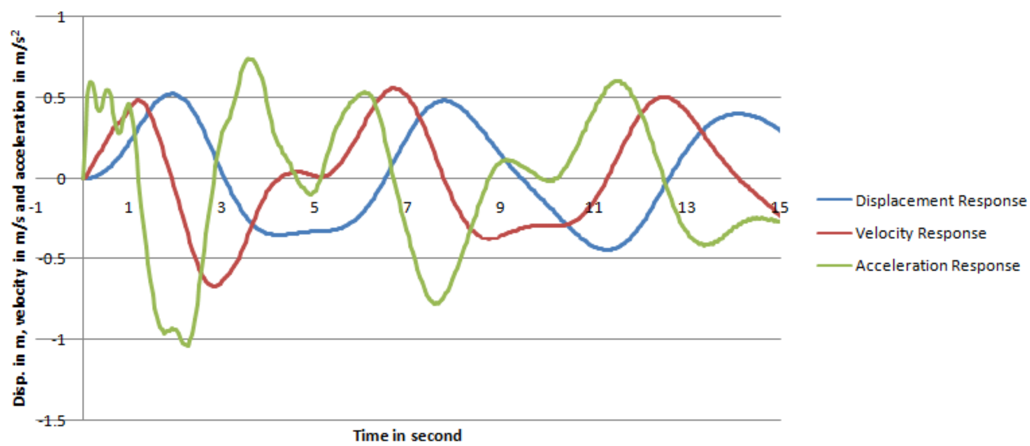

Figure 4.13: ETABS Results : Time History Analysis of 15 Storey Frame

Table 4.14: Maximum Response quantities for 15 Storey Frame

Maximum Responses		**Program Results**	**ETABS Results**
Displacement (m)	“+ ve”	0.528	0.524
	“- ve”	-0.454	-0.441
Velocity (m/s)	“+ ve”	0.534	0.557
	“- ve”	-0.665	-0.666
Acceleration (m/s^2)	“+ ve”	0.824	0.740
	“- ve”	-0.991	-1.036

Linear time history analysis is performed for 15 storey space frame using developed program and ETABS. Results of both analysis are presented in terms of time history curves in Fig. 4.12 and 4.13. Both curves are almost similar to each other. Maximum response of displacement, velocity and acceleration for t=0 to t=15 second are shown Table 4.14. Displacement based on developed computer program is same as displacement based on ETABS. Velocity and acceleration based on program and ETABS are differing by 4%-8%.

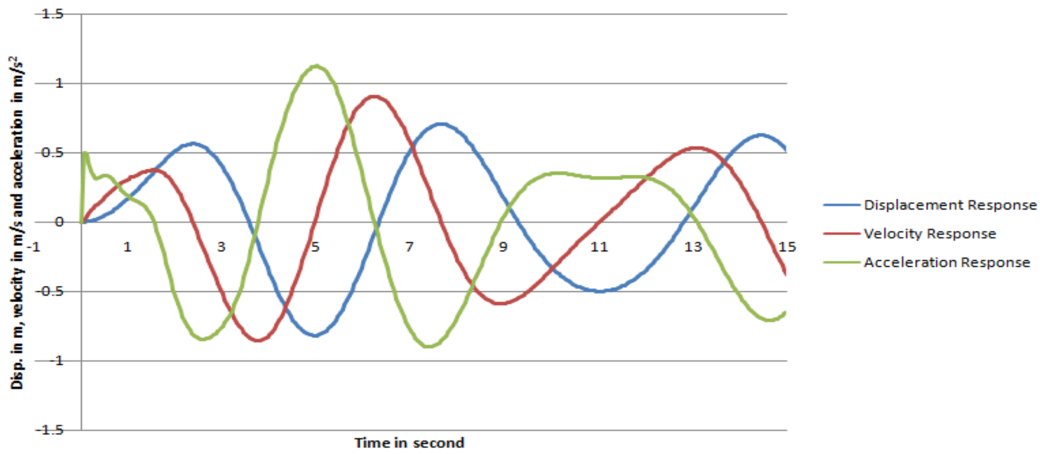

Figure 4.14: Program Results : Time History Analysis of 20 Storey Frame

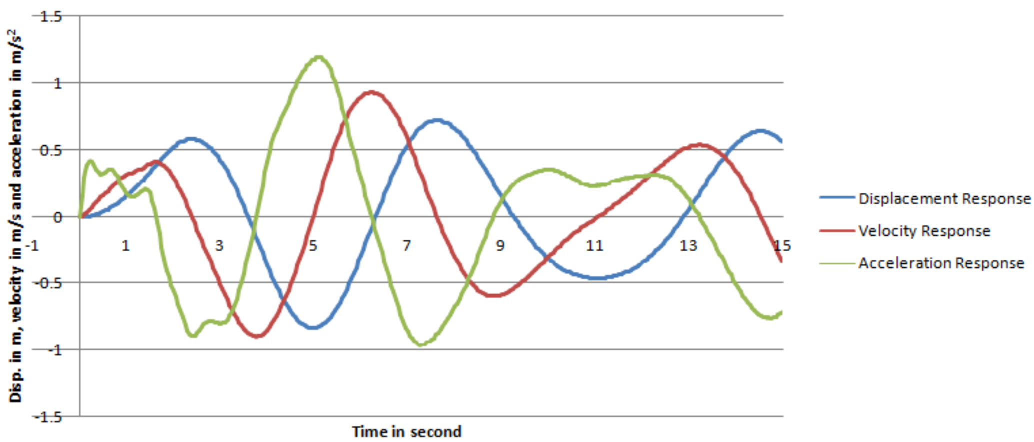

Figure 4.15: ETABS Results : Time History Analysis of 20 Storey Frame

Table 4.15: Maximum Response quantities for 20 Storey Frame

Maximum Responses		**Program Results**	**ETABS Results**
Displacement (m)	"+ ve"	0.705	0.721
	"- ve"	-0.820	-0.834
Velocity (m/s)	"+ ve"	0.904	0.929
	"- ve"	-0.859	-0.899
Acceleration (m/s^2)	"+ ve"	1.122	1.191
	"- ve"	-0.902	-0.958

20 storey frame subjected to dynamic force is analyzed using developed program. Results of node 20 for X-direction are recorded and time history curves are developed which is shown in Fig. 4.14. Same frame is also analyzed in ETABS. Results of ETABS are shown in Fig. 4.15. Maximum displacement, velocity and acceleration are shown in Table 4.15. Variation between results obtained from developed computer program and ETABS is about 2%-6%.

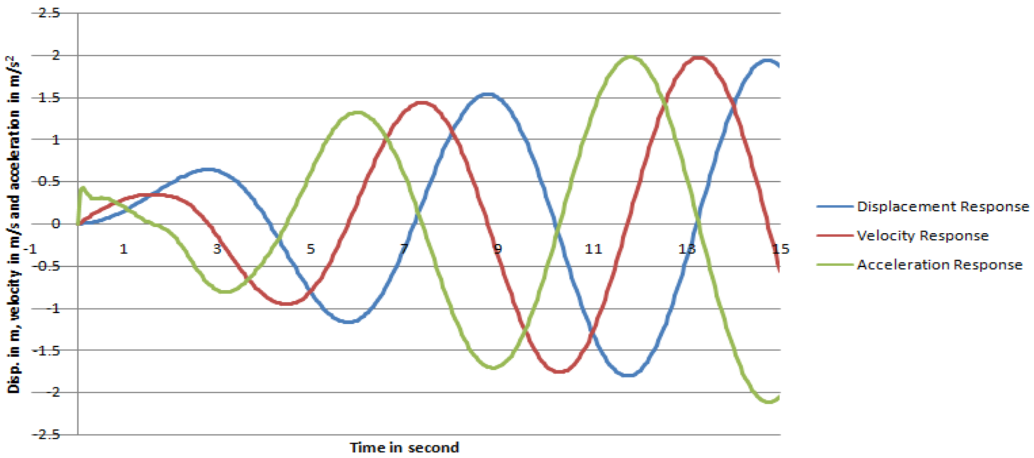

Figure 4.16: Program Results : Time History Analysis of 25 Storey Frame

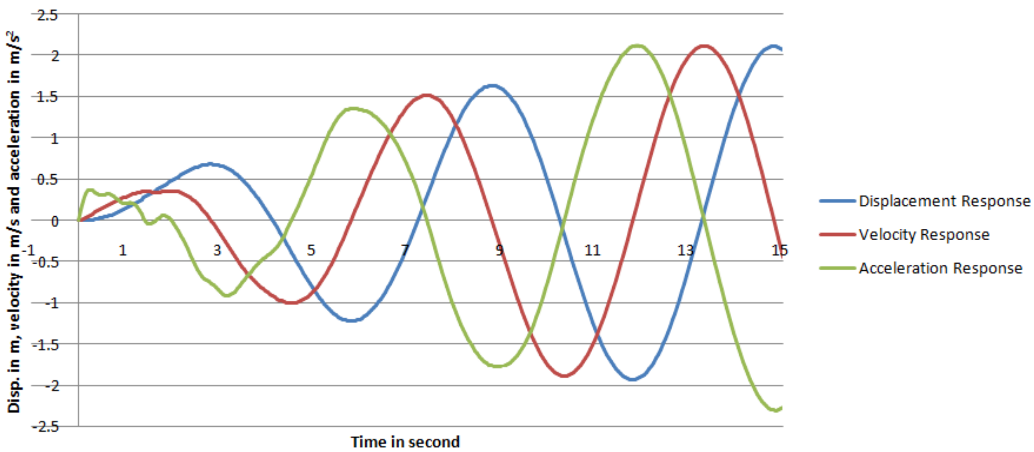

Figure 4.17: ETABS Results : Time History Analysis of 25 Storey Frame

Table 4.16: Maximum Response quantities for 25 Storey Frame

Maximum Responses		**Program Results**	**ETABS Results**
Displacement (m)	"+ ve"	1.940	2.111
	"- ve"	-1.809	-1.930
Velocity (m/s)	"+ ve"	1.968	2.115
	"- ve"	-1.765	-1.885
Acceleration (m/s^2)	"+ ve"	1.978	2.118
	"- ve"	-2.120	-2.304

Dynamic linear analysis results for 25 storey frame based on developed computer program and ETABS are shown in Fig. 4.16 and 4.17 respectively. Program results are verified with ETABS results. Maximum responses are also shown in Table 4.16. Here frequency of forcing function is same as natural frequency of space frame hence resonance is observed. Results of program and ETABS are almost similar which shows efficiency of program.

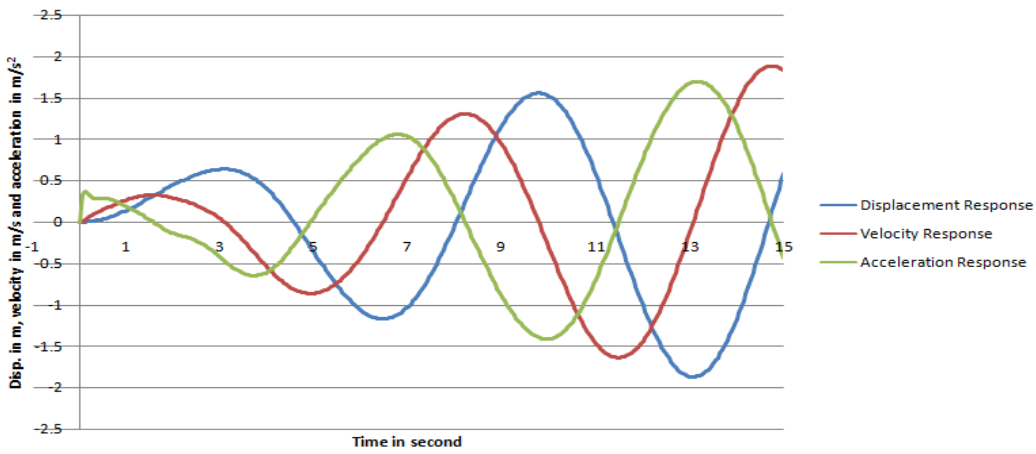

Figure 4.18: Program Results : Time History Analysis of 30 Storey Frame

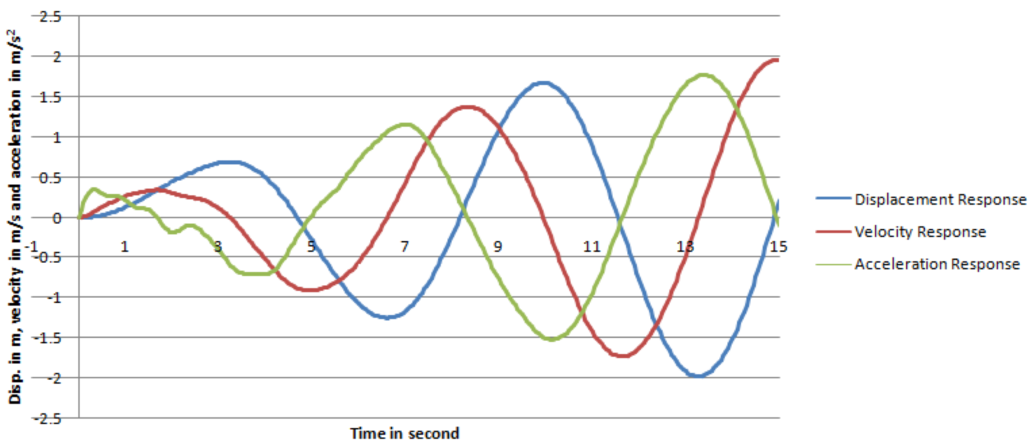

Figure 4.19: ETABS Results : Time History Analysis of 30 Storey Frame

Table 4.17: Maximum Response quantities for 30 Storey Frame

Maximum Responses		**Program Results**	**ETABS Results**
Displacement (m)	“+ ve”	1.555	1.675
	“- ve”	-1.871	-1.975
Velocity (m/s)	“+ ve”	1.882	1.962
	“- ve”	-1.640	-1.726
Acceleration (m/s^2)	“+ ve”	1.695	1.774
	“- ve”	-1.416	-1.515

Dynamic analysis for 30 storey frame subjected to time dependent force is performed. Results are shown in Fig. 4.18 and 4.19 in terms of time history curve. Maximum results are also shown in Table 4.17. Responses in form of displacement, velocity and acceleration obtained from computer program and ETABS are differing by 4%-8%.

Comparison of static and dynamic loading conditions for various space frames are made which validates developed programs. Also stability and efficiency of results are checked. Same programs are also extended for nonlinear analysis.

4.3 Summary

Linear static and dynamic analysis for various space frames are performed using computer program. Validation of results verifies analysis procedures of static and dynamic analysis. Analysis results obtained from computer program are verified with the analysis results of ETABS. Hence these procedures will be used to develop computer oriented analysis procedure for nonlinear case.

Chapter 5

Nonlinear Static and Dynamic Analysis

5.1 General

Linear analysis results are generally used for designing structural members but to understand exact behavior of structure, it is required to find nonlinear response. Nonlinearities may be due to large deformations or due to nonlinear constitutive relationship. Nonlinear analysis is nothing but incremental linear analysis in which applied loads are divided into small increments. In this chapter, nonlinear equilibrium equations are derived for static and dynamic loading conditions. Various methods to solve these nonlinear equations are also discussed. Main focus of this chapter is on geometric nonlinear analysis.

5.2 Various conditions of Nonlinear Analysis

Based on deformation and strain levels, various conditions are defined which can reduce efforts and time of analysis. Conditions are derived considering both material and geometric nonlinearities.

a. **Small Strain and Small deformation**

In this case, nonlinear analysis become material nonlinear analysis as overall structure deformation and local strain levels are small. No second order effects are considered.

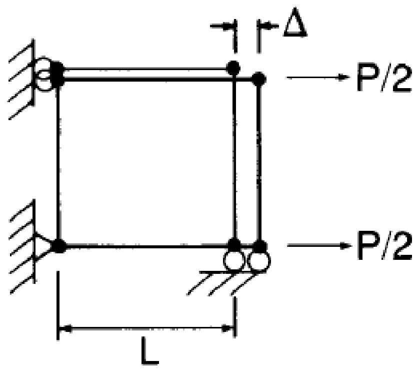

Figure 5.1: Small Strain and Small Deformation

b. **Small strain and Large deformation**

When structural deformation is large and member deformation or local strains are small, nonlinear analysis is adopted. Here due to small strain levels, geometry of space frame is not updated. Nonlinear response can be calculated by incorporating geometric stiffness matrix. Material nonlinearity may be there.

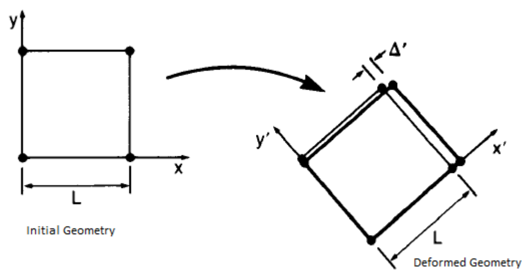

Figure 5.2: Small strain and Large deformation

c. **Large strain and Large deformation**

This is very general case where both structural deformation as well as local strains are large. In this condition, both geometry of space frame and geometric stiffness matrix are updated based on deformed geometry of structure. Both material as well as geometric nonlinearities are considered.

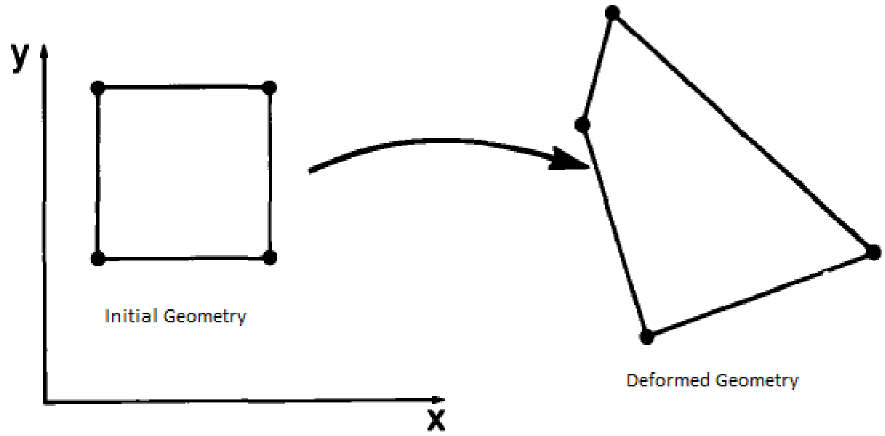

Figure 5.3: Large strain and Large deformation

5.3 Geometric Nonlinear Analysis

Space frames have large number of slender sections which may subjected to large deformations. Due to large deformations, second order effects are induced. Buckling of space frame is also caused due to large deformations. These result into change in geometry and hence load deformation behavior changes. Effects of large deformation are included by updating stiffness of space frame. Incremental approach is used to solve geometric nonlinear problems.

To understand incremental approach, Consider Fig. 5.4, in which three stages of deformations are shown. Applied loads are divided into two increments. C_0 is initial configuration, C_1 is last known deformed configuration and C_2 is current deformed configuration. From C_0 to C_1, frame is analyzed based on initial geometry of structure. Hence for every nonlinear analysis, first step is always elastic linear analysis. At the end of first step, induced forces are known. From C_1 to C_2, equilibrium equations are derived based on deformed geometry of C_1 configuration. Results of C_2 configuration are final results. Here, nonlinear analysis is done in two increments.

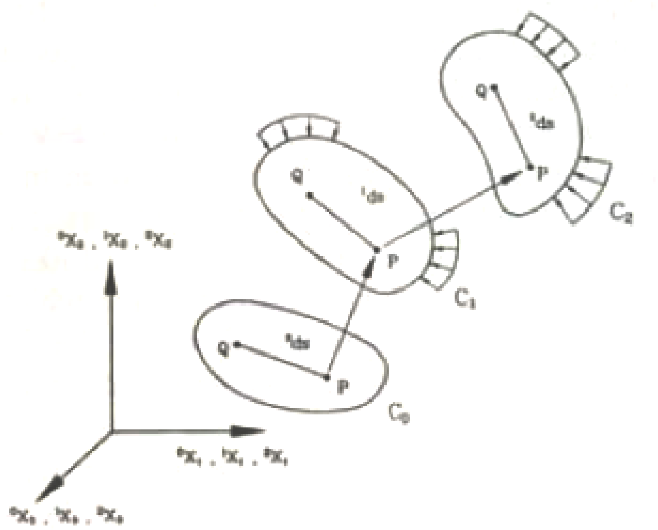

Figure 5.4: Deformation configurations [17]

This is general procedure of nonlinear analysis. For geometric nonlinear analysis, geometric stiffness of structure is calculated based on known deformed geometry (C_1 configuration).

5.3.1 Geometric Stiffness Matrix for Space Frame Element

Geometric stiffness means effects of unit deformation at one degree of freedom on other degrees of freedom. In simple words, force corresponding to nodal coordinate i due to unit displacement at coordinate j. (i and j are different DOFs). Geometric stiffness matrix is derived based on energy method. Consider two configuration C_1

and C_2, where induced forces for C_1 configuration are shown in Fig. 5.5. Now to calculate deformations for C_2 configuration, stiffness matrix should be updated based on known deformed geometry. Geometric Stiffness matrix[17] for space frame element (Fig. 5.5) is shown in Eq. 5.1.

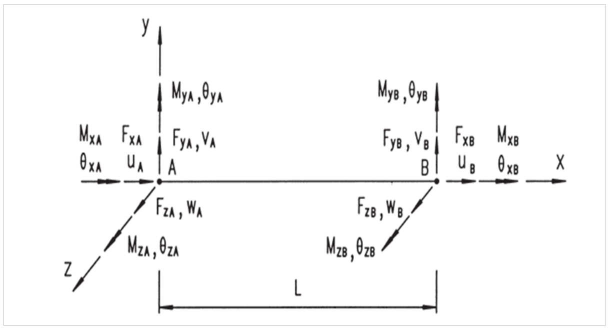

Figure 5.5: Induced actions for C_1 configuration

$$[K_g] = \begin{bmatrix} 0 & a & c & 0 & 0 & 0 & 0 & -a & -c & 0 & 0 & 0 \\ & b & 0 & d & g & -h & -a & -b & 0 & l & -g & -h \\ & & b & e & h & g & -c & 0 & -b & m & h & -g \\ & & & f & i & k & 0 & -d & -e & -f & -i & -k \\ & & & & j & 0 & 0 & -g & -h & -i & n & -o \\ & & & & & j & 0 & h & -g & -k & o & n \\ & & & & & & 0 & a & c & 0 & 0 & 0 \\ & & sym & & & & & b & 0 & -l & g & h \\ & & & & & & & & b & -m & -h & g \\ & & & & & & & & & f & a & c \\ & & & & & & & & & & j & 0 \\ & & & & & & & & & & & j \end{bmatrix} \tag{5.1}$$

where,

$a = \frac{M_{za}+M_{zb}}{L^2}$; $b = \frac{6F_{xb}}{5L}$

$c = -\frac{M_{ya}+M_{yb}}{L^2}$; $d = \frac{M_{ya}}{L}$

$e = \frac{M_{za}}{L}$; $f = \frac{F_{xb}I_x}{AL}$

$g = \frac{M_{xb}}{L}$; $h = -\frac{F_{xb}}{10}$

$i = \frac{M_{za}+M_{zb}}{6}$; $j = \frac{2F_{xb}L}{15}$

$k = -\frac{M_{ya}+M_{yb}}{6}$; $l = \frac{M_{yb}}{L}$

$m = \frac{M_{zb}}{L}$; $n = -\frac{F_{xb}L}{30}$

$o = -\frac{M_{xb}}{2}$

Above geometric stiffness matrix is symmetric. Components of geometric stiffness matrix depend on joint actions of last known geometry. In geometric nonlinear analysis, tangent stiffness matrix is calculated using Eq. 5.2.

$$[K_t] = [K_e] + [K_g] \tag{5.2}$$

5.4 Methods of Solution

In nonlinear analysis, whole analysis is divided into number of steps. Loads are applied as a series of small increments and for each step deformations are calculated. Change in stiffness can be employed by considering tangent stiffness matrix. Hence incremental equilibrium equation can be defined as Eq 5.3.

$$[K_t]\{\Delta w\} = \{\Delta W\} \tag{5.3}$$

where, $\{\Delta w\}$ is incremental displacement corresponding to load increment $\{\Delta W\}$. To solve this type of incremental equation, various methods[15] are available like

- Incremental Method
- Iterative Method (Newton Raphson Method)
- Combination of Incremental and iterative Method

5.4.1 Incremental Method

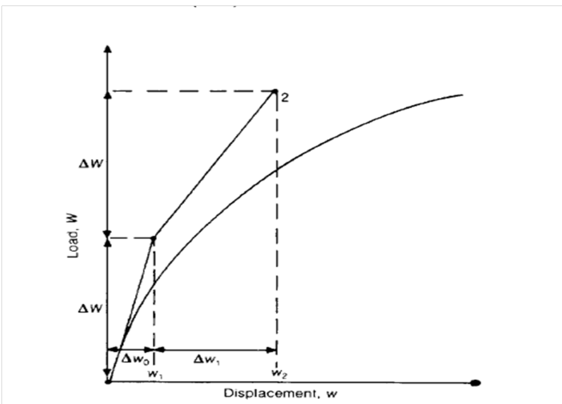

Figure 5.6: Incremental Method [15]

a. Specify initial conditions. $\{w\}_0$

b. Form tangent stiffness matrix using initial geometric condition. $[K_t]_0$

c. Specify Load Increment. $\{\Delta W\}$

d. Solve $[K_t]_0\{\Delta w\}_0 = \{\Delta W\}$ for $\{\Delta w\}_0$

e. Update displacement using $\{w\}_1 = \{w\}_0 + \{\Delta w\}_0$

f. Find joint actions and update tangent stiffness matrix. $[K_t]_1$

for each increments (i), where, i = 2,3,...,n.

g. Solve $[K_t]_{i-1}\{\Delta w\}_{i-1} = \{\Delta W\}$ for $\{\Delta w\}_{i-1}$

h. Update displacement using $\{w\}_i = \{w\}_{i-1} + \{\Delta w\}_{i-1}$

i. Find joint actions and update tangent stiffness matrix. $[K_t]_i$

Repeat steps **g** to **i** for all increment.

Graphical representation is shown in Fig. 5.6. Smooth line shows actual load deformation curve. At the end of each step, there is always some difference between actual and calculated solution. This error increases as load increment is kept large.

5.4.2 Iterative Method (Newton Raphson Method)

Iterative method is adopted to reduce the error in terms of unbalanced forces. Unbalanced force $\{dg\}$ which is difference between external and internal forces, is found out at the end of each iteration. Based on equilibrium equation, displacements $\{dw\}$ corresponding to unbalanced forces are found out using $[K_t]\{dw\} = \{dg\}$. As number of iteration increases, unbalanced force reduces. Iterations are continued till tolerance limit of unbalanced force is achieved. Graphical representation of iterative method is shown in Fig. 5.7.

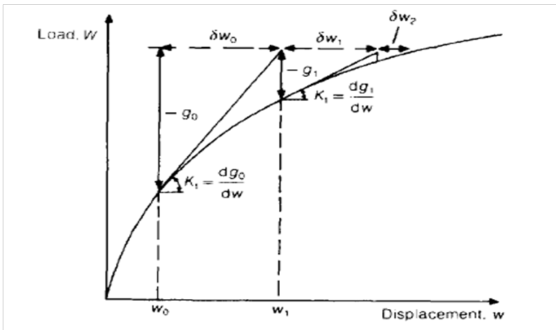

Figure 5.7: Iteration Method [15]

5.4.3 Combination of Incremental and iterative Method

This method is used to overcome the limitations of incremental method. Error in each incremental step is eliminated by iteration method. This combined steps can be understood with the help of graphical representation. Fig. 5.8.

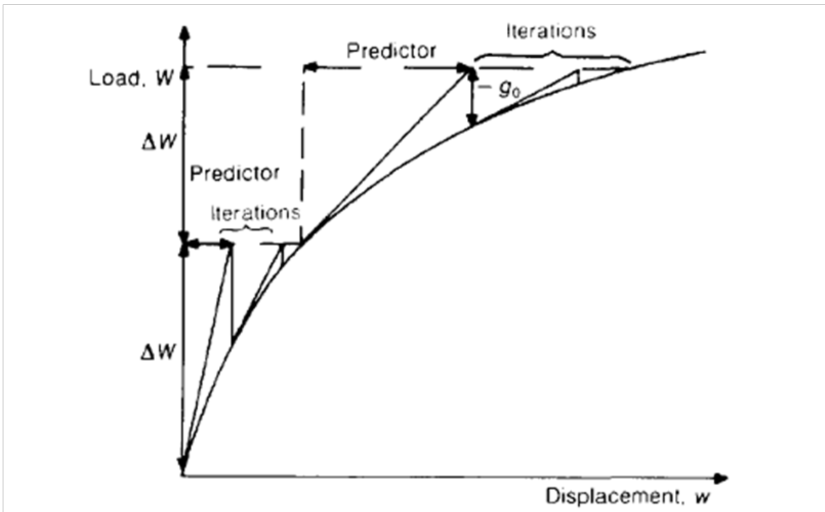

Figure 5.8: Combination of Incremental and iterative Method [15]

a. Specify initial conditions. $\{w\}_0$

b. Form tangent stiffness matrix using initial geometric condition. $[K_t]_0$

c. Specify Load Increment. $\{\Delta W\}$

d. Solve $[K_t]_0\{\Delta w\}_0 = \{\Delta W\}$ for $\{\Delta w\}_0$

e. Update displacement using $\{w\}_1 = \{w\}_0 + \{\Delta w\}_0$

f. Find joint actions and update tangent stiffness matrix. $[K_t]_1$

for each increments (i), where, i = 2,3,...,n.

g. Solve $[K_t]_{i-1}\{\Delta w\}_{i-1} = \{\Delta W\}$ for $\{\Delta w\}_{i-1}$

h. Update displacement using $\{w\}_i = \{w\}_{i-1} + \{\Delta w\}_{i-1}$

i. Find joint actions and update tangent stiffness matrix. $[K_t]_i$

j. **for each iteration (j)** where j = 1,2,...,itr

- Find unbalanced force. $\{g\}$ = external force - internal force.
- Find displacements corresponding to load vector $\{g\}$.
- Solve $[K_t]_i\{dw\}_j = \{g\}$ for $\{dw\}_j$.
- Update geometry. $\{w\}_i = \{w\}_i + \{dw\}_j$
- Find joint actions and update tangent stiffness matrix. $[K_t]_i$

k. Repeat step **j** till tolerance limit for unbalanced force reaches.

l. Repeat steps **g** to **k** for all increments.

Draw nonlinear load-displacement curve.

5.5 Static Nonlinear Analysis

As discussed earlier, nonlinear static analysis is linear incremental method. In this section, detail procedure of static nonlinear analysis is listed in which combined incremental and iteration method is used. Nonlinear static equation can be written in incremental form as Eq. 5.4. Algorithm of linear static analysis is extended to solve nonlinear problem. Detail steps are prepared for space frame structure. For combined incremental and iterative method, load factor(λ) should be properly selected to divide incremental loads. Load sequences should be properly maintained.

$$[K_t]\{\Delta D\} = \{\Delta F\} \tag{5.4}$$

Before proceeding for algorithm, it is important to understand derivation of unbalanced forces at the beginning of iteration. Unbalanced forces is found out by incorporating equilibrium conditions for each nodes. For matrix calculation, unbalanced forces should be in vector form and can be calculated using Eq. 5.5.

$$\begin{Bmatrix} g_1 \\ g_2 \\ . \\ . \\ . \\ g_{6*nfj} \end{Bmatrix} = \begin{Bmatrix} \Delta F_1 \\ \Delta F_2 \\ . \\ . \\ . \\ \Delta F_{6*nfj} \end{Bmatrix} - \begin{Bmatrix} \Delta A_{gg1} \\ \Delta A_{gg2} \\ . \\ . \\ . \\ \Delta A_{gg6*nfj} \end{Bmatrix} \tag{5.5}$$

where $\{\Delta A_{gg}\}$ = Difference of Joint actions of two consecutive increments or iterations with respect to global coordinate system. and nfj = number of free joints.

5.5.1 Detailed Analysis Procedure

Detail procedure for nonlinear static analysis is derived in two stages. First step is elastic analysis and second stage is incremental steps.

Elastic Analysis

a. **Identification of space frame data**

b. **Construction of Global Tangent Stiffness Matrix** $[K_t] = [K_e] + [K_g]$

   For first incremental step, $[K_t] = [K_e]$.

c. **Formation of Incremental Load Vector** $\{\Delta F\}$

d. **Solution of equation**

   Solve $[K_t]\{\Delta D\} = \{\Delta F\} + \{PL\}$ for $\{\Delta D\}$, where $\{PL\}$ = permanent loads (e.g. Gravity Loads)
   Calculate Total displacement, $\{D\} = \{\Delta D\}$.
   Calculate member end actions $\{A_m\}$ for each member using Eq. 3.11.
   Also find out induced forces at each free node of space frame $\{A_{gg}\}$ based on total displacement.

e. **Update geometry of space frame and construction of new elastic stiffness matrix**

   Update coordinates of each node using,

$$\begin{aligned} \{x\} &= \{x\} + \{D_x\} \\ \{y\} &= \{y\} + \{D_y\} \\ \{z\} &= \{z\} + \{D_z\} \end{aligned} \tag{5.6}$$

   Based on new coordinates, develop new elastic stiffness matrix.

f. **Construction of Geometric Stiffness Matrix** $[K_g]$

Based on member end action $\{A_m\}$, form geometric stiffness matrix for each member using Eq. 5.1. Assemble them to form global geometric stiffness matrix.

Incremental Steps

a. **Update Global Tangent Stiffness Matrix** $[K_t] = [K_e] + [K_g]$.

b. **Solution of equation**

Solve $[K_t]\{\Delta D\} = \{\Delta F\}$ for $\{\Delta D\}$.

Update Total displacement, $\{D\} = \{D\} + \{\Delta D\}$

Find member end actions for all members $\{A_m\}$.

For updated total displacement find joint actions. $\{A_{gg}\}$

c. **Update geometry of space frame and construction of new elastic stiffness matrix**

d. **Construction of Geometric Stiffness Matrix** $[K_g]$

Iteration Steps

(1) **Update Global Tangent Stiffness Matrix** $[K_t] = [K_e] + [K_g]$

(2) **Construction of Unbalanced Force Vector** $\{g\}$

Using Eq. 5.5, calculate unbalanced force.

(3) **Solution of Equation**

Solve $[K_t]\{dd\} = \{g\}$ for $\{dd\}$.

Update Total displacement, $\{D\} = \{D\} + \{dd\}$

Find member end actions for all members $\{A_m\}$.

For updated total displacement find joint actions. $\{A_{gg}\}$

(4) **Construction of Geometric Stiffness Matrix** $[K_g]$

(5) **Check tolerance limit for unbalanced force**

Find root mean square value of unbalanced forces and incremental loads.

$$g_{RMS} = \sqrt{\frac{g_1^2+g_2^2+...+g_{6*nfj}^2}{6*nfj}} \tag{5.7}$$

$$\Delta F_{RMS} = \sqrt{\frac{\Delta F_1^2+\Delta F_2^2+...+\Delta F_{6*nfj}^2}{6*nfj}} \tag{5.8}$$

Now ratio of unbalanced force to incremental load should be less than tolerance limit. If this condition is not satisfied repeat iteration steps till $\frac{g_{RMS}}{\Delta F_{RMS}} <$ tolerance limit

Repeat incremental steps till desire load limit is achieved.

Displacements response of desire node are recorded at the end of each incremental step. Above method is based on load control approach so it is not possible to trace load-deformation curve for snap-back curve. Whenever applied loads exceed the ultimate loads, results at the end of iteration will never converged and hence load deformation path can not be found.

5.6 Dynamic Nonlinear Analysis

In dynamic nonlinear analysis, combination of time stepped method and iteration method is used. Wilson-theta method is adopted to plot time history response. Algorithm for nonlinear analysis is same as linear time history analysis. Time increment method is used for both linear and nonlinear analysis. Only difference is inclusion of Newton-Raphson iteration method.
Dynamic equilibrium equation can be written in incremental form as Eq. 5.9. Here increments are applied with respect to time. Hence load increment vector is not same for all steps.

$$[M]\{\Delta\ddot{u}\} + [C]\{\Delta\dot{u}\} + [K]\{\Delta u\} = \{\Delta F(t)\} \ \ or \ \ [M]\{\ddot{X}_g\} \tag{5.9}$$

5.6.1 Detailed Analysis Procedure

With some modification in linear time history analysis procedure, algorithm for nonlinear analysis is prepared. Dynamic analysis is also divided into two stage. First is elastic analysis stage and second is incremental stage.

In first step of analysis, full gravity loads are included with first time incremental loads and then linear analysis is performed. It is also important to include gravity loads in nonlinear analysis because gravity loads produces second order effects due to lateral displacements. Also it is not advisable to use superimposition of gravity load analysis result and dynamic analysis result for nonlinear case. Hence, gravity load analysis is the first step of nonlinear dynamic analysis. Same steps of linear time history analysis are used for first step. Detail steps are listed below.

a. **Identification of Space frame data and Construction of Stiffness, Mass and Damping matrix**

b. **Initialization**

Read initial vectors of displacements $\{u_0\}$, velocity $\{\dot{u}_0\}$ and forces $\{F_0\}$ at each node of space frame for all degrees of freedom. Calculate initial acceleration based on following equation.

$$[M]\{\ddot{u}_0\} = \{F_0\} - [C]\{\dot{u}_0\} - [K]\{u_0\} \tag{5.10}$$

c. **Define values of θ and Δt**

d. **Formation of effective stiffness matrix**

Form effective stiffness matrix using following equation, where value of a_1, a_2, a_3 and a_4 are same as discussed in section 3.4.4.2

$$\overline{[K]} = [K_t] + a_4[M] + a_1[C] \tag{5.11}$$

e. **Formation of effective load vector**

Form incremental load vector for i^{th} step, by linear interpolating load for time interval t_i and $(t_i + \tau)$. Here load increment is considered for extended time period. $(\theta\Delta t)$.

$$\{\widehat{\Delta F_i}\} = \{F_{i+1}\} + (\{F_{i+2}\} - \{F_{i+1}\})(\theta - 1) - \{F_i\} \tag{5.12}$$

where,

$F(t)_i = F(i * \Delta t)$ for i^{th} time increment.

- For (i = 0) first step, full gravity load vector $\{PL\}$ is added to $\{\widehat{\Delta F_i}\} = \{\widehat{\Delta F_i}\} + \{PL\}$.
- For other steps (i $\geq$ 1), only incremental load vector $\{\widehat{\Delta F_i}\}$ is considered.

Find effective load vector $\overline{\widehat{\Delta F_i}}$,

$$\{\overline{\widehat{\Delta F_i}}\} = \{\widehat{\Delta F_i}\} + (a_2[M] + 3[C])\{\dot{u}_i\} + (3[M] + a_3[C])\{\ddot{u}_i\} \tag{5.13}$$

where,

$\{\dot{u}_i\}$ = velocity vector for i^{th} time step at each free nodes of space frame $\{\ddot{u}_i\}$ = acceleration vector for i^{th} time step at each free nodes of space frame

f. **Solution for incremental displacement vector $\{\widehat{\Delta u_i}\}$ for extended time period**

Solve following equation for $\{\widehat{\Delta u_i}\}$

$$\overline{[K]}\{\widehat{\Delta u_i}\} = \{\overline{\Delta F_i}\} \tag{5.14}$$

Calculated member end actions corresponding to $\{\widehat{\Delta u_i}\}$ displacement.

Construct geometric stiffness matrix.

For first step (i=0), there will be no iteration steps. So that Total displacement Vector $\{D\} = \{\widehat{\Delta u_i}\}$

From second incremental steps (for $i \geq 1$), unbalanced forces are generated due to modification of tangent stiffness. Iteration steps are included in **step f** for all next increments (i=1,2,...,n). Iteration steps are same as static nonlinear case.

Iteration Steps (Newton Raphson Method

(1) **Update effective stiffness matrix** $\overline{[K]}$

(2) **Find unbalanced Force** $\{g\}$

(3) **Solve** $\overline{[K]}\{dd\} = \{g\}$

(4) **Update Displacement Value**
$\{\widehat{\Delta u_i}\} = \{\widehat{\Delta u_i}\} + \{dd\}$ Update Total displacement Vector, $\{D\} = \{D\} + \{\widehat{\Delta u_i}\}$

(5) **Construct geometric stiffness matrix based on induced actions**

(6) **Check tolerance limit**

At the end of iteration, correct value of incremental displacement is calculated. using equation $\{\widehat{\Delta u_i}\}$ = Total displacement at the end of increment (i) - Total displacement at the end of increment (i-1)

g. **Calculation of incremental acceleration $\{\widehat{\Delta \ddot{u}_i}\}$ for extended time period**

$$\{\widehat{\Delta \ddot{u}_i}\} = a_4\{\widehat{\Delta u_i}\} - a_2\{\dot{u}_i\} - 3\{\ddot{u}_i\} \tag{5.15}$$

h. **Find incremental quantities for increment of Δt**

Linear variation of acceleration is assumed for extended time period. Using this assumption, incremental acceleration is calculated by,

$$\{\Delta \ddot{u}_i\} = \frac{\widehat{\{\Delta \ddot{u}_i\}}}{\theta} \tag{5.16}$$

Other incremental quantities are,

$$\{\Delta \dot{u}_i\} = \{\ddot{u}_i\}\Delta t + \frac{1}{2}\{\Delta \ddot{u}_i\}\Delta t \tag{5.17}$$

$$\{\Delta u_i\} = \{\dot{u}_i\}\Delta t + \frac{1}{2}\{\ddot{u}_i\}\Delta t^2 + \frac{1}{6}\{\Delta \ddot{u}_i\}\Delta t^2 \tag{5.18}$$

i. **Find response quantities at time** (t_{i+1})

$$\begin{aligned}\{u_{i+1}\} &= \{u_i\} + \{\Delta u_i\}\\ \{\dot{u}_{i+1}\} &= \{\dot{u}_i\} + \{\Delta \dot{u}_i\}\\ \{\ddot{u}_{i+1}\} &= \{\ddot{u}_i\} + \{\Delta \ddot{u}_i\}\end{aligned} \tag{5.19}$$

Repeat steps from (d) to (i) for each increments. Results of analysis gives response of space frame for each time steps. Using those, nonlinear time history curves can be developed.

5.7 Computer Program Development

Nonlinear analysis methods discussed in above sections are computer oriented methods. For effective and efficient solutions, computer programs are developed. Incremental-iterative method is used for static and dynamic analysis. Load control strategy is used to trace nonlinear path. Every nonlinear analysis method is divided into three step.

- Linear Analysis Step
- Increment Steps
- Iterative Steps

Last two steps are repeated for all load increments.

5.7.1 Static Nonlinear Analysis

Computer program is developed for procedure discussed in previous sections. Input file is generated based on user defined data. Input file includes initial geometry of space frame, material properties, cross-sectional details and boundary conditions. These data are generated by using C++ program. Core program consists of above three step. In first step of analysis, load vector is calculated in which full gravity load and first incremental load are considered. Displacement and member end forces are calculated based on linear elastic analysis. Subsequently program generates geometric stiffness matrix at the end of each step. Also geometry of space frame is updated at the end of each step. In second step, only incremental load vector is used and analyzed. Unbalanced forces generated at the end of step is reduced by iterations. Same procedure is adopted for each load increment. Analysis is performed till required load is achieved. Results in terms of load deformations data are calculated. Results at required node are recorded in output files. Based on those data, load deformation curves can be developed. Flow chart of developed computer program is presented in Fig. 5.9, 5.10 and 5.11.

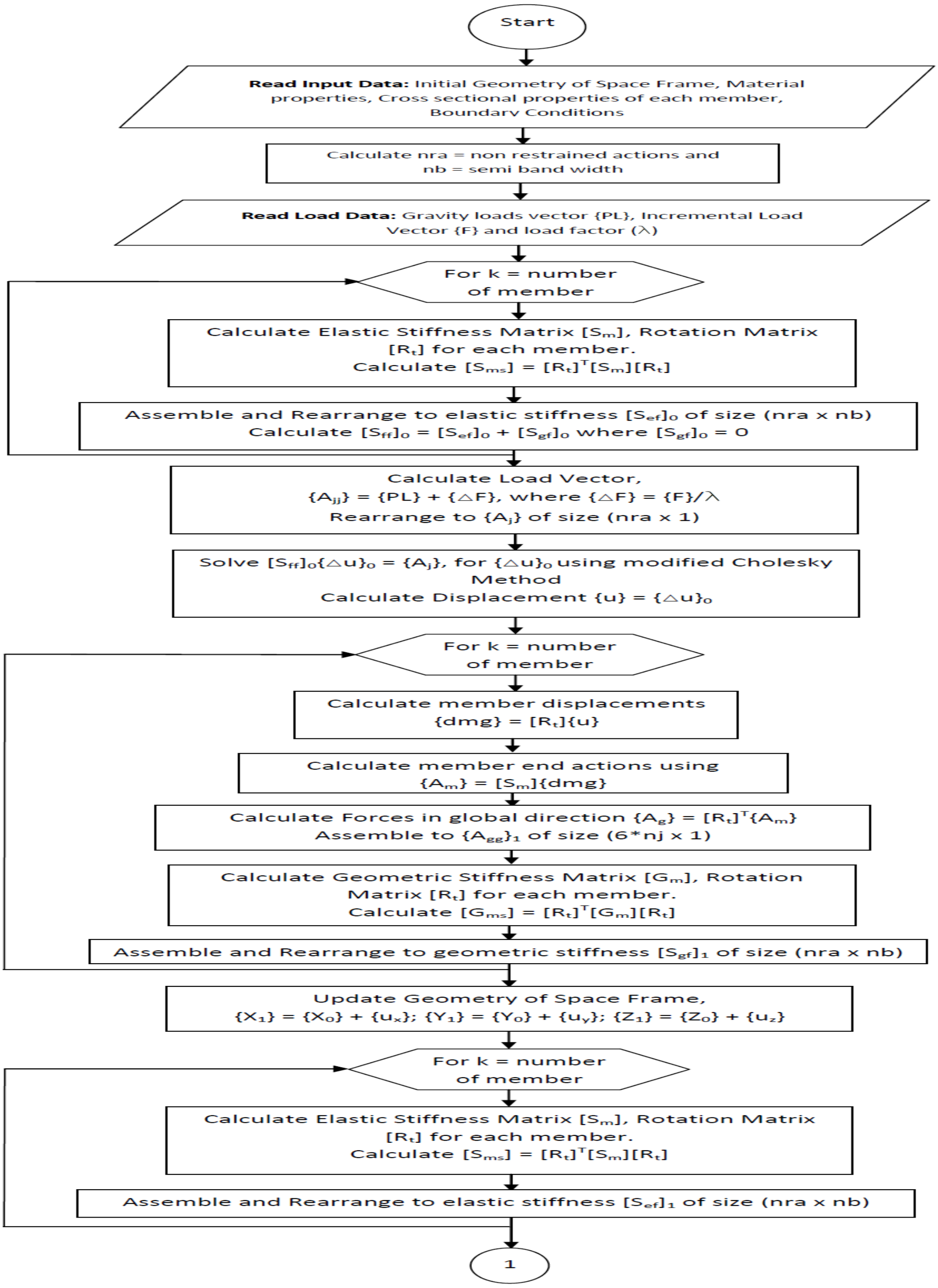

Figure 5.9: Flow Chart : Static Nonlinear Analysis (1^{st} Step)

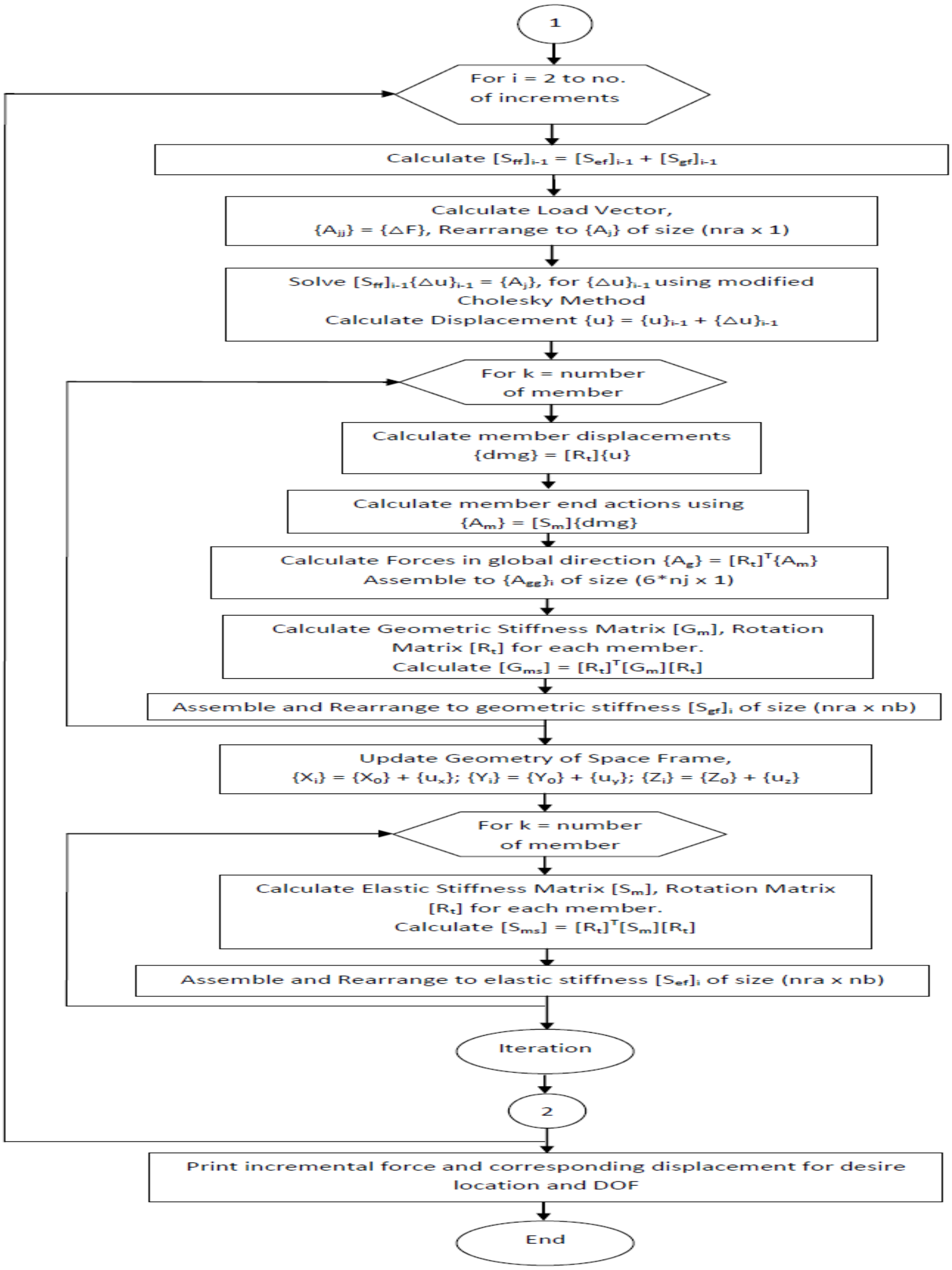

Figure 5.10: Flow Chart : Static Nonlinear Analysis (2^{nd} Step)

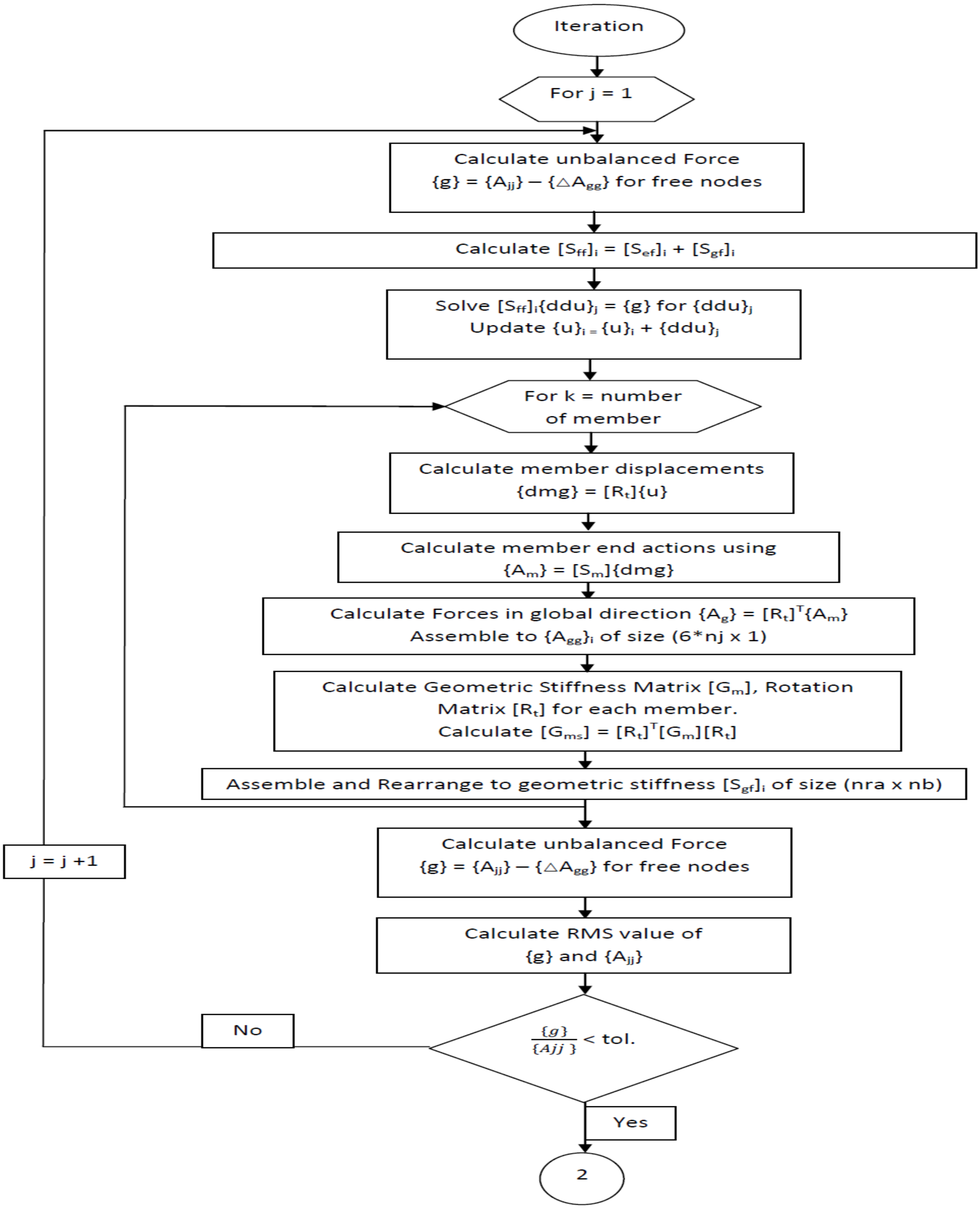

Figure 5.11: Flow Chart : Static Nonlinear Analysis (3^{rd} Step)

5.7.2 Dynamic nonlinear Analysis

Combination of Wilson-Theta method and Newton-Raphson method is used to analyze space frame for dynamic loading condition. Input file is generated as discussed in dynamic linear analysis. In pre-processing step, values of Δt and θ is selected by user. Consistent mass matrix and Rayleigh damping matrix are constant throughout the analysis. This analysis is also includes three steps. In first step, full gravity load and first incremental load are adopted and analyzed. Results of this step is used as initial conditions for second steps. Geometric stiffness matrix is calculated at the end of each step and based on that tangent stiffness matrix is updated. Unbalanced forces which are generated at the end of each steps are recovered by iterations. Results in form of time history responses for displacements, velocity and accelerations are calculated. Results of desire node and desire degree of freedom are recorded. Based on those results, time history responses can be plotted. Flow of Nonlinear dynamic analysis is shown in Fig 5.12, 5.13 and 5.14.

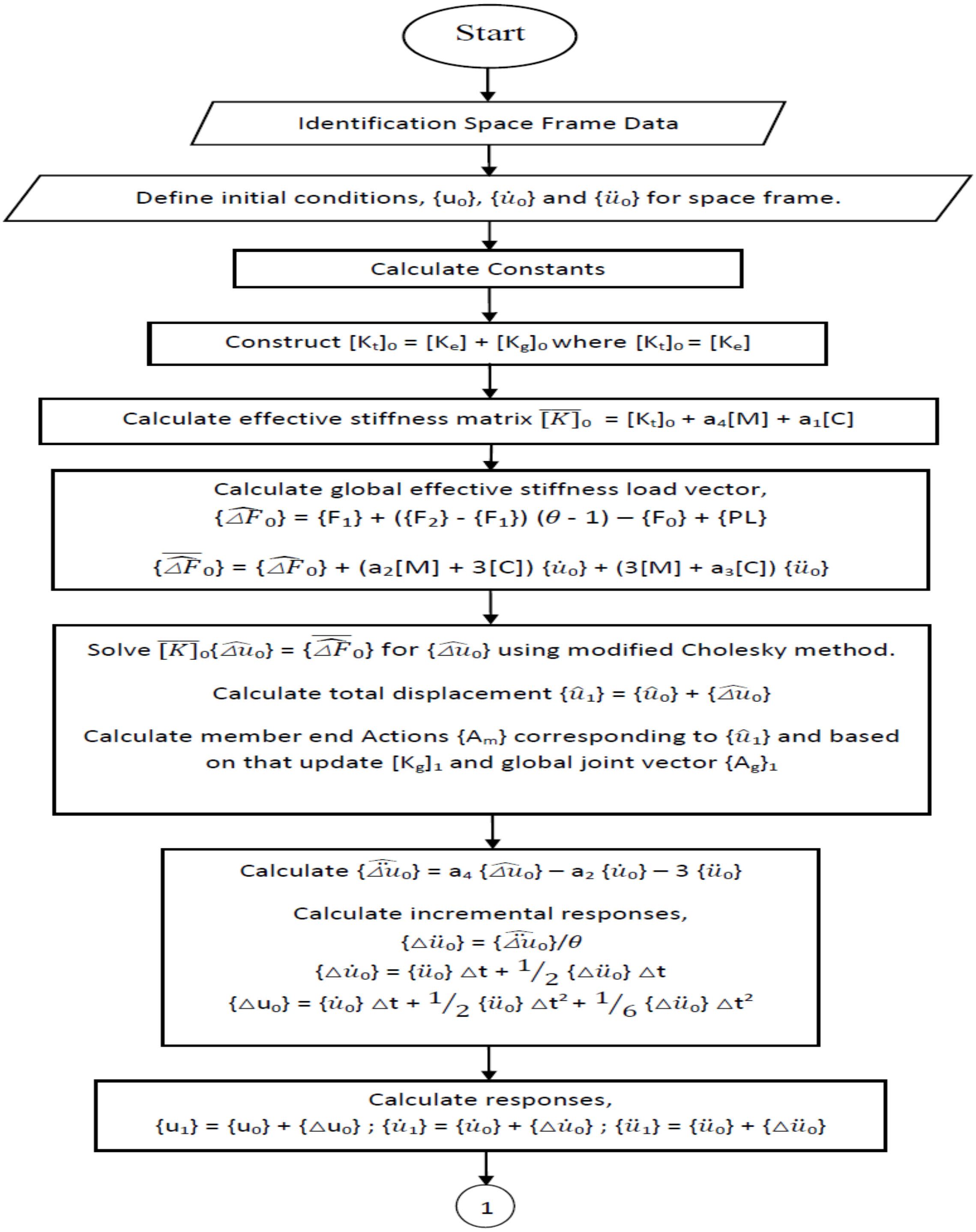

Figure 5.12: Flow Chart : Dynamic Nonlinear Analysis (1^{st} Step)

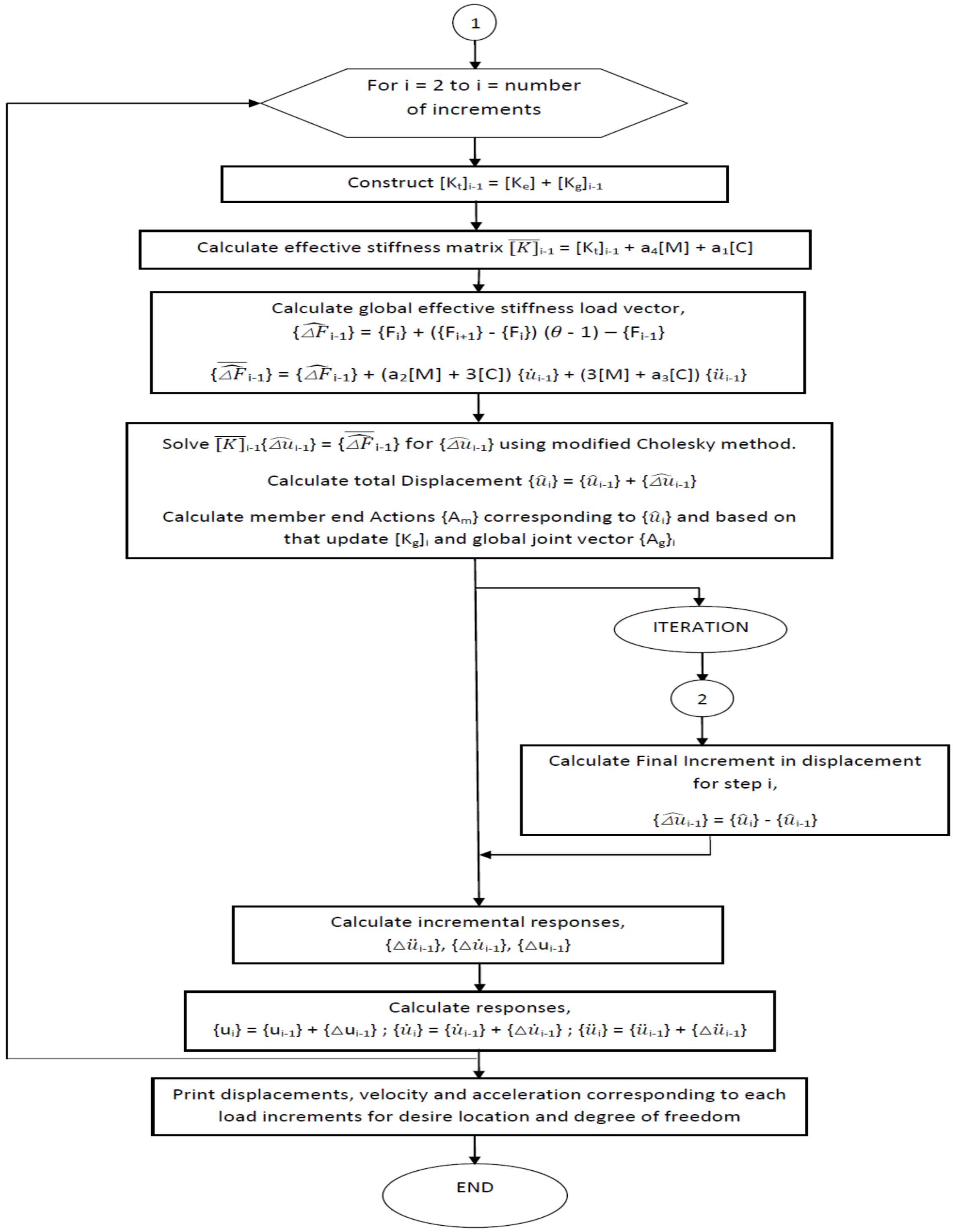

Figure 5.13: Flow Chart : Dynamic Nonlinear Analysis (2^{nd} Step)

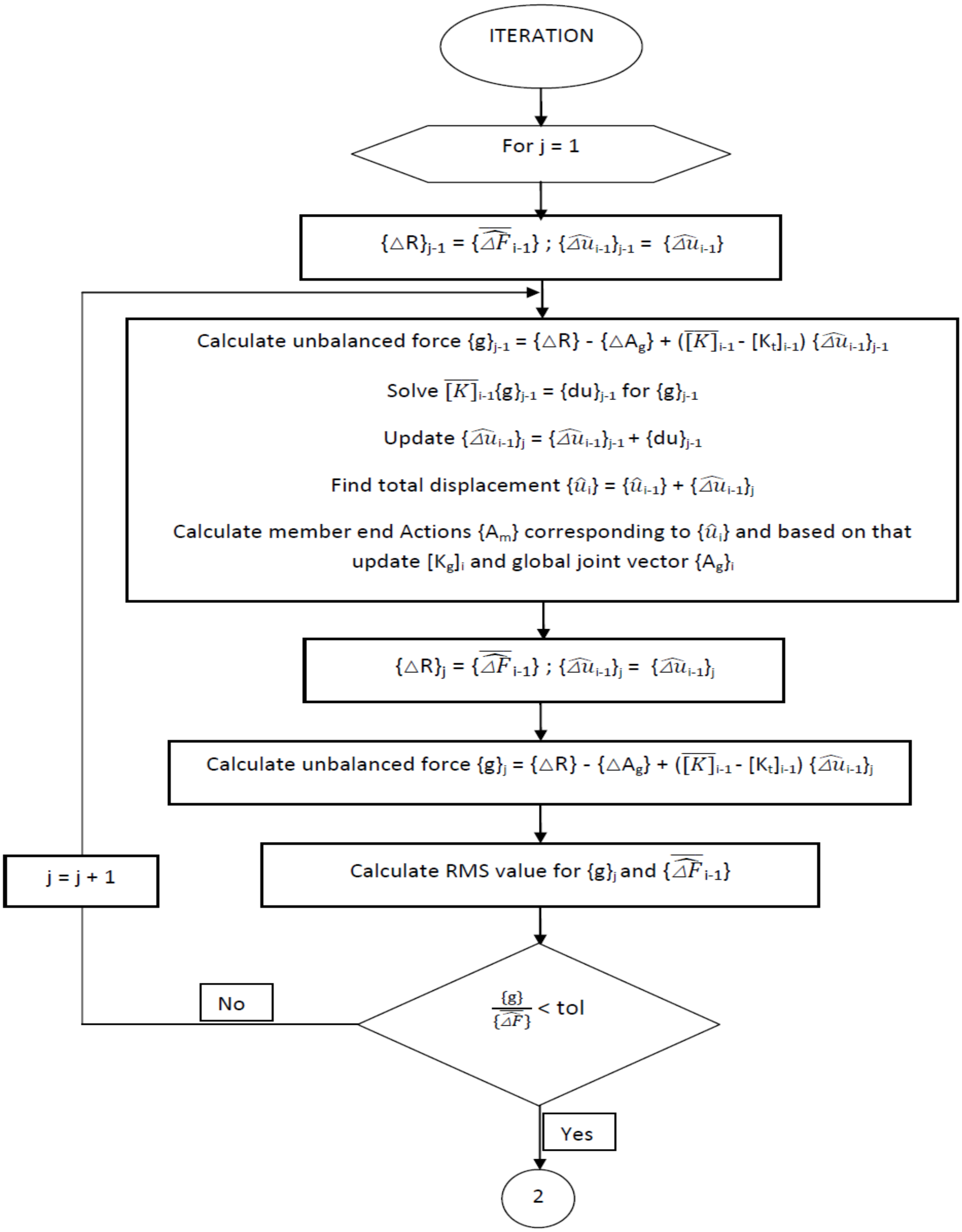

Figure 5.14: Flow Chart : Dynamic Nonlinear Analysis (3^{rd} Step)

5.8 Summary

In this chapter, Geometric stiffness matrix for space frame element is presented. Solution methods of nonlinear equation are discussed. Using incremental-iteration method, computer program for static and dynamic nonlinear analysis considering geometric nonlinearity are developed. Same procedure can be extended for material nonlinearity. The programs developed for nonlinear analysis are useful to find behavior of space frame subjected to wind force, earthquake forces or other abnormal loading like blast load or progressive collapse conditions.

Chapter 6

Nonlinear Analysis : Results and Discussions

6.1 General

Computer program development as discussed in chapter 5 are used used for geometric nonlinear analysis of space frame. In this chapter, space frames with varying height are analyzed by incorporating geometric nonlinearity. Static nonlinear analysis results in terms of load-displacement curves are presented. Nonlinear analysis is performed by updating only tangent stiffness matrix and by updating both tangent stiffness matrix and geometry of space frames. Results of static nonlinear analysis are compared with that of obtained FEM based software results. Dynamic nonlinear analysis is also performed and results in terms of time history responses are presented.

6.2 Results and Discussion

In this section, various frames are analyzed. Results are presented in form of load displacement plots and time history responses.

6.2.1 Single bay Single Storey Frame

Single bay single storey space frame as discussed in chapter 4 for linear analysis is considered here to illustrate nonlinear behavior.

- Plan Dimension : 5m×5m
- Storey Height : 8m
- Beam and Column Size : (200×200) mm
- Grade of Concrete : M25
- Poisson's Ratio (μ) : 0.2
- Support Condition : Fixed

6.2.1.1 Static Nonlinear Analysis Result

As nonlinear analysis is carried out in incremental manner, Loads are divided into small increments. Loading conditions for single storey frame is shown in Fig. 6.1.

A load factor(λ) of 10 is assumed which is used to decide the load increment i.e. Incremental load vector $\{\Delta F\} = \frac{\{F\}}{\lambda}$. If number of steps equal to λ, full load is applied. If number of steps are more than λ, load beyond actual load can be applied.

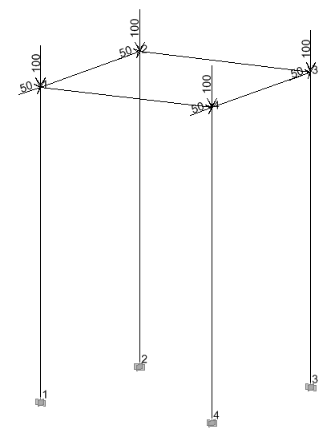

Figure 6.1: Space frame subjected to Loads

Here incremental load vector is calculated by dividing load vector by factor(λ). Response of space frame is measured in terms of displacement in X-direction for top node (3). Results of nonlinear analysis (NLA) is validated with FEM based software Results.

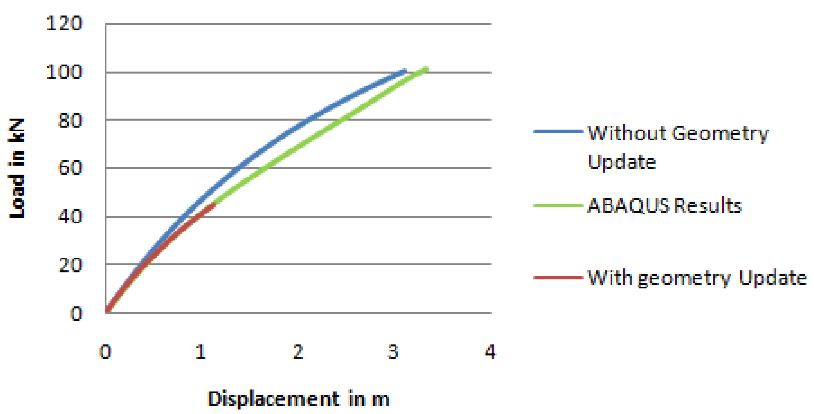

Figure 6.2: Load Displacement Curve for node 3 of first storey

Table 6.1: Displacement Results of NLA using Computer Program

Load (kN)	**Displacement in X-direction (m)**	
	NLA Without Geometry Update	**NLA With Geometry Update**
0	0.000	0.000
5	0.082	0.082
10	0.173	0.180
15	0.266	0.283
20	0.363	0.396
25	0.465	0.527
30	0.573	0.652
35	0.687	0.794
40	0.808	0.953
45	0.935	1.122
50	1.070	-
55	1.213	-
60	1.365	-
65	1.528	-
70	1.701	-
75	1.886	-
80	2.085	-
85	2.299	-
90	2.529	-
95	2.779	-
100	3.049	-

Table 6.2: Displacement Results of NLA using FEM based software

Load (kN)	**Displacement in X-direction (m)**
0.000	0.000
7.628	0.145
14.579	0.290
23.895	0.506
35.816	0.827
50.375	1.295
68.547	1.988
83.018	2.577
85.457	2.674
87.742	2.764
90.915	2.889
95.177	3.058
98.494	3.200
100.788	3.323

Results of nonlinear static analysis program are shown in Table 6.1. Both without geometry update and with geometry update conditions are considered. Results of nonlinear analysis with updating geometry condition are not available after load of 45kN due to some program error. Frame is also analyzed using FEM based software. Members of space frame are divided into number of elements for finite element analysis. Load displacement result are shown in Table 6.2. Results of analysis are plotted as shown in Fig. 6.2. For initial load increments, load deformation curves for all three analysis are almost same. After 20kN load, Results of NLA with geometry update and without geometry update are differed which indicate effect of deformed geometry. Hence it is required to update the geometry of structure at the end of each step for large deformation. Results of computer program for NLA with geometry update are good in agreement with FEM based results.

6.2.1.2 Dynamic Nonlinear Analysis Results

Space frame shown in Fig. 6.3 is subjected to gravity loads and time dependent lateral load.

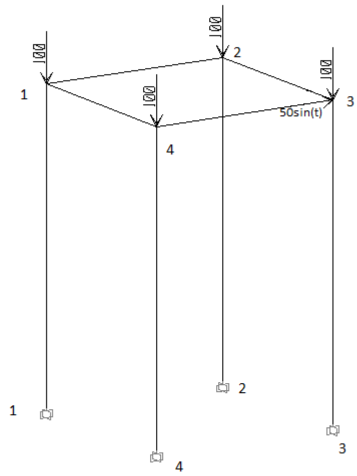

Figure 6.3: Loading Condition for Nonlinear Dynamic Analysis

Dynamic Analysis data,

- Damping Ratio $\xi = 5$ %
- $\omega = 8.82$ rad/sec
- Force : F(t) = 50 sin(t) at node 3 for t = 0 to t = 30 second(Figure 6.3)
- Time Increment : t = 0.1 second
- $\theta = 1.5$

Displacement, Velocity and Acceleration responses in x direction for node 3 of top storey is plotted. Gravity load is considered to trace second order effects. Comparison is made with linear analysis to get the idea of secondary effects.

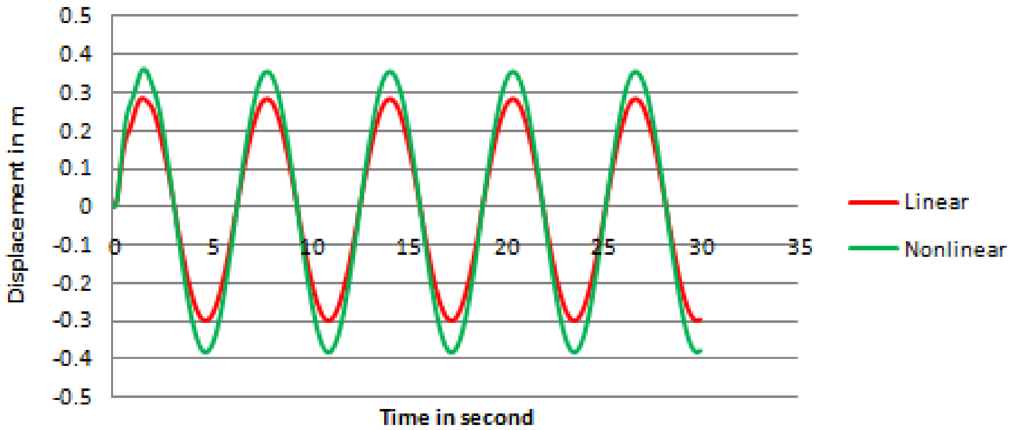

Figure 6.4: Time history Response for Displacement in X direction

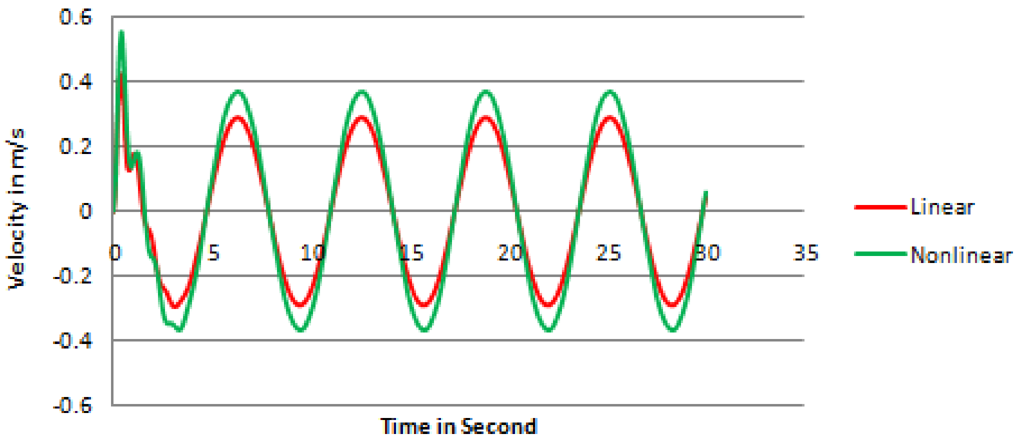

Figure 6.5: Time history Response for Velocity in X direction

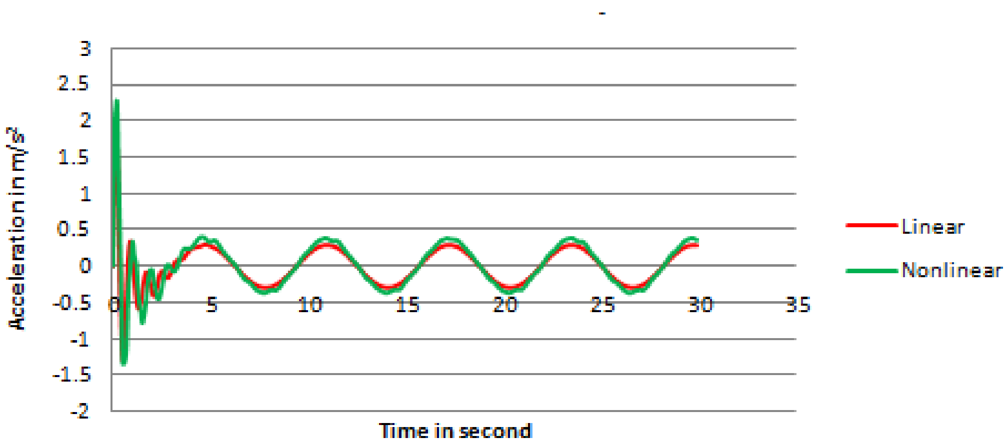

Figure 6.6: Time history Response for acceleration in X direction

Table 6.3: Comparison of maximum response of Linear and Nonlinear analysis

		Max u (m)	Max $\dot{u}$ (m/s)	Max $\ddot{u}$ (m/s^2)
"+Ve" value	**Linear**	0.283	0.424	1.988
	Nonlinear	0.363	0.553	2.282
"-Ve" value	**Linear**	-0.300	-0.296	-1.290
	Nonlinear	-0.384	-0.369	-1.351

Comparison between result of dynamic linear analysis as discussed in chapter 4 and results of dynamic nonlinear analysis are shown in Fig. 6.4, 6.5 and 6.6. Time history responses are changed due to inclusion of geometric nonlinearity. Second order effects are also traced in response curve. Maximum responses are also compared with linear analysis in Table 6.3. It is observed that due to second order effects, displacement is increased by 28%.

6.2.2 Multi-storey Space Frame

To understand the effect of height on nonlinear behavior, space frame with different heights are analyzed using program developed as discussed in chapter 5. Plan dimension of space frame as shown in Fig. 6.7, storey height, beam size are kept same for all space frame.

- Plan Dimension : (15×20) m
- Storey Height : 5m
- Beam Dimension : (300×450) mm
- Zone : V
- Soft soil condition
- I = 1 and R = 5
- Fixed at base

6.2.2.1 Static Nonlinear Analysis Results

In addition to self weight of structure, space frames are subjected to lateral load due to earthquake as per IS 1893. Gravity load analysis is performed in first step of analysis then lateral forces are applied incrementally. Results of analysis are presented in form of load displacement curve.

A load factor(λ) of 50 is assumed which is used to decide the load increment i.e. Incremental load vector $\{\Delta F\} = \frac{\{F\}}{\lambda}$.

Various models are considered in which sizes of columns are varying. Base shear for different models are also shown in Table 4.4. Results of displacements in x-direction corresponding to incremental loads are presented in tabular form. Displacements are calculated at node number 20 of top storey. (Fig. 6.7). Load vector calculations are also same as discussed in chapter 4.

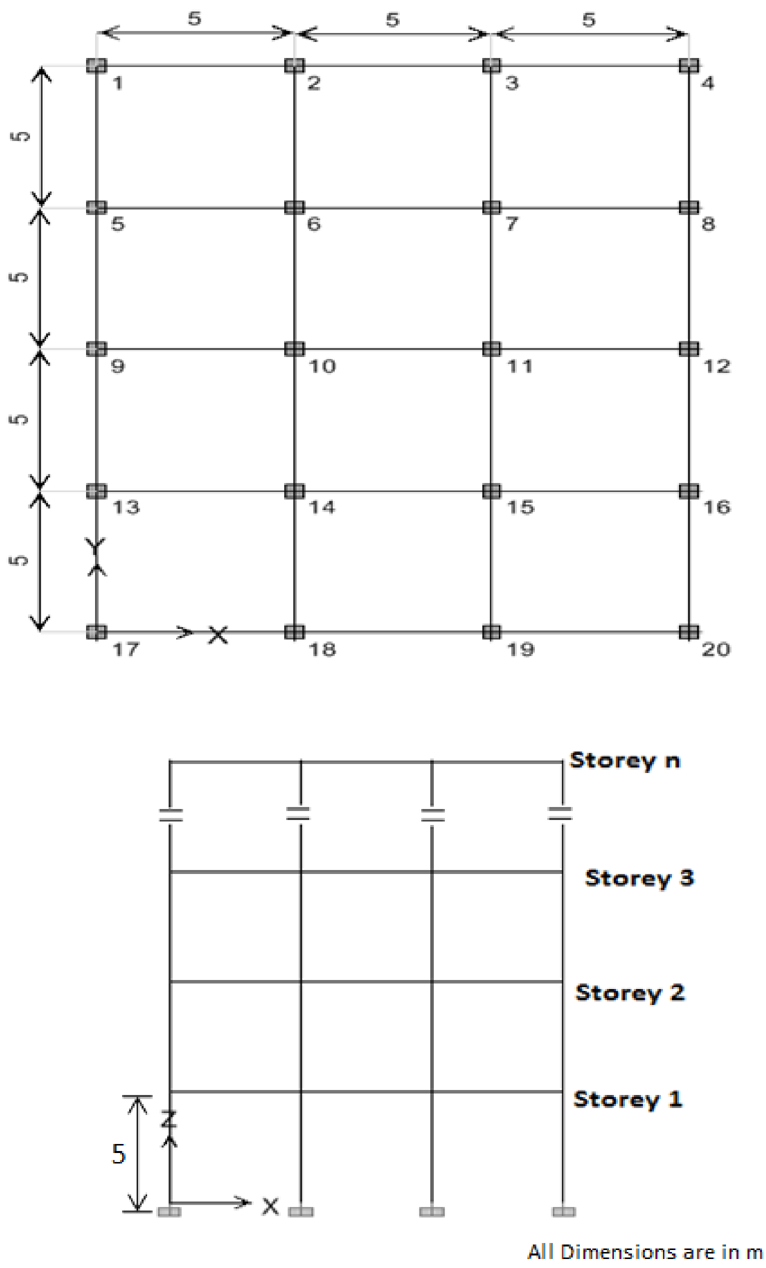

Figure 6.7: Plan and Elevation of multi-storey space frames (Common for all models)

Table 6.4: Load displacement data for 6 storey frame (V_{bx} = 3425 kN)

$\mathbf{F/V}_{bx}$	**Displacement in X direction (m)**	
	NLA Without Geometry Update	NLA With geometry Update
0	0	0
0.02	0.005	0.005
0.04	0.011	0.011
0.06	0.016	0.017
0.08	0.022	0.022
0.1	0.027	0.028
0.12	0.033	0.033
0.14	0.038	0.039
0.16	0.044	0.044
0.18	0.049	0.050
0.2	0.054	0.055
0.22	0.060	0.060
0.24	0.065	0.066
0.26	0.070	0.071
0.28	0.076	0.077
0.3	0.082	0.082
0.32	0.087	0.088
0.34	0.093	0.093
0.36	0.098	0.098
0.38	0.103	0.104
0.4	0.109	0.109
0.42	0.114	0.115
0.44	0.120	0.120
0.46	0.125	0.125
0.48	0.131	0.131
0.5	0.136	0.136
0.52	0.142	0.142
0.54	0.147	0.147
0.56	0.152	0.152
0.58	0.158	0.158
0.6	0.163	0.163
0.62	0.169	0.168
0.64	0.174	0.174
0.66	0.180	0.179
0.68	0.185	0.184
0.7	0.191	0.190
0.72	0.196	0.195
0.74	0.201	0.200
0.76	0.207	0.206
0.78	0.212	0.211
0.8	0.218	0.216
0.82	0.223	0.222
0.84	0.229	0.227
0.86	0.234	0.232
0.88	0.240	0.237
0.9	0.245	0.243
0.92	0.251	0.248
0.94	0.256	0.254
0.96	0.261	0.260
0.98	0.267	0.266
1	0.272	0.271

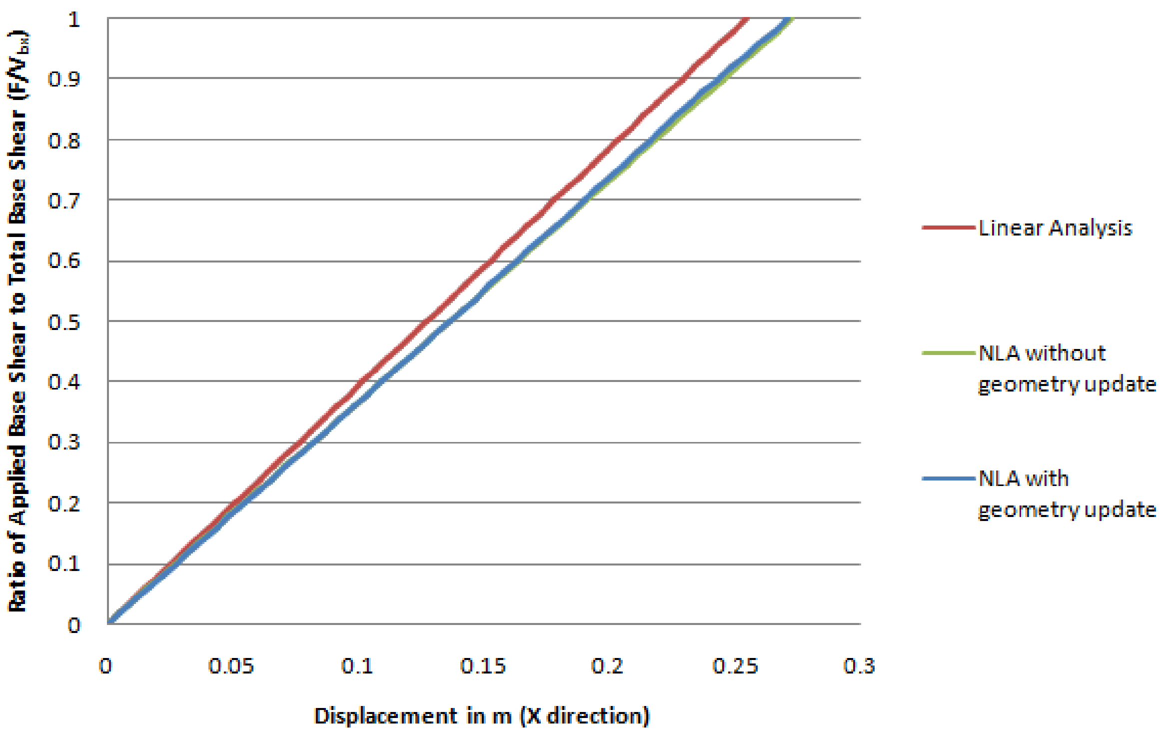

Figure 6.8: Load displacement curve at top for 6 storey frame

Results of nonlinear analysis of 6 storey frame is shown in Table 6.4. NLA is performed for both without geometry update and with geometry update. Results of both cases are almost same. Compared to displacement obtained from linear analysis, displacement obtained from nonlinear analysis is increased by 4.2 % for maximum lateral load due to earthquake.

Table 6.5: Load displacement data for 10 storey frame (V_{bx} = 2955.2 kN)

$\mathbf{F/V}_{bx}$	**Displacement in X direction (m)**	
	NLA Without Geometry Update	NLA With geometry Update
0	0	0
0.02	0.006	0.006
0.04	0.012	0.013
0.06	0.018	0.019
0.08	0.024	0.025
0.1	0.030	0.031
0.12	0.036	0.037
0.14	0.042	0.043
0.16	0.048	0.049
0.18	0.054	0.055
0.2	0.060	0.061
0.22	0.066	0.067
0.24	0.072	0.073
0.26	0.078	0.079
0.28	0.084	0.085
0.3	0.090	0.091
0.32	0.096	0.097
0.34	0.102	0.103
0.36	0.108	0.109
0.38	0.114	0.115
0.4	0.120	0.121
0.42	0.126	0.127
0.44	0.132	0.133
0.46	0.138	0.139
0.48	0.144	0.145
0.5	0.151	0.151
0.52	0.157	0.157
0.54	0.163	0.163
0.56	0.169	0.169
0.58	0.175	0.175
0.6	0.181	0.181
0.62	0.187	0.187
0.64	0.193	0.193
0.66	0.199	0.199
0.68	0.205	0.205
0.7	0.211	0.211
0.72	0.217	0.217
0.74	0.223	0.223
0.76	0.229	0.229
0.78	0.235	0.235
0.8	0.241	0.241
0.82	0.247	0.247
0.84	0.253	0.253
0.86	0.259	0.259
0.88	0.265	0.265
0.9	0.271	0.271
0.92	0.277	0.277
0.94	0.283	0.282
0.96	0.289	0.288
0.98	0.295	0.294
1	0.301	0.300

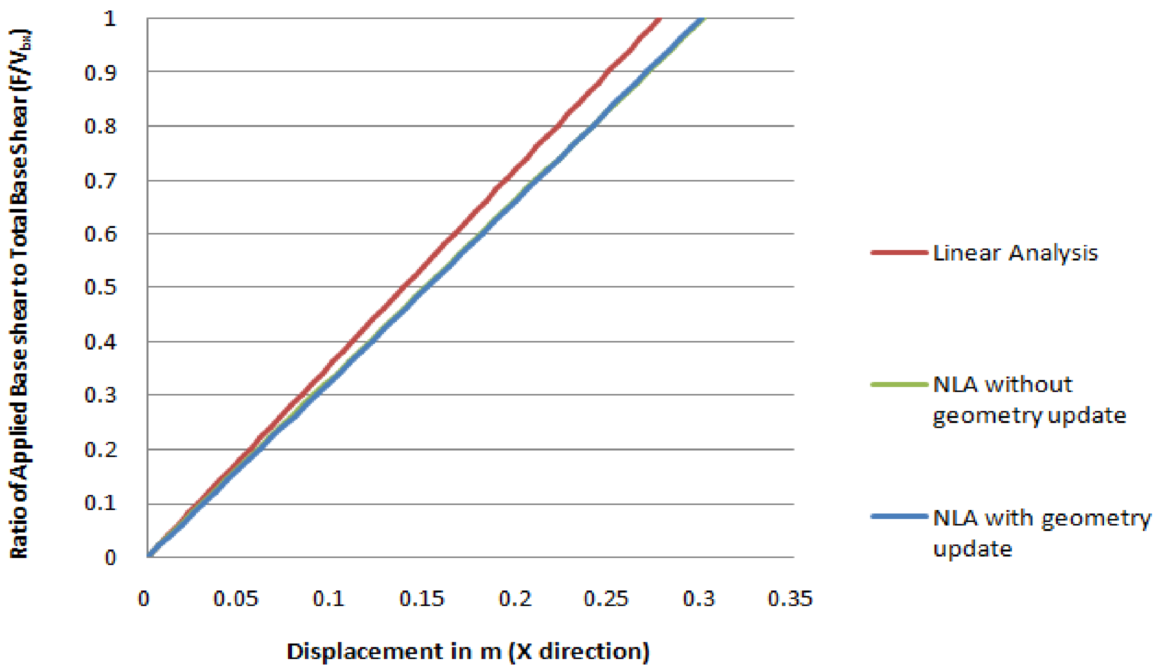

Figure 6.9: Load displacement curve at top for 10 storey frame

NLA results of load-displacement curve are shown in Table 6.5. For 10 storey frame results of NLA without geometry update and with geometry update are similar. At maximum lateral load due to earthquake, displacement obtained from nonlinear analysis is 7% more than displacement obtained from linear analysis.

Table 6.6: Load displacement data for 15 storey frame (V_{bx} = 3890.8 kN)

$\mathbf{F/V}_{bx}$	**Displacement in X direction (m)**	
	NLA Without Geometry Update	**NLA With geometry Update**
0	0	0
0.02	0.010	0.010
0.04	0.023	0.026
0.06	0.033	0.038
0.08	0.045	0.049
0.1	0.055	0.060
0.12	0.068	0.072
0.14	0.078	0.083
0.16	0.090	0.095
0.18	0.101	0.106
0.2	0.112	0.117
0.22	0.124	0.129
0.24	0.135	0.140
0.26	0.147	0.152
0.28	0.158	0.163
0.3	0.169	0.174
0.32	0.181	0.186
0.34	0.192	0.197
0.36	0.204	0.208
0.38	0.215	0.220
0.4	0.226	0.231
0.42	0.238	0.242
0.44	0.249	0.254
0.46	0.261	0.265
0.48	0.272	0.276
0.5	0.283	0.287
0.52	0.295	0.299
0.54	0.306	0.310
0.56	0.318	0.321
0.58	0.329	0.332
0.6	0.340	0.343
0.62	0.352	0.355
0.64	0.363	0.366
0.66	0.375	0.377
0.68	0.386	0.388
0.7	0.397	0.399
0.72	0.409	0.410
0.74	0.420	0.421
0.76	0.432	0.432
0.78	0.443	0.445
0.8	0.454	0.457
0.82	0.466	0.470
0.84	0.477	0.481
0.86	0.489	0.495
0.88	0.500	0.506
0.9	0.511	0.520
0.92	0.523	0.531
0.94	0.534	0.544
0.96	0.546	0.554
0.98	0.557	0.568
1	0.568	0.580

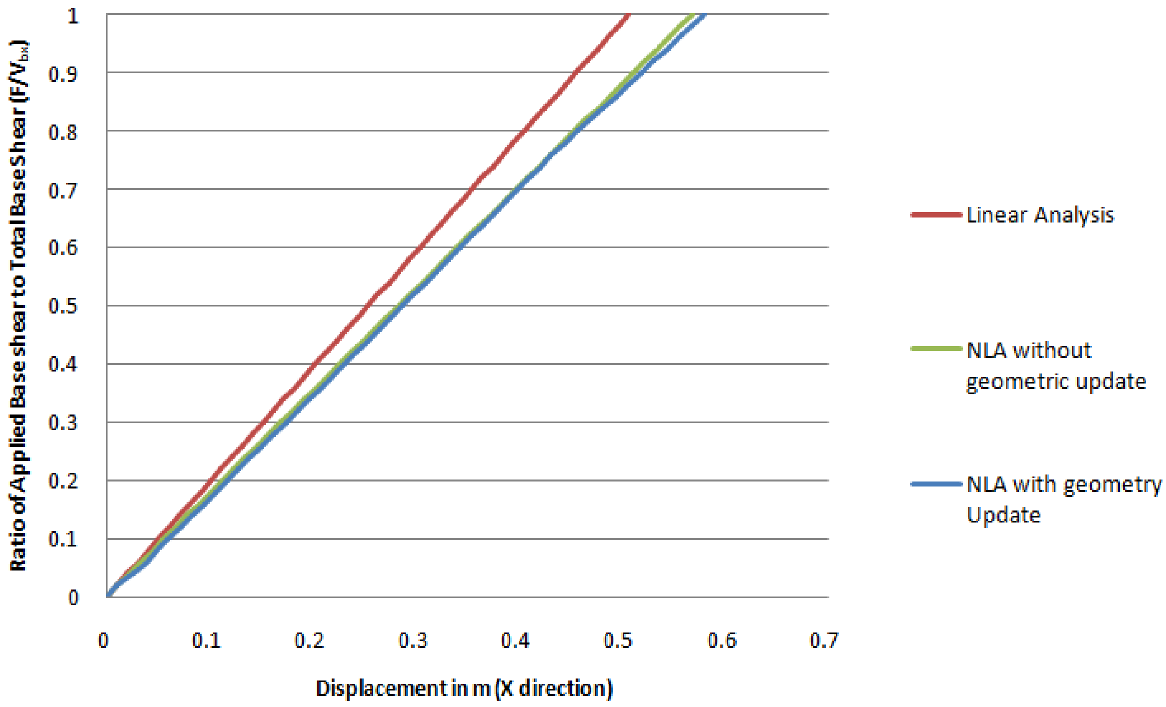

Figure 6.10: Load displacement curve at top for 15 storey frame

Load displacement results are shown in Table 6.6. For 15 storey frame also, Load-displacement curves for NLA without geometry update and with geometry update are almost similar. (Refer Fig. 6.10). Displacement based on nonlinear analysis for maximum lateral load is 13.73% more compared to displacement based on linear analysis.

Table 6.7: Load displacement data for 20 storey frame (V_{bx} = 4456.7 kN)

$\mathbf{F/V}_{bx}$	**Displacement in X direction (m)**	
	NLA Without Geometry Update	**NLA With geometry Update**
0	0	0
0.02	0.012	0.012
0.04	0.028	0.033
0.06	0.040	0.048
0.08	0.056	0.061
0.1	0.068	0.075
0.12	0.082	0.089
0.14	0.096	0.103
0.16	0.110	0.117
0.18	0.124	0.131
0.2	0.138	0.146
0.22	0.152	0.160
0.24	0.166	0.174
0.26	0.180	0.188
0.28	0.194	0.202
0.3	0.208	0.216
0.32	0.222	0.230
0.34	0.237	0.244
0.36	0.251	0.258
0.38	0.265	0.271
0.4	0.279	0.285
0.42	0.293	0.299
0.44	0.307	0.313
0.46	0.321	0.327
0.48	0.335	0.341
0.5	0.349	0.355
0.52	0.363	0.368
0.54	0.377	0.382
0.56	0.391	0.396
0.58	0.405	0.410
0.6	0.419	0.423
0.62	0.433	0.440
0.64	0.447	0.454
0.66	0.461	0.471
0.68	0.475	0.485
0.7	0.489	0.503
0.72	0.504	0.517
0.74	0.518	0.533
0.76	0.532	0.546
0.78	0.546	0.562
0.8	0.560	0.576
0.82	0.574	0.592
0.84	0.588	0.608
0.86	0.602	0.624
0.88	0.616	0.640
0.9	0.630	0.656
0.92	0.644	0.672
0.94	0.658	0.688
0.96	0.672	0.704
0.98	0.686	0.719
1	0.700	0.735

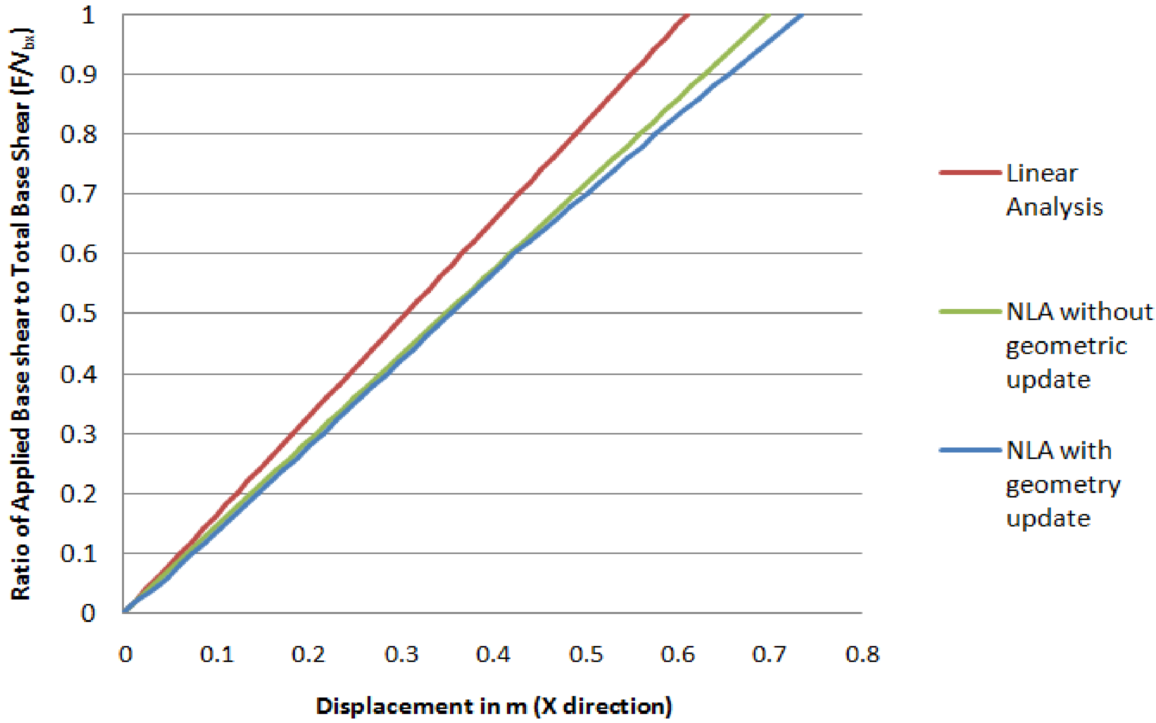

Figure 6.11: Load displacement curve at top for 20 storey frame

For 20 storey frame, NLA results are shown in Table 6.7. Load displacement curves are plotted. It is observed that after lateral load of $0.6V_{bx}$, effect of deformed geometry is observed and hence deflection results of NLA without geometry update and NLA with geometry update are deferred. Displacement based on NLA with geometry update is 5% more than displacement based on NLA without geometry update. At maximum lateral load of V_{bx}, deflection for nonlinear case is 19.9% more than that of linear analysis.

Table 6.8: Load displacement data for 25 storey frame (V_{bx} = 4456.7 kN)

$\mathbf{F/V}_{bx}$	**Displacement in X direction (m)**	
	NLA Without Geometry Update	NLA With geometry Update
0	0	0
0.02	0.015	0.015
0.04	0.036	0.045
0.06	0.052	0.066
0.08	0.072	0.083
0.1	0.091	0.100
0.12	0.109	0.119
0.14	0.128	0.138
0.16	0.146	0.157
0.18	0.165	0.175
0.2	0.183	0.194
0.22	0.202	0.212
0.24	0.220	0.231
0.26	0.239	0.249
0.28	0.257	0.268
0.3	0.276	0.286
0.32	0.294	0.305
0.34	0.313	0.323
0.36	0.331	0.342
0.38	0.350	0.360
0.4	0.368	0.378
0.42	0.387	0.397
0.44	0.405	0.415
0.46	0.424	0.433
0.48	0.442	0.451
0.5	0.461	0.469
0.52	0.479	0.492
0.54	0.498	0.511
0.56	0.516	0.535
0.58	0.535	0.554
0.6	0.553	0.577
0.62	0.572	0.595
0.64	0.590	0.617
0.66	0.609	0.636
0.68	0.627	0.658
0.7	0.646	0.680
0.72	0.664	0.703
0.74	0.683	0.726
0.76	0.701	0.749
0.78	0.720	0.772
0.8	0.738	0.794
0.82	0.757	0.817
0.84	0.775	0.839
0.86	0.794	0.861
0.88	0.813	0.883
0.9	0.831	0.906
0.92	0.850	0.927
0.94	0.868	0.949
0.96	0.887	0.971
0.98	0.905	0.993
1	0.924	1.014

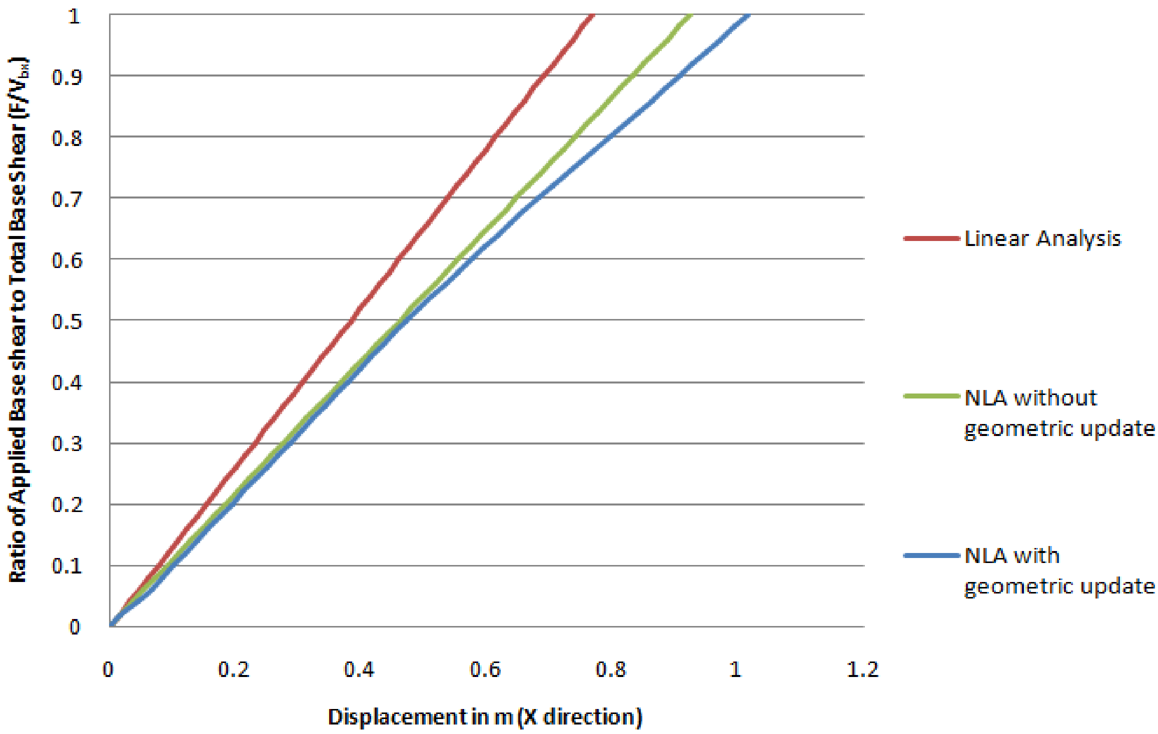

Figure 6.12: Load displacement curve at top for 25 storey frame

Load displacement results of developed programs are shown in Table 6.8. It is observed that after lateral load of $0.5V_{bx}$, effect of deformed geometry is observed . Displacement based on NLA with geometry update is 10% more than displacement based on NLA without geometry update. At maximum lateral load of V_{bx}, deflection for nonlinear case is 24.5% more than that of linear analysis. Hence it is important to consider large deformation condition for high rise building.

Table 6.9: Load displacement data for 30 storey frame (V_{bx} = 4728.3 kN)

$\mathbf{F/V}_{bx}$	**Displacement in X direction (m)**	
	NLA Without Geometry Update	NLA With geometry Update
0	0	0
0.02	0.020	0.020
0.04	0.051	0.068
0.06	0.072	0.103
0.08	0.099	0.125
0.1	0.126	0.148
0.12	0.152	0.175
0.14	0.179	0.204
0.16	0.205	0.232
0.18	0.232	0.258
0.2	0.258	0.285
0.22	0.285	0.312
0.24	0.312	0.339
0.26	0.338	0.366
0.28	0.365	0.392
0.3	0.392	0.419
0.32	0.418	0.446
0.34	0.445	0.472
0.36	0.472	0.499
0.38	0.498	0.525
0.4	0.525	0.561
0.42	0.552	0.591
0.44	0.578	0.627
0.46	0.605	0.655
0.48	0.632	0.691
0.5	0.659	0.730
0.52	0.685	0.769
0.54	0.712	0.808
0.56	0.739	0.848
0.58	0.765	0.887
0.6	0.792	0.925
0.62	0.819	0.964
0.64	0.845	1.002
0.66	0.872	1.040
0.68	0.899	1.077
0.7	0.925	1.114
0.72	0.952	1.151
0.74	0.979	1.188
0.76	1.005	1.224
0.78	1.032	1.260
0.8	1.059	1.295
0.82	1.085	1.331
0.84	1.112	1.366
0.86	1.139	1.400
0.88	1.165	1.435
0.9	1.192	1.469
0.92	1.219	1.502
0.94	1.245	1.536
0.96	1.272	1.569
0.98	1.299	1.602
1	1.326	1.634

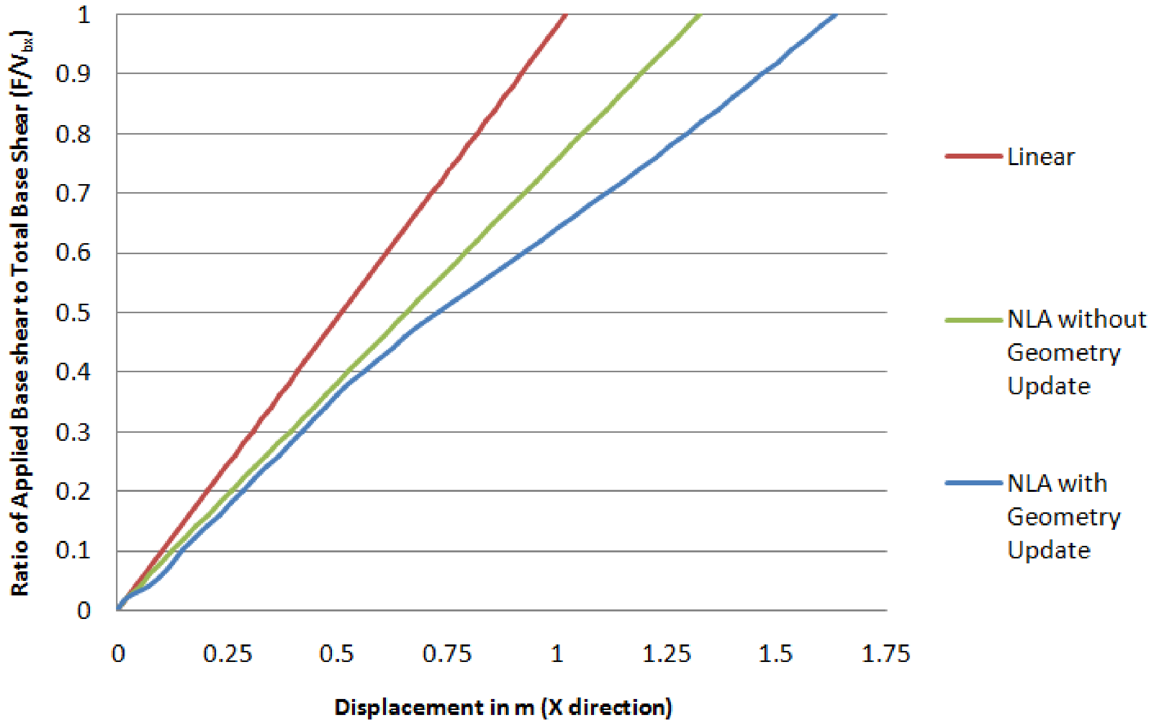

Figure 6.13: Load displacement curve at top for 30 storey frame

Load displacement data for 30 storey frame for nonlinear cases are shown Table 6.9. NLA without geometry update and with geometry update are performed based on developed computer program. It is observed that after lateral load of $0.4V_{bx}$, effect of deformed geometry is observed. Displacement based on NLA with geometry update is 18.9% more than displacement based on NLA without geometry update. At maximum lateral load of V_{bx}, deflection for nonlinear case is 38% more than that of linear analysis.

6.2.2.2 Dynamic Nonlinear Analysis Results

Dynamic nonlinear analysis of space frame with different height are performed. Effects of gravity loads are considered. Dynamic data of various models are same as discussed in Chapter 4. Dynamic forces are applied on top storey of each frame. Results are compared with linear analysis.

Dynamic Data for Each models are

- Forcing Function : F(t) = 500sin(t) for t = 0 to t = 15 second at node number 1, 4, 17 and 20 of top storey (Fig. 6.14).
- $\theta = 1.5$
- $\Delta t = 0.05$
- Responses in terms of displacements, velocity and acceleration in X - direction at node number 20 of top storey are recorded.

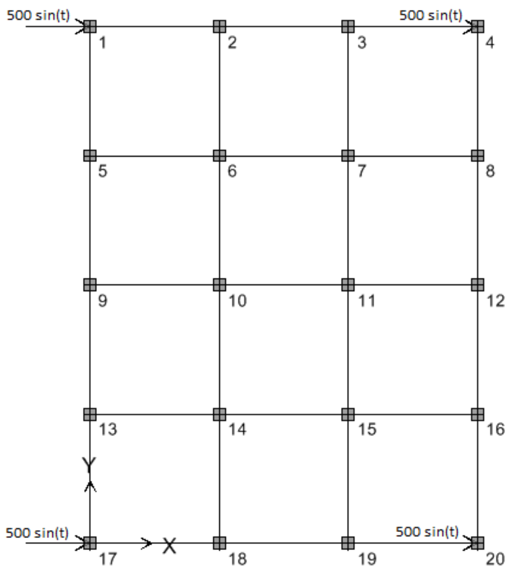

Figure 6.14: Magnitude and direction of time dependent Force

Results of various space frame models in form of time history curves are shown below,

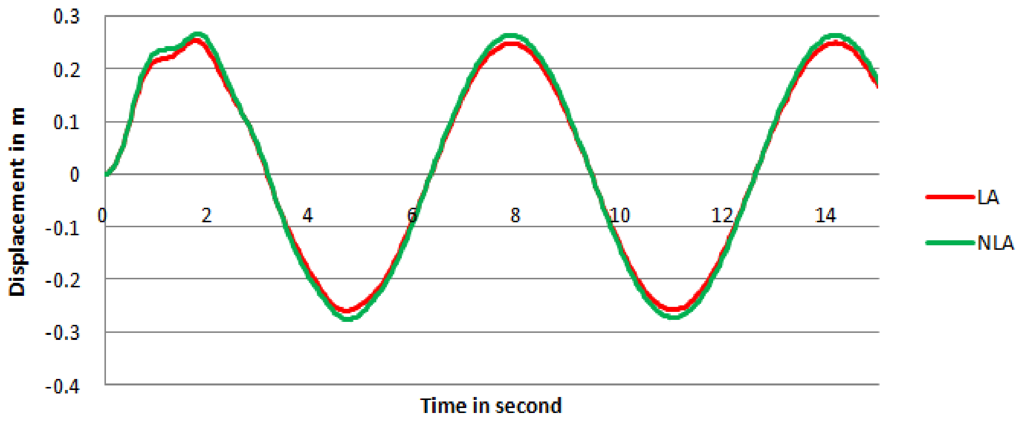

Figure 6.15: Comparison of Displacement Response for 6 Storey Frame

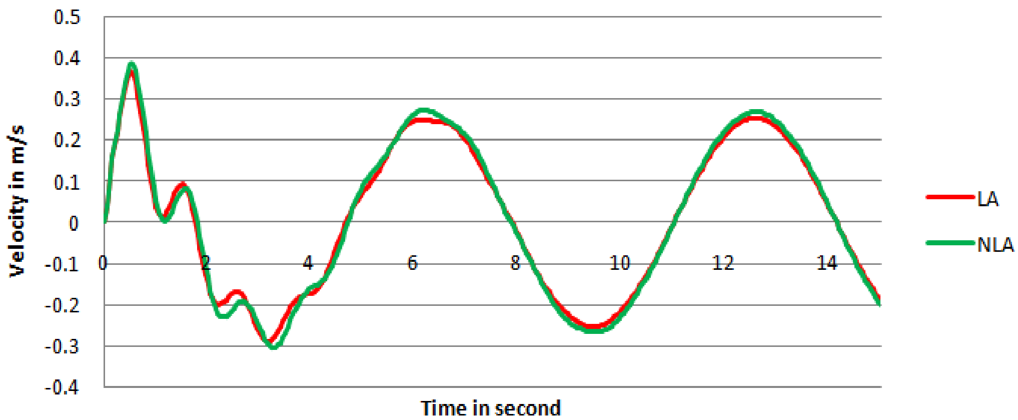

Figure 6.16: Comparison of Velocity Response for 6 Storey Frame

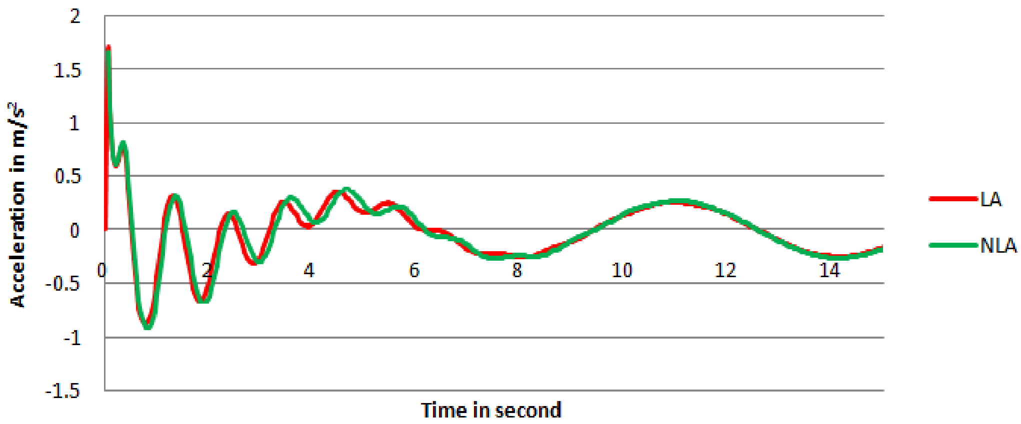

Figure 6.17: Comparison of Acceleration Response for 6 Storey Frame

Table 6.10: Comparison of Maximum Responses : 6 Storey Frame

		Max u (m)	**Max $\dot{u}$ (m/s)**	**Max $\ddot{u}$ (m/s^2)**
"+Ve" value	**Linear**	0.254	0.365	1.675
	Nonlinear	0.266	0.384	1.668
		Max u (m)	**Max $\dot{u}$ (m/s)**	**Max $\ddot{u}$ (m/s^2)**
"-Ve" value	**Linear**	-0.258	-0.289	-0.875
	Nonlinear	-0.275	-0.303	-0.916

Dynamic nonlinear analysis for 6 storey frame is performed using developed computer program. Also results of nonlinear and linear analysis are compared in Fig. 6.15, 6.16 and 6.17. For 6 storey frame, nonlinear responses are almost same as linear analysis. Maximum responses are also compared. Difference between responses based on nonlinear analysis and response based on linear analysis are approximately 4% - 7%.

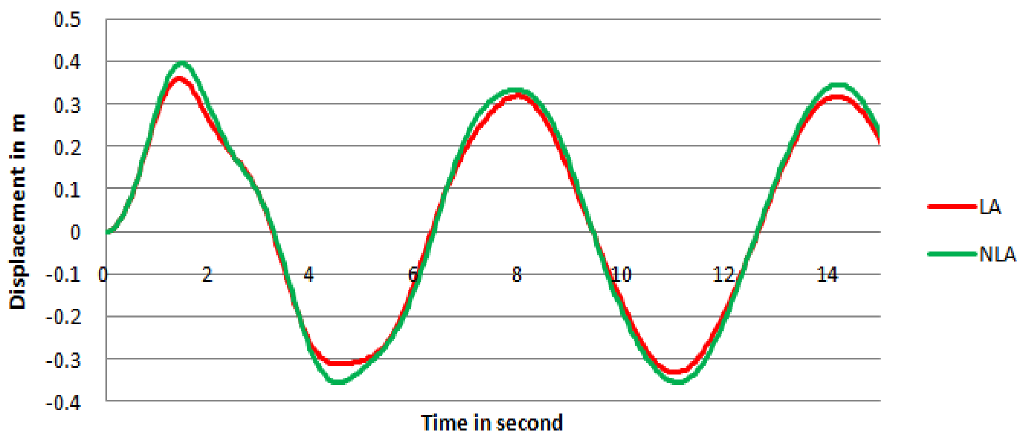

Figure 6.18: Comparison of Displacement Response for 10 Storey Frame

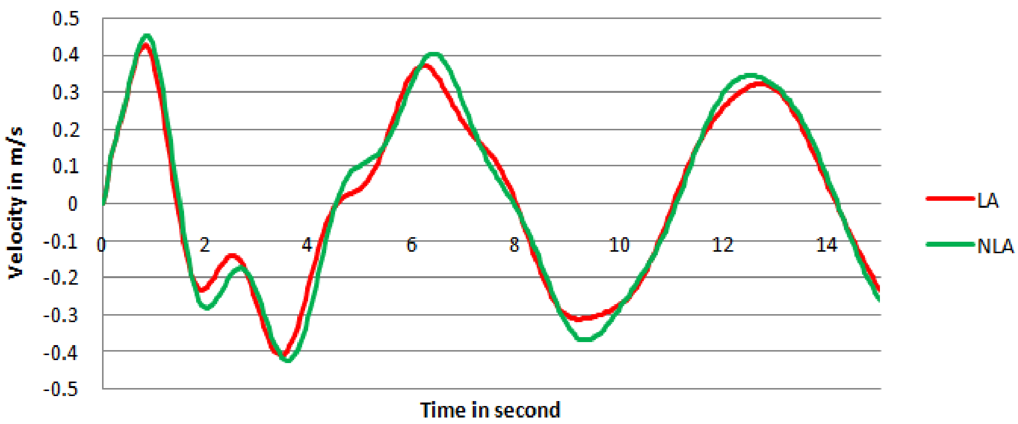

Figure 6.19: Comparison of Velocity Response for 10 Storey Frame

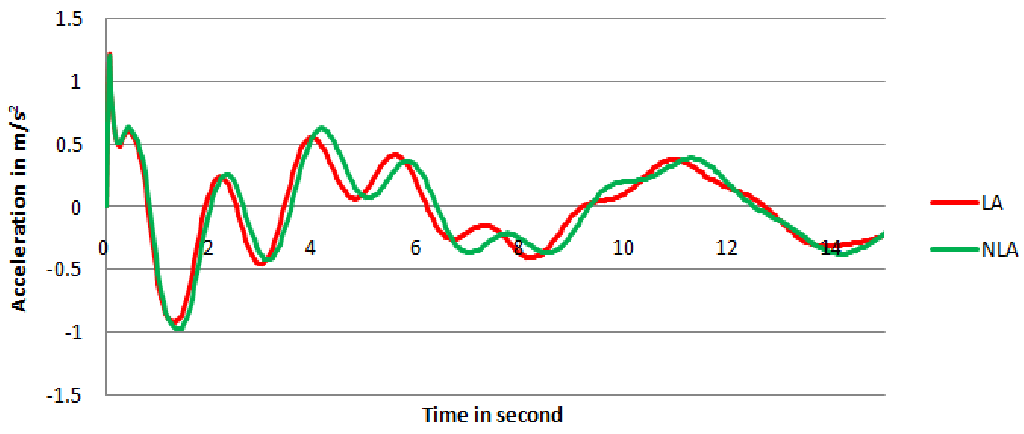

Figure 6.20: Comparison of Acceleration Response for 10 Storey Frame

Table 6.11: Comparison of Maximum Responses : 10 Storey Frame

		Max u (m)	**Max $\dot{u}$ (m/s)**	**Max $\ddot{u}$ (m/s^2)**
"+Ve" value	**Linear**	0.360	0.425	1.180
	Nonlinear	0.395	0.453	1.169
		Max u (m)	**Max $\dot{u}$ (m/s)**	**Max $\ddot{u}$ (m/s^2)**
"-Ve" value	**Linear**	-0.333	-0.410	-0.919
	Nonlinear	-0.355	-0.424	-0.980

Dynamic nonlinear analysis is performed for 10 storey space frame using developed computer program. Second order effects are also included by incorporating geometric nonlinearity. Results are shown in Fig. 6.18, 6.19 and 6.20. Velocity and acceleration based on nonlinear analysis and linear analysis are almost same. Maximum displacement, velocity and acceleration are presented in Table 6.11. Maximum displacement based on nonlinear analysis is 10% more compared to that of linear analysis.

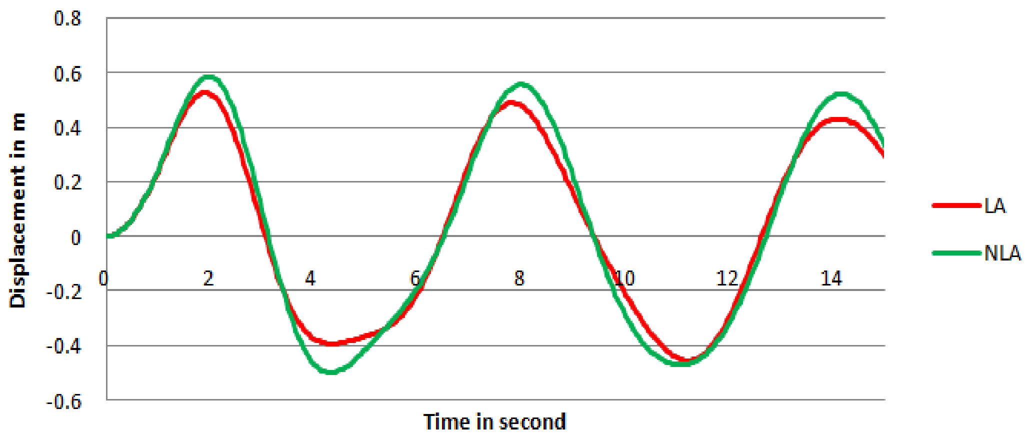

Figure 6.21: Comparison of Displacement Response for 15 Storey Frame

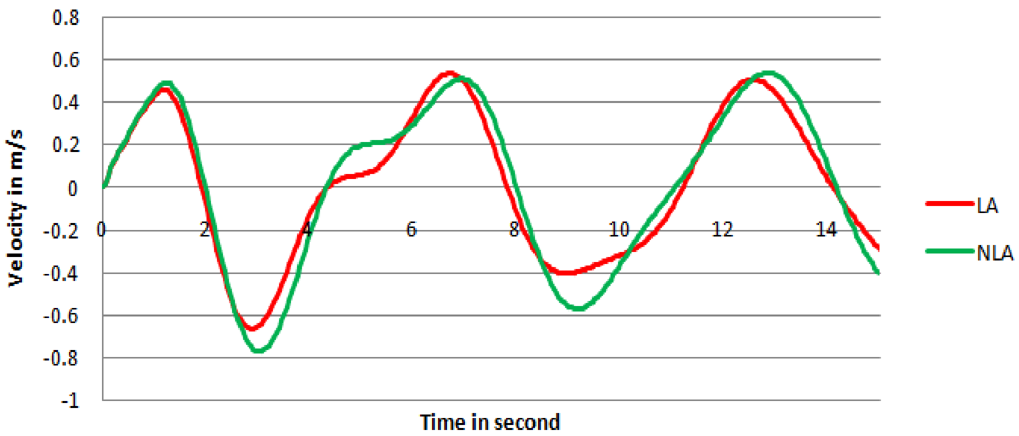

Figure 6.22: Comparison of Velocity Response for 15 Storey Frame

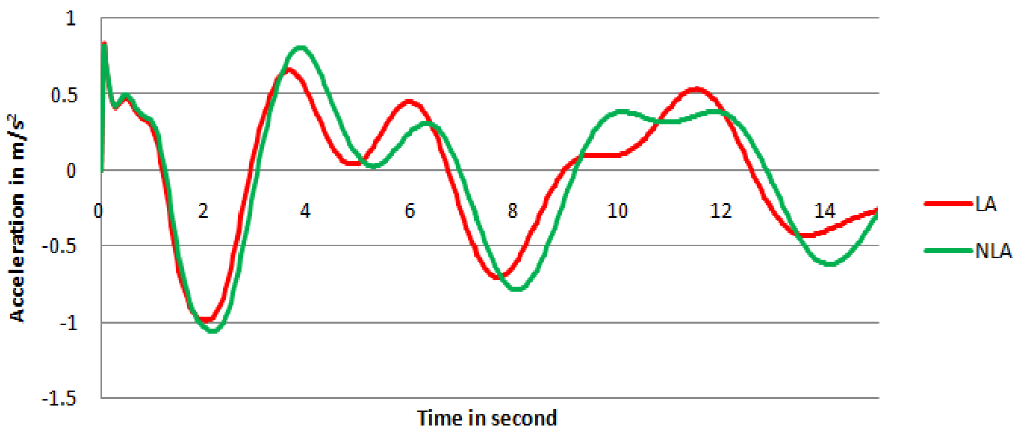

Figure 6.23: Comparison of Acceleration Response for 15 Storey Frame

Table 6.12: Comparison of Maximum Responses : 15 Storey Frame

		Max u (m)	**Max $\dot{u}$ (m/s)**	**Max $\ddot{u}$ (m/s^2)**
"+Ve" value	**Linear**	0.528	0.534	0.824
	Nonlinear	0.585	0.535	0.813
		Max u (m)	**Max $\dot{u}$ (m/s)**	**Max $\ddot{u}$ (m/s^2)**
"-Ve" value	**Linear**	-0.454	-0.665	-0.991
	Nonlinear	-0.498	-0.771	-1.065

Nonlinear dynamic analysis for 15 storey frame is performed and results in terms of time history plots are shown in Fig. 6.21, 6.22 and 6.23. Maximum displacement based on nonlinear analysis is increased by approximately 10% compared to maximum displacement based on linear analysis as discussed in chapter 4. Maximum velocity and acceleration based on nonlinear analysis are also increased by 16% and 7% respectively compared to that of linear analysis due to second order effects.

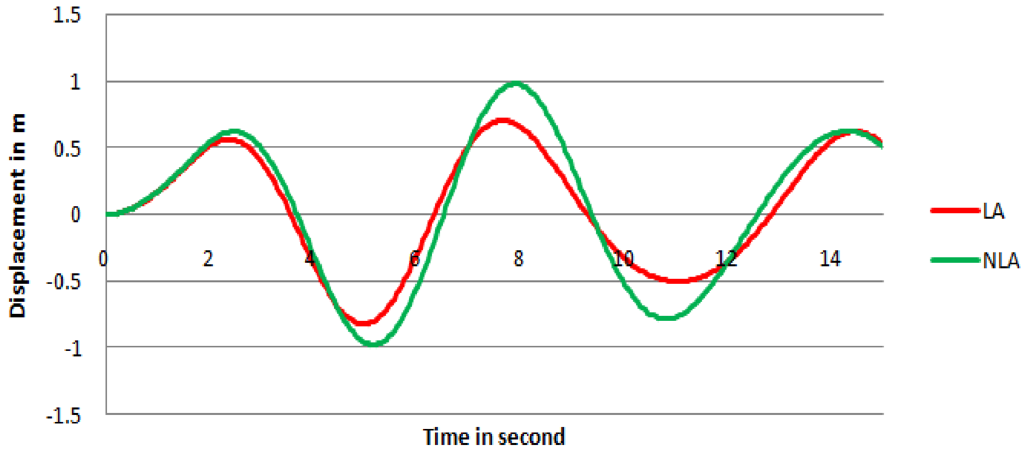

Figure 6.24: Comparison of Displacement Response for 20 Storey Frame

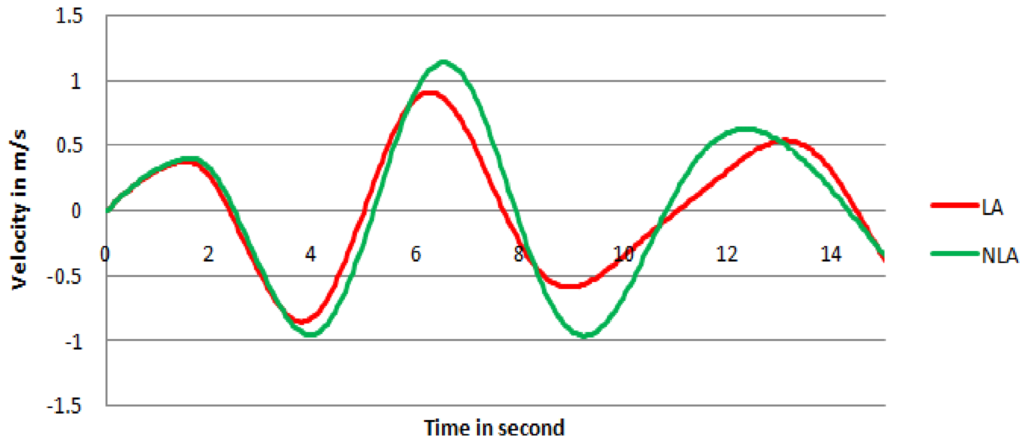

Figure 6.25: Comparison of Velocity Response for 20 Storey Frame

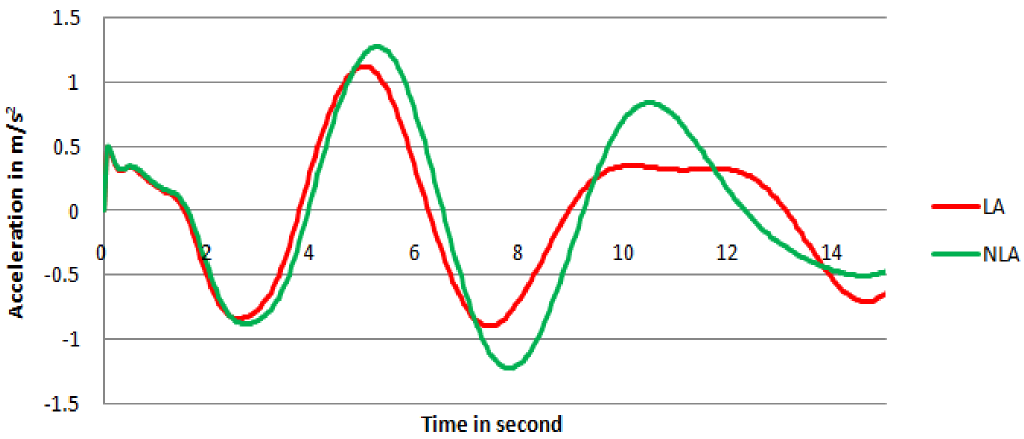

Figure 6.26: Comparison of Acceleration Response for 20 Storey Frame

Table 6.13: Comparison of Maximum Responses : 20 Storey Frame

		Max u (m)	**Max $\dot{u}$ (m/s)**	**Max $\ddot{u}$ (m/s^2)**
"+Ve" value	**Linear**	0.705	0.904	1.122
	Nonlinear	0.979	1.133	1.278
		Max u (m)	**Max $\dot{u}$ (d/s)**	**Max $\ddot{u}$ (m/s^2)**
"-Ve" value	**Linear**	-0.820	-0.859	-0.902
	Nonlinear	-0.975	-0.964	-1.233

Dynamic nonlinear analysis for 20 storey frame is performed using developed program. Comparison of nonlinear and linear analysis results are shown in Fig. 6.24, 6.25 and 6.26. For 20 storey frame, displacement obtained from nonlinear analysis is 39% more than that of linear analysis. Velocity and acceleration results are also affected due to large deformation. Maximum velocity and acceleration based on nonlinear analysis are 12% and 36% increased respectively compared to that of linear analysis. For given loading conditions, responses of 25 storey and 30 storey are diverging after some increments.

Results of space frame models suggest

- For lower storey frames or stiff frames have almost similar behavior when analyzed without updating geometry and with updating geometry. Also nonlinear analysis results are marginally higher than linear analysis results. Displacement based on static nonlinear analysis are 4% to 8% more compared to displacement based on static linear analysis.

- As storey height increases, variations between the results of analysis without updating geometry and with updating geometry are increased. It suggests that for flexible type of space frame, large deformation condition should be considered with second order effects. Displacements based on static nonlinear analysis are 20% to 40% more compared to that of static linear analysis.

- For dynamic loading condition, results of nonlinear analysis for stiff frame are similar to linear analysis whereas for for flexible type of frame, nonlinear analysis increases as height increases. It suggests that, second order effects are affecting behavior of flexible type of building. This can be understood by comparing nonlinear result with linear analysis result.

6.3 Summary

In this chapter, application of computer programs for nonlinear analysis of space frame with various height are discussed. Load displacement curves for static nonlinear analysis and time history plots for dynamic nonlinear analysis are presented. Nonlinear analysis is performed by considering with geometry update and without geometry update. Comparison is made between linear and nonlinear analysis results.

Chapter 7

Summary and Conclusion

7.1 Summary

Space frames are widely used structural system which requires rigorous analysis so that efficient and economical design can be proposed. It is difficult to understand exact load path as loads are transferred in three dimensional manner. New analysis techniques, which are based on performance of space frame, require detailed analysis which can predict accurate response. In this work, analysis of space frame structure is performed by considering linear and nonlinear condition to predict actual behavior. Linear analysis is performed for static and dynamic loading conditions. Computer Programs using C language are developed for static linear analysis, free vibration analysis and forced vibration analysis. Direct numerical integration method (Wilson Theta method) is used to obtain time history responses. For verification of linear analysis result, space frames are modeled and analyzed in ETABS.

Space frame are mostly having slender sections which can have large deformation. Additional forces are generated due to large deformation which can be incorporated by considering geometric stiffness matrix. Computer Program using C language is developed by extending linear static analysis procedure for nonlinear incremental analysis.

Newton-Raphson method is included to recover unbalanced forces induced at the end of each increment. Large strain and large deformation conditions are considered for nonlinear static analysis. Space frame subjected to incremental forces is analyzed using FEM based software for validation of nonlinear analysis results obtained from computer program. Space frames of varying heights under seismic loading are analyzed to understand effects of geometric nonlinearity. Developed computer programs are applicable for analysis of large size space frames. Computer program using C language for nonlinear dynamic analysis is also prepared and analysis results for space frames are presented.

Single bay single storey frame subjected to gravity and lateral loads is analyzed for static condition. Loads are applied incrementally and results are calculated for nonlinear case. Results of static nonlinear analysis are compared with FEM based software results. For dynamic loading condition, time dependent lateral force is applied at top storey. Dynamic response in form of displacement, velocity and acceleration are plotted in terms of time history curve. Multi-storey frames like 6 storey, 10 storey, 15 storey, 20 storey and 30 storey with plan dimension (15m $\times$ 20m) and storey height 5m are analyzed for static and dynamic case to understand effect of height on behavior of frame. Self weight of all members are calculated and applied at each node. Equivalent earthquake load as per IS 1893(Part 1):2002 is considered for each space frame and applied incrementally for static nonlinear analysis. Base shear v/s displacement are plotted using computer program results and compared with linear analysis. For dynamic nonlinear analysis, time dependent sinusoidal forces are applied at top storey of each frame. Time history curves are plotted for both linear and nonlinear case.

7.2 Conclusion

- **Nonlinear Static Analysis of Space Frame**

  Based on geometric nonlinear static analysis of regular space frame subjected to seismic forces as per IS 1893, following conclusions are derived.

  – For low rise space frame, difference between displacement based on nonlinear static analysis without updating geometry and with updating geometry is small. In present study, displacement at top for 6 storey and 10 storey frames under lateral load due to earthquake are almost similar for NLA without and with geometry update. This suggest that analysis of stiff frame can be performed by updating geometric stiffness matrix at the end of each step. This will reduce computational efforts.

  – As height of frames increase, difference in displacement, based on nonlinear static analysis without geometry update and with geometry update, is significant. For 15 storey, 20 storey, 25 storey and 30 storey space frames under lateral load due to earthquake, displacements based on nonlinear analysis with geometry update are 2%, 5%, 10% and 23% more compared to displacement based on nonlinear analysis without geometry update respectively. Hence it is required to update geometry of space frame along with geometric stiffness matrix for geometric nonlinear analysis of flexible space frames.

  – When ratio of height to larger lateral dimension(H/L) is less than 4, nonlinear analysis can be performed by only updating geometric stiffness matrix. For (H/L) greater than 4, geometric nonlinearity as well as effect of deformed geometry should be included to trace actual nonlinear load-displacement curve.

  – In present study, displacements based on nonlinear static analysis for 6 storey, 10 storey, 15 storey, 20 storey, 25 storey and 30 storey frames are approximately 4%, 7%, 14%, 20%, 25% and 38% more than the displacements based on linear static analysis respectively. It suggests that for high rise frame, second order effects are affecting the behavior and hence geometric nonlinearity should be considered.

- Gravity loads need to be considered at the initial steps of analysis to track nonlinear behavior of space frames for static and dynamic loading condition.

- **Nonlinear Dynamic Analysis of Space Frame**

  Based on geometric nonlinear analysis of regular space frame subjected to sinusoidal load at top storey, following conclusions are derived.

  - Maximum displacement based on nonlinear dynamic analysis for 6 storey, 10 storey, 15 storey and 20 storey frames are approximately 5%, 10%, 12% and 38% more compared to displacement based on linear dynamic analysis respectively.
  - Velocity and acceleration responses are affected due to geometric nonlinearity. For 6 storey, 10 storey, 15 storey and 20 storey, maximum velocity based on nonlinear dynamic analysis are 5%, 7%, 16% and 25% more compared to maximum velocity based on linear dynamic analysis and maximum accelerations based on nonlinear dynamic analysis are 5%, 7%, 7% and 36% more compared to maximum acceleration based on linear dynamic analysis. Response of dynamic analysis are also affected by frequency of applied force.

7.3 Future Scope of Work

The work presented in this report can be extended in future as follows

- Development of analysis procedure for static and dynamic conditions in which both the material and geometric nonlinearities are incorporated.
- Study nonlinear dynamic effects of earthquake and wind for space frame structure.
- Development of computer program for nonlinear analysis using displacement control method.
- Effect of abnormal loading conditions like progressive collapse conditions, blast load etc can be studied using computer program developed in this study.
- Effect of material and geometric nonlinearity on irregular building configuration can be studied.

Appendix A

List of Paper Published / Communicated

List of Communicated

- Jaymin R. Desai, Paresh V. Patel, "Second order Elastic Analysis of Space Frame ", 9^{th} biennial event, Structural Engineering Convention 2014, IIT Delhi. 22^{nd} - 24^{th} December, 2014.

References

[1] Yang Y. B. and McGuire W. , "Stiffness Matrix for Geometric Nonlinear Analysis", Journal of Structural Engineering, ASCE, Volume 112, pp. 853-877, April 1986.

[2] Chang J. T. , "Derivation of the higher-order stiffness matrix of a space frame element", Finite elements in analysis and design, Volume 41, pp. 15-30, 2004.

[3] Kassimali A and Abbasnia R., "Large deformation analysis of elastic space frames", Journal of Structural Engineering, ASCE, Volume 117, pp. 2069-2087, July 1991.

[4] Teh L. H. and Clarke M. J., "Symmetry of tangent stiffness matrices of 3D elastic frame", Journal of Engineering Mechanics, ASCE, Volume 125, pp. 248-251, February 1999.

[5] Chajes A. and Churchill J. E., "Nonlinear Frame analysis by Finite Element Method" , Journal of Structural Engineering, ASCE, Volume 113, pp. 1221-1235, June 1987.

[6] Yang Y. B. et al., "Incrementally small-deformation theory for nonlinear analysis of structural frames", Engineering Structures, Volume 24, pp. 783-798, 2002.

[7] Suk J. W. et al., "A predictor algorithm for fast geometrically-nonlinear dynamic analysis", Computer methods in applied mechanics and engineering, Volume 192, pp. 2521-2538, 2003.

[8] Thai H. T. and Kim S. E., "Nonlinear inelastic analysis of space frames", Journal of Constructional Steel Research, pp. 585-592, 2011.

[9] Nguyen P. C. and Kim S. E., "Nonlinear elastic dynamic analysis of space steel frames with semi-rigid connections" , Journal of Constructional Steel Research, pp. 72-81, 2013.

[10] Abbasnia R. and Kassimali A., "Large deformation elastic-plastic analysis of space frames", Journal of Constructional Steel Research, pp. 275-290, 1995.

[11] Liew J. R. et al., "Improved nonlinear plastic hinge analysis of space frame structures", Engineering Structures, pp. 1324-1338, 2000.

[12] Jiang X. M. et al., "Spread-of-plasticity analysis of three-dimensional steel frames", Journal of Constructional Steel Research, pp. 193-212, 2002

[13] Chan S. L., "Large deflection dynamic analysis of space frame", Computers and Strucutres, Vol. 58, pp. 381-387, 1996.

[14] Mansouri I. and Saffari H., "An efficient nonlinear analysis of 2D frames using a Newton-like technique", Archives of civil and mechanical engineering, pp. 485-492, 2012.

[15] Crisfield M. A., Non-linear Finite Element Analysis of Solids and Structures, Volume 1, John Wiley and Sons, April 2000.

[16] Li G. Q. and Li J. J., Advanced Analysis And Design Of Steel Frames, John Wiley and Sons, 2007.

[17] Yang Y. B. and Kuo S. R., Theory and analysis of nonlinear framed structures, Prentice Hall, 1994.

[18] Paz M. and Leigh W., Structural Dynamics - Theory and Computation, Fifth Edition, 2004.

[19] Chopra A. K., Dynamics of Structures - Theory and Applications to Earthquake Engineering, Prentice Hall, 1995.

[20] IS : 1893 (Part-1) : 2002 Indian Standard : Criteria for Earthquake Resistant Design of Structures, Fifth Revision.

[21] IRC : 112 (2011) Code of Practice for Concrete Road Bridges, Indian Road Congress.