

STABILITY ANALYSIS OF FRAME TUBE TALL BUILDINGS

by

Amit Urs

A thesis submitted in partial fulfillment of the requirements for the degree of
Masters of Science

Department of Civil and Environmental Engineering
Worcester Polytechnic Institute.
Worcester, MA. 01609.

Oct, 2002

Approved by _____

Thesis Advisor: Prof. P. Jayachandran

Prof. Leonard D. Albano

Prof. F. Hart

Head of the Department

Civil & Environmental engineering

Worcester Polytechnic Institute, MA.

DATE_____

Worcester Polytechnic Institute

Abstract

STABILITY ANALYSIS OF FRAME
TUBE TALL BUILDINGS

by Amit Urs

Thesis Advisor: Professor P. Jayachandran

Department of Civil Engineering

The frame tube buildings have been the most efficient structural system used for building which is in the range of 40-100storey. The soaring heights and the demanding structural efficiency have led to them having smaller reserves of stiffness and consequently stability.

In this thesis a Non-linear analysis and stability check of frame-tube building is done. Nonlinear analysis offers several options for addressing problems of nonlinearity and in this work focus is on Geometric Non-linearity. The main sources can be identified as P- Δ effect of gravity loading acting on a transversely displaced structure due to lateral loading and can also be due to member imperfections, such as member camber and out of plumb erection of the frame. During analysis the element response keep continuously changing as a function of the applied load so simple step computing methods have been employed instead of direct analytical methods. The problem here is dealt in a piece wise linear way and solved. In this thesis a program using the matrix approach has been developed. The program developed can calculate the buckling load and can do Linear and Non-linear analysis using the Mat-lab as the computing platform.

Numerical results obtained from the program have been compared with the Finite Element software Mastan2. The comparative solutions presented later on in the report clearly prove the accuracy of the program and go on to show, how exploiting simple matrix equation can help solve the most complex structures in fraction of seconds.

The program is modular in structure. It provides opportunity for user to make minor manipulation or can append his own module to make it work for his specific needs and will get reliable results.

ACKNOWLEDGMENTS

The author wishes to express sincere appreciation to Professors P. Jayachandran for his thesis advising. The author also would like to thank Prof. William. W. Durgin for the financial assistance and encouragement.

In addition, author wishes to thank his mom Mrs. Kaivalya V.K. Urs, Aunt Meera, Aunt Sudha, his sister Ms. Geeta Urs and his Dad Mr. Venkatakrishna Urs, for all the blessings and support they have provided him. Author wishes to thank his friend Mr. Badri Krishna for all the help and valuable assistance. Author would like to thank the members of Weidlinger Associates for their trust, support and patience.

Mainly, Author wishes to express his humble gratitude to "PARAMATMA" without whose mercy he could not have done anything.

TABLE OF CONTENTS

Abstract	i
Acknowledgement.....	iii
List of figures.....	vi
Preface	vii
Chapter I: Introduction.....	1
Introduction.....	1
Objective.....	2
Scope	3
Previous Work.....	4
Chapter II: Conceptual Framework	8
Matrix methods.....	8
Stiffness method.....	8
Level of analysis	21
Different approach to nonlinear analysis	21
Load control method	21
Chapter III: Tall buildings.....	27
Introduction.....	27
Approaches to analysis.....	31
Reduction modeling.....	32
Frame tube building.....	37
Mode of behavior	38
Structural Analysis	41
Reduction modeling of Frame tube building.....	44
Chapter IV: Findings and Discussion.....	51
Description of Findings	51
Recommendations for future research.....	55

Glossary	57
References.....	58
Appendix A:.....	61
Manual	61
Algorithm.....	62
Tutorial 1.....	63
Tutorial2.....	68
Appendix B:	72
Frame tube building reduction models.....	72
Frame tube building 40 Stories.....	88

LIST OF FIGURES

<i>Number</i>	<i>Page</i>
1. Nodal forces and displacement.....	10
2. Nodal forces.....	11
3. Load Deflection curve.....	16
4. Finite strain of planar element.....	19
5. Prototype rigid frame.....	44
6. Equivalent lumped girder.....	45
7. Axial stress distribution in column.....	51
8. deformation of flange frame.....	52
9. Frame Tube structural plan.....	58
10. connection details.....	59
11. Eulers column	33
12. Side sway prevented.....	33
13. Side sway permitted	34

PREFACE

The high speed precise computing and increased memory of the computers has made it possible to solve complex models. Finite Element Method and Matrix method are the two methods which show great compatibility for computing process and has become the most power tool in many engineering branches. In this work an effort has been made to develop programs that will do an analysis of highly indeterminate structures. In this work a frame tube building of 40 stories high, which is highly indeterminate, has been selected.

Tall buildings like frame tube buildings are usually 40 - 100 stories high. Their analysis is extremely complex due to highly degree of indeterminacy and hand calculation is very tough. In chapter 1, an introduction to work is given followed by clearly laid out objectives of the study and its scope.

In Chapter 2, a conceptual frame work of the thesis has been laid out. As the work is pivoted on the matrix method of structural analysis, different techniques and approaches to analysis of planar frames have been presented. Linear elastic analysis, nonlinear analysis and stability analysis have been discussed appropriate enough, which form the basis for the programs developed.

In Chapter 3, an introduction to tall buildings followed by a detailed discussion on structural action of frame tube buildings has been presented. Methods used for mathematical modeling along with logics involved in the structural behavior of frame tube building have been discussed.

In chapter 4, results and conclusion of the work have been presented.

In Appendix A and Appendix B, Manual, results and comparative values have been tabulated. Tutorial problems have been presented and these problems should aid for a proper understanding of the programs.

Chapter 1

INTRODUCTION

Introduction:

In the past, designers had no need to accommodate for terror attacks from the air, but now that this threat is a reality; designers have to look for sensible ways to defend our high-rises

All the evidence so far points that a combination of the impact and the fires triggered the collapse of the World Trade Center buildings; we must now ask, what we can do in the future to prevent another failure like this one. Some believe that nothing can, or should, be done. While some opine that "nothing is designed or will be designed to withstand that [kind of] fire." Numbers of theories have evolved to explain exact reason for the collapse of the Twin towers; some believe the impact alone was the actual cause of the failure by providing evidence that the structure failed quicker than the time required for the fireproofed steel to fail while some argue that the fires where to blame, the steel-tube construction of the towers could not resist the intensity of the fires resulting from the energy released by the 2400 gallons jet fuel that brought the fires to higher than normal temperatures, as high as 2000 degrees Fahrenheit. But combination of these two appears to be logical explanation to the tragedy on 9/11. Bottom line is one or combination of them may have caused the structure to fail, from now on it would be necessary to do a thorough stability check of tall buildings for all possible circumstances.

In “Limit state design”, buildings are designed for limit state of strength and limit state of serviceability, leaving the structure with minimum reserve energy. If it is the design case of a low-rise structure subjected to low horizontal loads, deflections are small with insignificant change in geometry of the structure. Thus the reserve energy of the building is sufficient to bring back the structure to equilibrium state after the load is removed, rendering stability to the structure as a whole. There by first order analysis of structures to satisfy the equilibrium conditions is sufficient to verify the design. In case of tall buildings about 40 to 50 story’s high horizontal loads cause huge deflection of the building and a significant change in the geometry of the structure. The large deformation and low reserve energy can prove to be cataclysmic if this small energy in the structure fails to resist loads. Thus failing to satisfy the equilibrium conditions the structure could become highly unstable. To predict the exact behavior of the structure a second order analysis of the structure has to be conducted. In this thesis I have devolved a program to do first order, second order analysis and also to calculate the elastic critical buckling loads of planar framed structures.

Objectives

The main objective of the thesis work is to develop software to check the stability of structures and also to do a second order analysis of the 40 story Frame-Tube Buildings. Matrix approach is used in the analysis. It is realized that the fulfillment of the following sub-objectives would in turn fulfill the main objective.

1. To understand the concepts of Matrix analysis of structure.
2. To study the behavior of Frame tube buildings.
3. To study various technique in nonlinear analysis of frame-tube buildings.

4. To develop a Mat Lab program using the principles of Matrix/Finite Element approach for both Linear and Nonlinear analysis. To develop modular structured program versatile and should be able to analyze a wide variety of problem.
5. Run the program for Non Linear analysis of Frame Tube building and verify its structural stability as a whole and to compare these results with commercial software called Mastan2 for accuracy.

Scope:

Focus of the work is on planar frames with the following characteristics.

1. The behavior of the structure is elastic.
2. Structure may behave like a truss or frame depending on the element node connection. In frame element for example all member are considered beam column elements.
3. The structures are considered to be an assembly of planar elements. Different frames may interact through common columns and through the floor diaphragms, which are considered to be rigid in their plane. If the structure contains shear wall or cores it has to be idealized as a single isolated column.
4. Static gravity loads and horizontal wind loads are applied as nodal forces.
5. During second order analysis geometric non-linearity due to large member deformation is the main focus and material non-linearity due to change in material property is not considered.
6. Connections are considered either fixed or hinged and it does not account for partial fixity.

The program is based on Matrix analysis of structure. Each structural member is idealized, i.e. its depth, width and elemental length are conventionally reduced to line elements. The material properties are represented by area (A), young's modulus of elasticity (E), torsion constant (J) and Poisson's ratio (ν). For second order analysis simple step method has been employed

Previous work:

Refs (10 & 12) have introduced the topic of high-rise structure building by introducing the different structural forms, with the main intention of explaining the different approximate methods for analysis.

A number of papers have been published for the analysis and design of tall buildings. In case of tall buildings stability being the main criterion, selection of structural form is important. Frame tube buildings have been considered to be efficient structural form for building ranging from 40 stories to 100 stories height. Refs (1-9) have attempted to analyze the frame tube building as accurately as possible with the limited computing capability that computers of those times had possessed. Importantly, each paper helps to understand the logic behind the analysis of the frame tube building. Through time, computers have evolved so has the complexity of the problem, various advanced techniques like the finite element method have evolved in tantamount. In Ref (1), a simplified approach to the analysis of frame tube buildings is made. The dominant modes of behaviors of the frame tube buildings resting lateral loads are recognized as

1. Rigidly – jointed frame action of the shear-resisting panels parallel to the direction of the loads and

2. The axial deformation of the frame panels normal to the direction of the load.

After which, they have proved that it is possible to simulate the behavior of the structure using a reduced equivalent plane frame. The transfer of vertical shear has been achieved by giving large values to appropriate shear transfer elements in the stiffness matrix. Once the modification to stiffness matrix is made then the process of analysis is as usual. Thus consequently reducing the amount of computation required in a conventional three-dimensional analysis to a simple 2-Dimensional planar frame analysis.

In Ref (2) author has taken consideration of the shear lag effects and has tried to predict the possible response of the building so as to include this phenomenon. The author accounts for the shear lag effect by assuming a parabolic stress distribution. He further goes on to simplify the problem by assuming that the spacing of columns and beams are uniform and they have all, uniform cross section area throughout the height. The framework panel of the column and beam are replaced by equivalent orthotropic plate, to form a closed tube structure. The orthotropic plate is chosen that the both the modulus of elasticity of the plate in both the horizontal and vertical direction be represent the axial stiffness of the beam and the column respectively. Shear modulus representing the shear stiffness of the framework. The structure then is analyzed as a plate. The method of plate analysis is fairly accurate and provides design curves for standard load cases for easy and quick reference; the assumption of a parabolic stress distribution to simulate the shear lag effect is surely one of the limitations. In next part ref (3), which is the 2nd of the 3 paper series published by the authors, problem of torsion is tackled by replacing the discrete structure by equivalent orthotropic tube and then a analysis it as a plate. Here, it is assumed that the shear rigidity of the structure to be low so as to simulate the shear racking

effect, which the ordinary elastic modulus would not. In Ref (4) third and the final part authors tackle the deflection in frame tube buildings.

Papers by Fazlur Khan and Amin papers regarding approximate analysis of framed tube buildings provide graphs for a quick and rapid analysis and design of frame tube building. The assumption made for the hand method is its limitations.

In ref (8) author has devolved a computer program and has provided a number of table to assist the analysis of frame tube buildings. He assumes the building to be square in plan and all columns other than the corner columns to be of same cross section. The corner column is assumed to have a twice the cross section area. The columns are connected by equal cross section spandrel beam. For the analysis he converts the structure into a planar model and for the corner column interaction he introduces a unit force at each floor level along lines column lines(for each frame he takes half of the corner column cross section area), one pair at a time and he establishes the actual interaction forces based on the compatibility condition. In his report he has presented tables for 40, 50, 60 story building. His program does not do the stability check.

In Ref (9) authors has submitted his work as a partial fulfillment of his PhD .In his dissertation author has presented a Macro element method to analyze the frame tube building. In the process he has also developed two programs, one for static and dynamic analysis and the other software for the optimization of frame tube building. Ref (18) has used his software for his MS dissertation on the analysis of frame tube buildings.

Ref (11) together with introducing the topic of tall buildings has also dealt with important structural forms in detail- the technique of structural analysis and design. He has also shown the various reduction techniques that can be used for transforming the structure so as to quickly verify the solutions. Reference (12)

covers the topic on the Matrix analysis, importance is laid for the solution of planar framed structure. He also has developed software using the matrix analysis principle for solving structures, which shall be used as reference software and will be compared with the software being developed in this thesis work.

Chapter 2

CONCEPTUAL FRAMEWORK

Matrix Methods:

Behaviors of all types of structures are described by differential equations. In practice, it is common to represent frames as planar structures for which the approximate or exact solution of the individual members are available. By using these solutions, relationship between force and displacement in the end of the members in combination with equilibrium and compatibility equations at the joints and supports can be written which yields a system of algebraic equations that describe the behavior of the structure. Thus these simultaneous equations can be solved using matrix methods. Once the equations are written in Matrix form taking advantages of the power computing capabilities of the computers the large system of equations can be solved quickly. In matrix method there are generally two approaches the flexibility and the stiffness approach and the former approach is used in this thesis work

Stiffness Approach:

In matrix method there are generally two approaches the flexibility and the stiffness approach and the later approach is used in this thesis work. Every element in the structure is idealized as a line element with section properties (A) area, (I) moment of Inertia and (J) torsion constant and material properties as (E) modulus of elasticity, (G) shear rigidity, (NU) poissons ratio. The whole planar

structure is considered as mesh, formed by individual finite elements connected together at joints called nodal points. Each element consists of 2 nodes and each node is associated with corresponding degree of freedom or nodal degree of freedom. The term Degree of freedom (DOF) is defined as “ the number of independent coordinate necessary to define the configuration of the system” In space Nodal DOF are 12 in number but in the case of framed structure with rigid joints 3 DOF are considered. The frame of reference to identify entire structure is called global coordinate system referred to as (XY axis). The orthogonal axis with X-axis along the member longitudinal axis refers to local coordinate axis. Forces are applied to the nodes as point load and the corresponding displacement is measured at nodal points. Thus relationship between nodal forces and corresponding nodal displacement is written in terms of stiffness equation or flexibility equations.

The procedure involved in the formation of member stiffness matrix is quite simple can be summarized in 3 steps

1. Equilibrium.
2. Compatibility
3. Hooke's law.

For illustration consider a 1D spring element which has 2 DOF, the element force–displacement relationship relates 2 nodal forces (F_i, F_j) to 2 nodal displacements through 2X2 stiffness matrix.

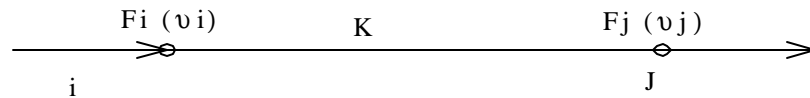


fig 1

$$\{f_e\} = [k]\{u_e\}$$

Expanding, the above equation, we get

$$F_i = k_{11}u_i + k_{12}u_j$$

$$F_j = k_{21}u_i + k_{22}u_j$$

In matrix form, we get

$$\begin{bmatrix} F_i \\ F_j \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{bmatrix} U_i \\ U_j \end{bmatrix}$$

Thus $k_{11}, k_{12}, k_{21}, k_{22}$ are to be defined

Apply springs law, force (F) Vs displacement (Δ) is a linear curve, it can be written as,

$$\{f\} = [k]\{\Delta\}$$

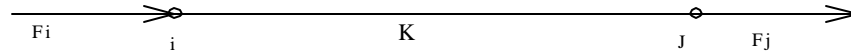


Fig.2

Nodal forces.

We need to find force (F) in terms of end displacements u_j and u_i . Using the compatibility conditions,

$$\Delta = u_j - u_i$$

$$f_i = k (u_j - u_i)$$

Now applying the equilibrium equations

$$\Sigma f_x = 0;$$

$$f_i + f_j = 0$$

$$\text{Thus, } f_i = -f_j = -(k u_i + k u_j)$$

In matrix form can be written as,

$$\begin{bmatrix} f_i \\ f_j \end{bmatrix} = \begin{bmatrix} K & -K \\ -K & K \end{bmatrix} \begin{bmatrix} u_i \\ u_j \end{bmatrix}$$

Thus this is the member stiffness matrix, the fundamental stiffness equations is of the form

$$\{\mathbf{F}\} = [\mathbf{k}]\{\mathbf{D}\}$$

Where, [K] is the element stiffness matrix. In case of an axially loaded member, if unit displacement is applied keeping all the other degrees of freedom fixed then the force applied will be equal to the stiffness of the member or in other words, nodal forces and nodal displacements are connected with each other by element stiffness matrix. This is formulated depending upon the material and sectional properties of the element. For information about the formulation of the complete stiffness matrix please refer ref (13). In our program focus is on beam-column and truss. So the element stiffness matrix will have 3 Dof at each node, thus for a beam element stiffness matrix will be of the dimension [6X6].

$$\begin{bmatrix} F_{X1} \\ F_{Y1} \\ M_{Z1} \\ F_{X2} \\ F_{Y2} \\ M_{Z2} \end{bmatrix} = E \begin{bmatrix} \frac{A}{L} & 0 & 0 & -\frac{A}{L} & 0 & 0 \\ 0 & \frac{12Iz}{L^3} & \frac{6Iz}{L^2} & 0 & -\frac{12Iz}{L^3} & \frac{6Iz}{L^2} \\ 0 & \frac{6Iz}{L^2} & \frac{4Iz}{L} & 0 & -\frac{6Iz}{L^2} & \frac{2Iz}{L} \\ -\frac{A}{L} & 0 & 0 & \frac{A}{L} & 0 & 0 \\ 0 & -\frac{12Iz}{L^3} & -\frac{6Iz}{L^2} & 0 & \frac{12Iz}{L^3} & -\frac{6Iz}{L^2} \\ 0 & \frac{6Iz}{L^2} & \frac{2Iz}{L} & 0 & -\frac{6Iz}{L^2} & \frac{4Iz}{L} \end{bmatrix} \begin{bmatrix} u_1 \\ v_1 \\ \mathbf{q}^1 \\ u_2 \\ v_2 \\ \mathbf{q}^2 \end{bmatrix}$$

If beam or truss is inclined at any angle θ w.r.t to horizontal the element stiffness matrix, which is in the local coordinate system, has to be multiplied with a

transformation matrix to convert it into global coordinates. Thus the transformation matrix is

$$A = \begin{bmatrix} \cos \mathbf{q} & \sin \mathbf{q} & 0 & 0 & 0 & 0 \\ -\sin \mathbf{q} & \cos \mathbf{q} & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos \mathbf{q} & \sin \mathbf{q} & 0 \\ 0 & 0 & 0 & -\sin \mathbf{q} & \cos \mathbf{q} & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$$

Where θ is the angle made between the element and the horizontal. The matrix operation involved is

$$[K]_{\text{global}} = [A] [k]_{\text{local}} [A]^T$$

Once the member stiffness matrix is formulated in Global co-ordinate system, then they are assembled to form the combined global stiffness matrix $[K]$.

Nonlinear analysis:

Most of the civil engineering structures fall in the category of low-rise structure and the deflection are very small; there by do not exhibit significant nonlinearity. . In structures like suspension bridges and very tall buildings that are slender, before these structures reach limit of resistance they exhibit significant nonlinear behavior. Thus even though most engineers are fluent in first order analysis it is important to conduct an Nonlinear analysis which will help analytically simulate

more appropriate and better behavior of the structure than the first order analysis. This will ultimately help in the optimal design of the structures.

In linear elastic structures, as seen before, it was assumed that the material is unyielding and its properties invariable and the equations of equilibrium are formulated on the geometry of the unloaded structures or, on the initial reference of the configuration. In Nonlinear analysis the structure may lose its geometry due to large deflections and the material can also exhibit nonlinearity. Nonlinear analysis offers many options for solving the problems resulting from the above cases. Geometric nonlinearity alone can be considered making an assumption that the material is elastic including the effect of finite displacement and deformation during the formulation of equilibrium equation. It is also possible that material nonlinearity alone can be considered taking into account that the member property changes under loads. Finally material nonlinearity and geometric nonlinearity can be considered together in the analysis. Thus it is the responsibility of the engineer to use his judgment and discretion while predicting the most dominant behavior of the structure he wishes to simulate analytically. According to W.Mcguire, R.H.Gallagher and R.D.Ziemian in their textbook sources of nonlinearity are as follows.

Geometric effects:

1. Initial imperfections such as member camber and out of plumb erection of a frame.
2. The $P-\Delta$ effect, destabilizing moment equal to gravity load times the horizontal displacement it undergoes as a result of lateral displacement.
3. The $P-\delta$ effect, the influence of the axial on the flexural stiffness of the individual member.

Material effect :

1. plastic deformation of the steel structures.
2. Cracking or creep of reinforced concrete structure.
3. Inelastic interaction of axial force, bending, shear and torsion.

Combined effects:

1. Plastic deformation plus $P-\Delta$ effect and/ or $P-\delta$ effects.
2. Connection deformation
3. Panel deformation
4. Contribution of secondary system to strength and stiffness.

Levels of analysis

The fig below shows load-deflection behavior of plane frame building analyzed using various refined and simplified models. The broken line is obtained by F^1 order analysis ignoring the effects of both change in geometry and the yielding of the material. The broken curved line is that of second order elastic analysis that includes only the effect of change in geometry and instability of the structure and this is the focus of thesis work. The curve approaches asymptotically to the elastic critical limit generated by an Eigen value analysis. The piecewise linear curve without a descending branch represents a first order elastic-plastic hinge analysis when the effect of change in geometry has been ignored. Its value is plastic limit load, which can be obtained by plastic analysis. The piecewise linear curve with a descending branch shows the result of a second-order elastic-plastic hinge analysis allowing for simple plastic analysis for

the change in the geometry associated with the sway deflection. The plastification, initial imperfections, residual stresses and strain hardening are all accounted for, the smooth continuous curve is obtained by doing an second-order spread plasticity analysis. The peak load is the true strength of the frame or the true load carrying capacity of the structure.

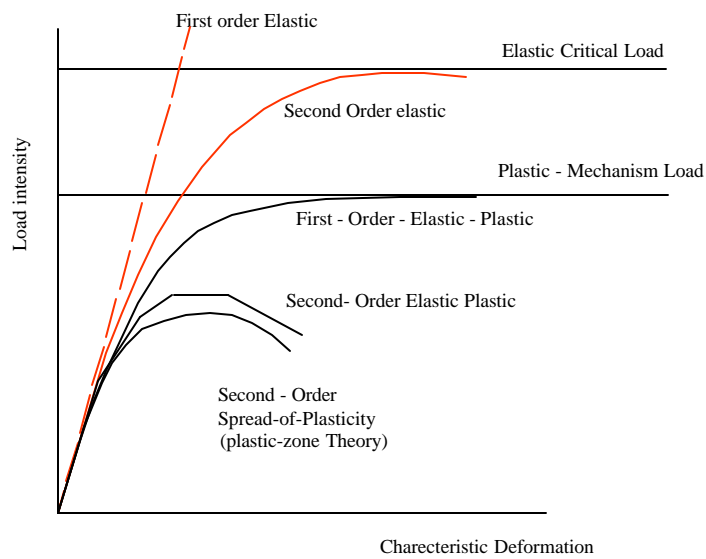


Fig 3

Load Deflection curve for a plane frame

In our thesis work the focus is laid on second order elastic analysis, taking into consideration only the geometric nonlinearity ignoring the material nonlinearity. The critical load is calculated by conducting a simple eigen value analysis. The whole procedure has been converted in to a program which can be used to solve problem.

Nonlinear analysis:

In IInd order analysis nonlinear equilibrium equations are reduced to a set of simultaneous equations, which is written in matrix form. Once simultaneous equations are written in matrix form, different methods can be devised to solve there by displacement and deformations of the members can be traced incrementally. Thus each method is a variation of global stiffness matrix and symbolically written as

$$[K_t] \{d\Delta\} = \{dP\}$$

$[K_t]$ = Tangent stiffness matrix.

$\{d\Delta\}$ = incremental displacement.

$\{dP\}$ = incremental loading.

The various levels differ in type of nonlinearity included in the types of nonlinearity included $[K_t]$, the way the equations are formulated and the details of the equation solution, and the way members are subdivided.

Focusing on elastic IInd order analysis, Finite displacement and deformations are accounted for in the equations of equilibrium and can be written as

$$[K_e + K_g] \{d\Delta\} = \{dP\}$$

$[k_g]$ is the geometric stiffness matrix. Many different ways are there to develop the $[K_g]$ matrix, dividing the members into sub elements yields very accurate results. Many techniques for solving the equations are given later on in this report. For the calculation of the elastic critical loads the global stiffness equations is cast in the form of a generalized eigen value problem in which the equations at the critical state is

$$\left[k e + I \hat{k}_g \right] \{d\Delta\} = \{dp\}$$

Where K^g is called the geometric stiffness matrix computed for a reference load $\{P_{ref}\}$, $\{\lambda\}$ (the eigen value) is the load w.r.t $\{P_{ref}\}$, and $\{\Delta\}$ the eigen vector is the buckled shape. The lowest value λ that satisfies the equation of $\{\Delta\}$ not equal to 0 yields the Elastic buckling load.

Formulation of Kg:

As any standard text book Matrix analysis of structure would indicate the technique involved in the derivation of Kg matrix, but we shall derive the Kg matrix basic axial force member.

Let's consider the segment of length dx as shown in the Fig4. Their initial configuration is designated as ab. But after rigid body rotation and axial straining its length is

$$a'b' = \left(1 + 2 \frac{du}{dx} + \left(\frac{du}{dx} \right)^2 + \left(\frac{dv}{dx} \right)^2 \right)^{\frac{1}{2}} dx$$

$$\text{Let's designate, } d_{ab} = \left(2 \frac{du}{dx} + \frac{du}{dx} + \frac{dv}{dx} \right)$$

$$\frac{a'b'}{dx} = (1 + d_{ab})^{\frac{1}{2}}$$

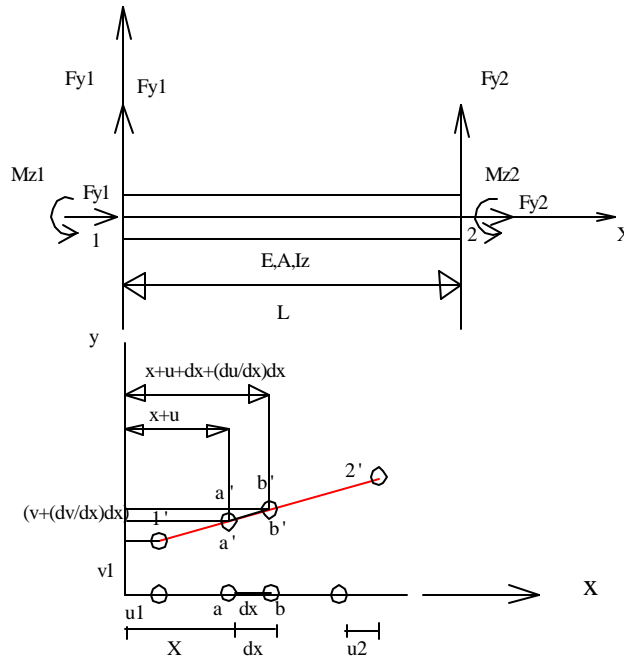


Fig 4

Finite strains of planar element

or expanding $a'b'$ by binomial theorem and defining the extension per unit length,

$(a'b' - ab)/dx$, finite strain, e_{fin}

$$e_{fin} = \frac{du}{dx} + \frac{1}{2} \left[\left(\frac{du}{dx} \right)^2 + \left(\frac{dv}{dx} \right)^2 \right]$$

Applying the principles of virtual displacement to the reference configuration

$$dW_{int} = \int_{vol} \mathbf{s}_x d e_{fin} d(vol)$$

Integrating over the depth of the member, we get

$$dW_{\text{int}} = \left(\int_0^L \mathbf{s}_x A \left(\frac{d\mathbf{d}u}{dx} \right) dx + \frac{1}{2} \int_0^L \mathbf{s}_x A \left[\mathbf{d} \left(\frac{du}{dx} \right)^2 + \mathbf{d} \left(\frac{dv}{dx} \right)^2 \right] dx \right)$$

let $\left(\frac{d\mathbf{d}u}{dx} \right) = \delta \left(\frac{du}{dx} \right)$ which is valid for infinite small displacement. Now using the conventional elastic stress- strain relationship in the first integral and letting $\sigma_x A = F_{x2}$ we get

$$dW_{\text{int}} = \left(\int_0^L \left(\frac{du}{dx} \right) EA \left(\frac{d\mathbf{d}u}{dx} \right) dx + \frac{1}{2} F_{x2} \int_0^L \left[\mathbf{d} \left(\frac{du}{dx} \right)^2 + \mathbf{d} \left(\frac{dv}{dx} \right)^2 \right] dx \right)$$

The first term in the equation above is for elastic stiffness matrix of the axial force element and the second integral term is of great concern as it produces geometric stiffness matrix $[k_g]$. Treating δ as differential operator w.r.t variables du/dx and dv/dx , the internal virtual work can be written as

$$dW_{\text{int}} = \left(F_{x2} \int_0^L \left[\left(\frac{d\mathbf{d}u}{dx} \frac{du}{dx} \right) + \left(\frac{d\mathbf{d}v}{dx} \frac{dv}{dx} \right) \right] dx \right)$$

Using the shape function the above equation can be written as

$$[K_g] = F_{x2} \int_0^L \left[\{N'_u\} \{N'_u\} + \{N'_v\} \{N'_v\} \right] dx$$

Where, $\{N'_u\}$ and $\{N'_v\}$ are the shape functions, in this case for axially loaded members. Thus for axially loaded members displacement equation in the form of shape function can be written as

$$u = (1-\xi)u_1 + \xi u_2 \text{ and } v = (1-\xi)v_1 + \xi v_2$$

Therefore,

$$[N'_u] = \begin{bmatrix} -1 & 1 \\ L & L \end{bmatrix} \text{ and } [N'_v] = \begin{bmatrix} -1 & 1 \\ L & L \end{bmatrix}$$

Combining the equation suitably we get

$$[K_g] = \frac{F_{x2}}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 1 & 0 & -1 \\ -1 & 0 & 1 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

Combining the bending and the axial forces the geometric stiffness matrix will be,

$$[K_g] = \frac{F_{x2}}{L} \begin{bmatrix} 1 & 0 & -1 & 0 & 0 & 0 \\ 0 & \frac{6}{5} & \frac{L}{10} & 0 & \frac{-6}{5} & \frac{L}{10} \\ -1 & \frac{L}{10} & \frac{2L^2}{15} & 0 & \frac{-L}{10} & \frac{-L^2}{30} \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & \frac{-6}{5} & \frac{-L}{10} & 0 & \frac{6}{5} & \frac{-L}{10} \\ 0 & \frac{L}{10} & \frac{-L^2}{30} & 0 & \frac{-L}{10} & \frac{2L^2}{15} \end{bmatrix}$$

Different approaches to nonlinear analysis:

Basic principle involved in nonlinear analysis is to perform a set of matrix-structural operations to update the stiffness matrix constantly so that all nonlinear effects of the structure are accounted. As equilibrium configuration of the structure changes constantly, series of load increments are made to carry out the analysis. The equilibrium and kinematics states of the structure from previous cycle are used to formulate the stiffness relation for the solution of the next load

increment cycle. Thus, the solution of nonlinear analysis problem is got by a series of linearized analysis. The size of the load increment has a profound influence on the solution time and the convergence characteristics of the problem. Smaller load steps yield accurate solution and in turn will take larger number of load steps to cover the loads range. On the other hand if the increment is too large, the number of iteration required to obtain a solution for a given load step may be large and in some case the solution may even diverge. If a non-iterative scheme is followed the use of a large load increment may result in a large drift off from the actual equilibrium path.

Thus nonlinear analysis is iterative or can be non iterative There are many techniques and four major solution schemes for nonlinear problems are studied, they are

1. Load control method.
2. Displacement control method
3. Arc control method
4. Work control method.

In applying these methods to trace the Non-linear load-deflection curve of the structure, it is assumed that the equilibrium path to be followed is continuous and unique.

It is assumed that the equilibrium and kinematics states of the structure at the (I-1)th load step is know and it is now required to find the state of the structure or the stiffness relation at ith interval. Relation can be written as

$$K_i^{j-1} \Delta D_i^j = \Delta R_i^j + Q_i^j$$

Where K_i^{j-1} = stiffness matrix for the load step I based on the equilibrium and kinematics states at the end of the (j-1)-th load step are known.

ΔD_i^j = incremental displacement vector at the jth iteration load step i.

ΔR_i^j = load increment at the j-th iteration of load step I

Q_i^j = force vector at the j-th Iteration of the load step i. (difference between the external and internal forces)

Suppose the solution converges after n iterations; the load at the end of the i-th load step is given by

$$R_i = R_{i-1} + \sum \Delta R_i^j$$

And the displacement is given by

$$D_i = D_{i-1} + \sum \Delta D_i^j$$

The updated stiffness matrix can then be built and the solution can move to next increment. The solution schemes differ by size of the ΔR is determined and the way iteration is carried out inside a load step.

Load control method:

Here the size of the incremental load at load step I, ΔR_i , is obtained as a fraction of the total applied load

$$\Delta R_i = \lambda_i R_i$$

In the above equation, λ_i is the load increment factor and R is the total applied load. The user determines the value of the λ_i by using personal judgment and discretion. The load control method can be iterative or non-iterative. Non-iterative method is referred to as simple step method and Iterative method referred to as Load control Newton-Raphson method. In this thesis work to calculate the nonlinear response of the frame tube building simple incremental load method has been employed.

Simple incremental method:

In this method the unbalanced forces that exist in each load step is ignored. That means no iteration is done. Thus stiffness relation that exists at load step I, reduces to

$$K_{i-1} \Delta D_i = \Delta R_i$$

Here only one iteration not done, so the value of $j=1$. Can be rewritten as

$$K_{i-1} \Delta D_i = \lambda_i R$$

From which ΔD_i can be calculated. ΔD_i calculated is added to the cumulative sum ΔD_{i-1} from the previous cycles to get the new displacement D_i

$$D_i = \Delta D_{i-1} + \Delta D_i$$

Thus updated stiffness matrix can be calculated.

Advantage of using simple incremental method is its simplicity. Its unloading branch of the load-deflection can be investigated by giving negative load increments. The main disadvantage of this method is that the error tends to accumulate and may prove to be disastrous if the structure is highly nonlinear in nature. This is because iterations are not employed within a load step to bring the calculated equilibrium path back to true equilibrium path. This is called drift off error.

Load control Newton Raphsons method:

In this method iteration employed within the load cycle eliminate the unbalanced force. The presence of unbalanced force indicate that the internal and the external forces are not in equilibrium, and this errors occurs because of linearization process in which current stiffness matrix is calculated based on the passed configuration the structure. If this error is not corrected then the calculated equilibrium path will drift off from the true equilibrium path. Lets take i-th load increment for the first iteration (j=1)

$$K_i^0 \Delta D_i^1 = \Delta R_i^1$$

$$K_i^0 \Delta D_i^1 = \lambda_i^1 R$$

Where ΔD_i^1 can be calculated from the above equation and added to the cumulated displacement of the structure. Now the stiffness matrix can be formed. Corrections are applied to the displacement by subjecting the structure to the unbalanced forces. Unbalanced forces are calculated as the difference between internal and external applied load. Therefore subsequent iteration, equation is

$$K_i^{j-1} \Delta D_i^j = Q_i^j$$

The iteration stops when ΔD_i^j or Q_i^j is negligible. The advantage of this method is that the drift off error is greatly reduced or controlled. The limitation of this method is that it fails at limit points. This method fails also if the system exhibits snap through behavior.

Chapter 3

STRUCTURAL ANALYSIS OF TALL BUILDINGS

Introduction:

Is there an absolute solution to a problem? Hypothetically may be “yes”, in reality we dare not to claim. Can this be our limitation? Or is it that the solution is already in absolute state and our mind fails to perceive it. Have we to wait for solution to evolve into absolute forms or rhetorical manner wait for the mind to transcend into absolute form to get the solution. Whatever the answer may be, in real world there is always a possibility for problems to arise even from well accepted fact. When problem becomes conspicuous, complete understanding is demanded for it. As problems get solved, they keep evolving, may be this is our greatest problem. Question your self, are we really solving anything? Well, this is our inspiration for the quest of perfection.

Rome saw its skyline pushed up by the construction of multistory buildings up to 4 stories high using wood, but eventually the great fire of Nero made conspicuous its problems. As a solution, arches and barrel domes were constructed of concrete and brick. Industrial revolution put lot of pressure on construction industry to meet its demands, more and more different problems became conspicuous and solution were demanded immediately. As a solution, tall building were built to ease the space jams in urban areas, bridges and dams for smooth communication and urban development and many more. All this could be realized only by improved materials and structural innovations made during the post industrial revolution. The tragic event of 9/11 has made conspicuous

another short come, which needs to be dealt with immediately. Solution for this problem is inevitable, it just a matter of time until research will be done on every aspect of tall buildings and solution will evolve.

There are a number of structural forms that are available for selection, selecting an appropriate and optimal form is responsibility of a structural engineer. A range of factors influence the selection of structural form they are internal planning, material and the type of construction, the external architectural planning, the planned location and the routing of service systems, the nature and the magnitude of horizontal loading and the height and proportioning of the structure. Taller the building slender it will be and becomes more dependant on the structural forms. A major factor influencing the structural form is the function of the building. Office building desire column free interiors so structural form selected will be such that a lot of material is placed on the external perimeter of the plan and internally placed around the elevators, stairs and service shaft. For buildings taller than 10 stories, they need extra material to account for resisting the wind loads.

In addition to the above factors, buildings structural forms are arranged such that they support the gravity, dead and live loads and also to resist all levels of external horizontal load shear. These necessities should be accomplished as economically as possible. In the case of horizontal loading, tall buildings behave like a cantilever. They may comprise of individual members acting in the form of a cantilever, such as shear wall or cores, each bending about its own axis acting unison through the horizontal in plane rigidity of the floor slab. It may also comprise of a number of vertical columns and walls all acting compositely which is made possible by using proper shear resistant connectors, there by each member act as chord of a massive cantilever. Moreover, within the selected structural form advantage can be taken by placing main vertical members in plan

so then the dead load of the structure can counter balance some all or a portion of the tensile stresses developed in members due to horizontal load. This avoids net tension occurring in vertical members and prevents this uplifting force to rip of the foundation.

Tall buildings are usually constructed of concrete, steel or composite. Steel framing has played a prominent role in the development of tall buildings due to its high strength to weight ratio. It is possible to have longer spans, partial prefabrication leading to reduce on site work and more rapid erection, major disadvantage being corrosion problem, fire resistance, expensive to clad, and requiring expensive diagonal bracing or rigid frame connection. Concrete on the other hand with shear walls resisting the horizontal loads and improved concrete strength and innovation in the structural forms have allowed the height of the structures to touch the 100-story mark. The different structural forms are braced-frame structures, rigid-frame structures, in filled-frame structures, flat-plate and flat-slab structures, shear wall structures, coupled wall frame structures, wall-frame structures, framed-tube structure, tube-in-tube structures, bundled-tube structures, braced-tube structures, outrigger-braced structures, suspended structures, space structure and hybrid structure.

Mathematical Modeling of a tall building:

A building's response to loading is governed by the components that are stressed as the building deflects. Ideally for the ease of accuracy of structural analysis, the participating components would include only the main structural elements, the slabs, beams, columns girders, walls and cores. In reality, however, other non-structural elements are stressed and contribute to the building's behavior; these include, for example, the staircase, partitions and cladding. To simplify the problem it is usual in modeling a building for analysis to include only the main

structural members and to assume that the effects of the nonstructural component are small and conservative.

To identify the main structural elements, it is necessary to recognize the dominant modes of action of the proposed building structure and to assess the extent of the various member contributions to them. Then by neglecting consideration of the non-structural components, and less essential components, the problem of analyzing a tall building structure gets reduced to viable sizes. In case of very tall and complex building it is necessary to reduce even further the size of the analysis problem by representing some of the structural elements with simpler analogous components. Thus purpose of structural idealization is to represent all the important features of the buildings for the purpose of deriving mathematical equation's governing the behavior of the system. The mathematical model should include enough details to be able to describe the system in terms of the equations without making it too complex. The mathematical model may be linear or nonlinear, depending upon the behavior of the component system. Linear model permit quick solution and are simple to handle, however nonlinear models sometimes reveal certain characteristics of the systems that cannot be predicted using linear model.

There by the mathematical model is gradually improved to obtain more accurate results. Sometimes for approximate initial analysis, approaches crude or elementary model are used to get a quick insight into the overall behavior of the system. The following paragraphs to follow shall illustrate the modeling and refining procedure.

Approaches to analysis:

The modeling of the tall building structure for analysis is dependent to some extent on the approach to analysis which is in turn related to the type and the size of the structure and the stage of the design or which the analysis is made. The usual approach is to conduct approximate rapid analysis in preliminary design, and a more detailed and accurate analysis for the final design stage. A hybrid approach is also possible in which a simplified model of the total structure is analyzed first after which the results are used to allow part by part detailed analysis of the structure.

Preliminary Analysis:

The purpose of the preliminary analysis, that is analysis for early stage of the design, may be to compare the performance of the alternative proposals for the structure, or to determine the deflection and major member forces in a chosen structure so as to allow it to be properly proportioned. The formation of the model and the procedure for a preliminary analysis should be rapid and produce results that are dependable approximates. The model and its analysis should therefore represent fairly well, if not absolutely accurately, the principle modes of action and interaction of major structural elements. It is important that the solutions after preliminary analysis should yield deflection close to the exact model and main member forces that are dependably within about 15% of the value from an accurate analysis.

Intermediate and Final analysis:

The requirement of intermediate and the final analysis is that they should give, as accurately as possible, results for deflection and member forces. The model

should, therefore, be as analysis program and computer capacity will allow for its analysis. All the major modes of action and interaction and as many as possible lesser modes, should be incorporated except where a structure is symmetrical in plan and loading, the effect of the structure twisting should be included. If the structure and loading are symmetrical, a three dimensional model would be acceptable, or if repetitive regions up to the height of the structure can be simplified by a lumping technique, this also would be acceptable.

In contrast to the reductions above, however certain final analysis may require separate, more detailed analysis of particular parts, using forces or applied displacement from the main analysis for example, in deep beams at transition level of the structure around irregularities or holes in the shear wall. With easy access to fast computer FEM model of the structure must be done as a part of a final analysis.

Reduction technique:

When detailed modeling consumes lot of memory space and time, reduction modeling can be used for simplicity and the ease of analysis. There are many ways that a model can be reduced to simpler models, some of them are

1. Symmetry and Anti-symmetry.
2. Two-dimensional modeling of non-twisting model.
3. Lumping
 - a. Vertical lumping.
 - b. Horizontal lumping

The details for the above techniques can be found in ref (text book by Alex Coull).A brief insight into lumped girder frame technique is presented below,

which is used as rapid analysis of frame tube building later on. In this method the repetitive floor system are identified and is exploited to form a model with fewer stories, which in turn benefits a rapid analysis. This method allows an accurate estimate of the drift and a good estimate of the member forces. The girders are usually lumped in sets of three girders or five girders into single girders thereby reducing the number of stories considerably. For very accurate analysis it is preferable not to include the first floor and roof girders during the lumping process because the simulation of the frame behavior near the top and the base of the building differ from the middle portion. In the figure below the first floor and the top floor are untouched in order to maintain the boundary condition same while the remaining floors are lumped up. In the building show below story height is h and the column inertia is I_c and that of the girder is I_g .

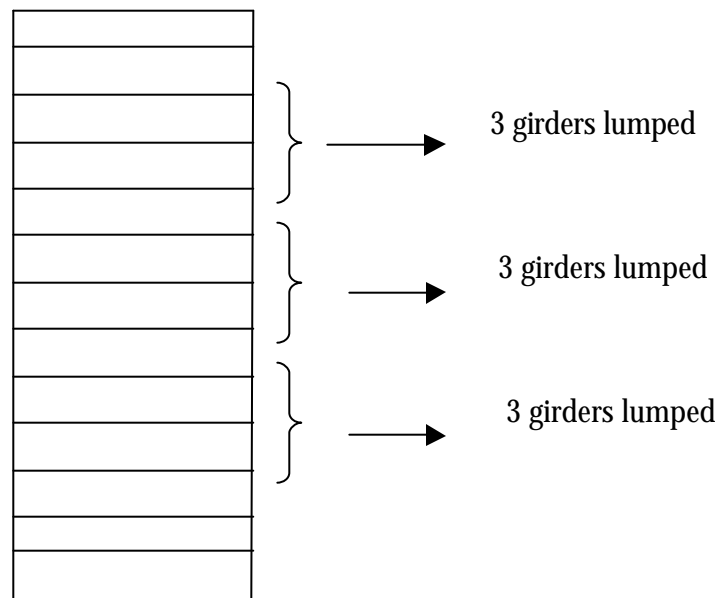
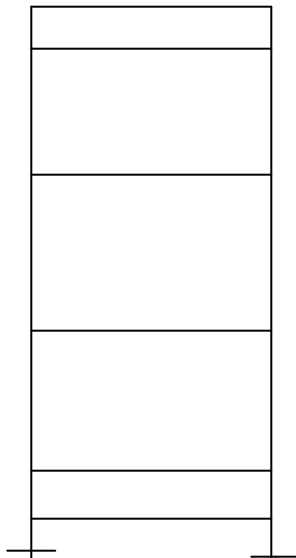


Fig 5
Prototype rigid frame

I_{ge} = Equivalent moment of inertia of the girder

I_{ce} = Equivalent moment of inertia of the column.



$$I_{ge} = (I_{g1} + I_{g2} + I_{g3})$$

Fig6
Equivalent lumped girder frame

The requirement of a substitute frame is that, for horizontal loading, joint translations should be the same as those of the original structure. For translations caused by girder flexure, equation below shows the requirements to be satisfied

by assigning the inertia of the equivalent girder to be equal to the sum of the lumped n-girder inertias of the original frame, that is,

$$I_{ge} = \sum^n I_g$$

To determine the properties of the columns in lumped girder method, the component drift caused by the double curvature column bending story height nh in the equivalent frame to the corresponding drift over n stories in the original frame to the corresponding drift over n stories in the original frame are equated to get the equivalent column inertias. Or in other words the flexural strengths of the equivalent frame to original frame are equated to displacements to get the equivalent column inertias.

$$\frac{Q(nh)^2}{12E \sum \frac{I_{ce}}{nh}} = \frac{Qh^2}{12E} \sum^n \frac{1}{\sum \left(\frac{I_c}{h} \right)_i}$$

$$I_{ce} = \frac{n^3}{\sum^n \left(\frac{1}{I_c} \right)_i}$$

The column sectional areas, which control the cantilever deflection, must have the same second moment about their common centroid in the lumped and the original structure. Consequently, the areas of the equivalent and the original structure remain the same as the original structure. The horizontal loading on the equivalent structure frame is applied as equivalent concentrated loads at the lumped girder levels, taking half new- story-height regions above and below the lumped girder as tributary areas.

When the lumped girder frame has been analyzed the , the results are transformed back to the original frame The moment in the original girder at the same level will be $1/n^{\text{th}}$ of the resulting moment and the moments in the girders between the lumped girder level shall be obtained by interpolating the moment.

Assumptions:

Following simplifying assumptions are necessary to reduce the problem to viable sizes. The assumptions were made keeping in mind the frame tube building, its anticipated mode of behavior, and the type of analysis.

The common assumptions are

- Participation components: Only the primary structural components participate in the overall behavior. The effect of secondary structural components and non-structural components are assumed to be negligible and conservative. Although the assumption is generally valid, heavy cladding may not be negligible and be significantly stiffening the structure. Similarly masonry infill may significantly change the behavior and increase the forces un-conservatively in a surrounding frame.
- Floor slab: Floor slab is assumed to be infinitely rigid in plane. This assumption causes horizontal plane displacement of all vertical elements at a floor level to be definable in terms of the horizontal plane rigid body rotation & translation of the floor slab. Thus the number of unknown displacements to be determined in the analysis is greatly reduced.
- Negligible stiffness: components of relatively small stiffness are assumed negligible. These include for example, the transverse bending stiffness of slab, the minor axis stiffness of shear wall, and the torsion stiffness of column beams, and walls.

- Negligible deformation: deformations, which are relatively small and of little significance are neglected like the axial deformation of the beams, the previously discussed in plane bending and shear of the floor slab.

Once the whole structure has being modeled for analysis, it has to be furthered modeled for individual member and then joined together back into the structure, satisfying all the compatibility the equilibrium equations.

Mathematical modeling begins with structural idealization members which have depth and width as well as length, are conveniently reduced to line elements. Their resistance to deformation is represented by material properties such as young's modulus (E) and Poisson ratio (ν), and by geometric properties of the cross section such as are (A), moment of inertia (I), and torsion constant (J). The behavior of the connection, that is, whether pinned, rigid, semi rigid, yielding, etc., must be stipulated. How these idealization decisions are made, or the necessary property determined, is extremely important and involves considerable judgment.

The different types of Idealization used in mathematical modeling for a frame structure is explained in the following paragraphs. For the purpose of mathematical modeling the state of stress in the member is represented by forces at the element ends. The corresponding displacements of these nodes - the degree of freedom – are employed in the characterization of the displaced state of the element. The complete frame work has 12 nodal degrees of freedom and 12 nodal force components as shown in figure.

If the size problem still persists then a reduction analysis can be used for the analysis of the structure. The technique is explained below. Our work concentrates on the modeling and the analysis of frame-tube building. All of the above assumptions are included for the analysis of frame-tube building.

Frame tube buildings:

Frame tube buildings consist of closely spaced column connected by deep spandrel beams placed on the perimeter in plan. Framed-Tube buildings are generally suited for story height from (40-100). The close spacing of the columns are viewed with mixed reaction with respect to aesthetics. Tube-form though original developed for rectangular plans, it is now used for different shapes and sometimes used for circular and triangular plans too.

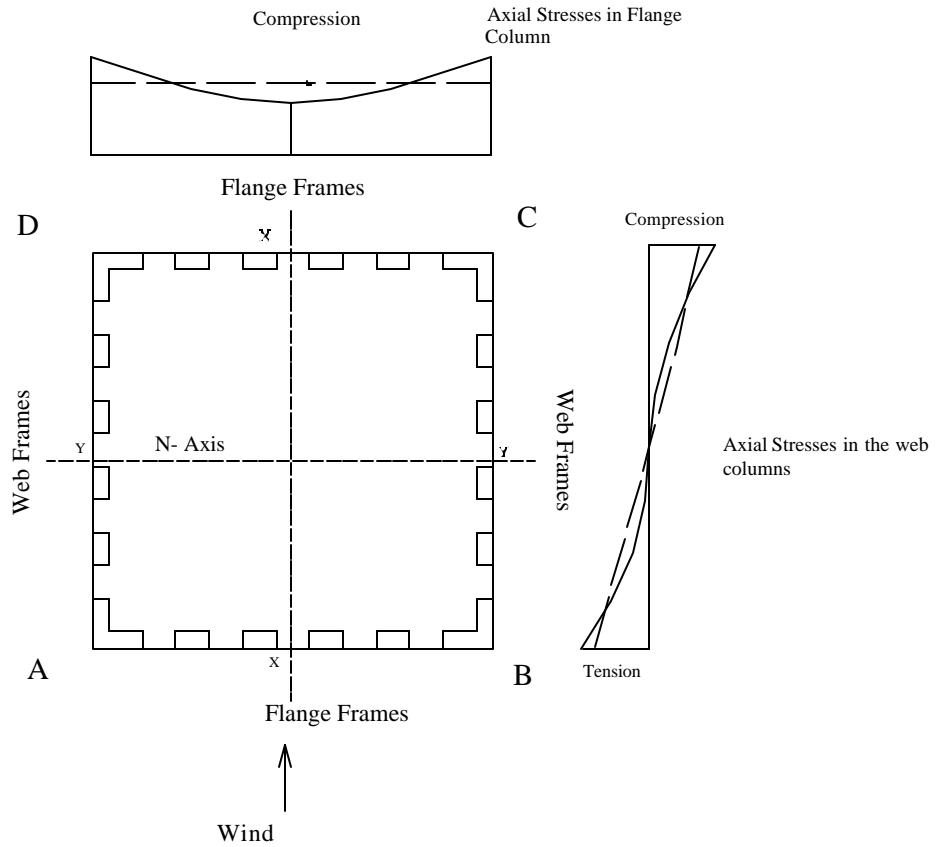
In frame tube building columns are arranged unlike the frame structure with the strong bending direction of the columns aligned along the face of the building. Main reason do so is to make sure large amount of material is placed on the periphery of the building to maximize the inertia of the building. In many buildings this is used to resist entire lateral load on the building to be resisted by the exterior columns while the gravity loads are shared between exterior columns and the interior structure in the form of columns and load inner core. The panel's normal to the direction of the wind is considered as flange and the panel perpendicular is considered as web of the perforated cantilever. In the lower floor it is desired to have columns spaced wide apart for which a deep transfer girder is provided which connects all the loads from the upper floors and transfers to the columns below.

The uniformity of the structure makes construction simpler and enables the industrialized production of the components. For steel buildings large elements can be prefabricated in the factory and easily erected in the work site. In case of concrete construction use of gang forms raised enables speedy construction.

Mode of behavior:

Although the structure has a tube-like structure, its behavior is more complex than a plain imperforated tube as the stiffness may be less. When subjected to lateral load the flange portion of the structure, on the windward is subject to tension force and the flange portion on the leeward side is subjected to compression, as indicated in the fig 7. In addition to this, the parallel frames to the wind are subjected to the usual in-plane bending and the shearing or racking action associated with an independent rigid frame. This action gets problematical by the fact that flexibility of the spandrel beams produces a shear lag effect, thereby increasing the compressive stresses in the corner column and decreasing the stresses in the interior columns of both the flange and the web portion of the structure. This can be seen in the fig 8.

When a frame tube building is subjected to lateral loading it is resisted by web-side panel, which deform such that the columns A and B are in tension and columns C and D are in compression. The principle interaction between the web and the flange would be through the vertical displacement of the corner columns. These displacements correspond to the vertical shear in the girder of the flange frame, which mobilizes

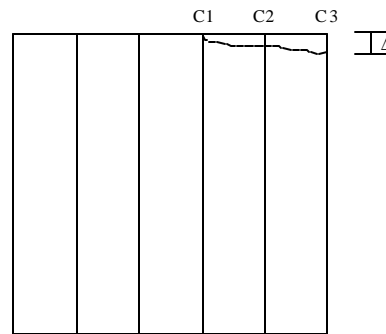


Axial stress distribution in columns of laterally loaded framed tube

Fig 7

the column forces in the flange column. For example if the column C is in compression it will try to compress the adjacent column since the two are connected by spandrel beam, the compressive deformation will not be identical since the spandrel beam will bend. Thus the axial deformation in adjacent columns will be less, by an amount equal to the stiffness of the

connecting beam. Hypothetically, beams of infinite stiffness will develop a pure tubular action. Thus the deformations in successive adjacent columns in frame will be lesser than the previous one. Since the external applied moment is to be resisted by internal couple produced by the compressive and the tension stress produced on either side of neutral axis of structure, it follows that the stresses in the corner column will be greater than those in the interior column.



Deformation of the flange frame causing shear

Fig 8

Structural analysis:

Ref (12) During analysis has assumed that the horizontal diaphragm slab has infinite in plane stiffness. This maintains the cross section shape of the structure at each level and the cross-section at these positions under go only rigid body

movements in plane. Horizontal displacement of the building can be expressed in terms of orthogonal translation and rotation. The out of plane stiffness of the floor slab is assumed low so that they do not resist twisting and bending. The floor slab is assumed unable to provide the coupling between opposite normal to wind which act as flange of the side frame. Both the side and normal frames are therefore subjected largely to in plane action, and the out of plane actions are generally negligible. When a frame-tube building is subjected to lateral loading the action of the floor system is mainly to transmit the horizontal forces between the different vertical members of the structure. As floor systems does not participate otherwise in the lateral load resistance of the structure, then, provided that the floor loading is constant through out the structure, a repetitive floor loading can be economically used for the design and the analysis of the structure.

If the plan of the frame tube building happens to be a square i.e. is doubly symmetrical about the plan, any applied load can be resolved into two orthogonal force components about the axis, and a twisting moment about the central axis of the structure. The response of the structure can be obtained by a superposition of the separate bending action about the X-X axis and the Y-Y axis and a pure torsion action. Further the individual action may be simplified by utilizing the double symmetry of the structure in plan there by $\frac{1}{2}$ structure analyses or $\frac{1}{4}$ structure analyses can be made.

Usually a three dimensional structural analysis of frame tube buildings can be carried out using stiffness approach, but its feasibility depends on the computer capacity. Though the analysis is possible it is time consuming and very costly, even though it is theoretically straightforward. Analysis of Frame-Tube buildings is a complex problem as they are highly indeterminate structure, consisting of series of rigidly joined frame works connected together at the corners of the buildings. Each frame consists of large number of elements and number of

degree of freedom may run into thousands. Consequently, the size of the stiffness matrix can be considerably reduced and hence the computation, by doing an analysis of $\frac{1}{2}$ structure or even $\frac{1}{4}$ structure if the structure is symmetrical in plan about one or two central axes. The applied load may be treated as a combination of symmetric or skew-symmetric system, acting on either a $\frac{1}{2}$ or $\frac{1}{4}$ structure.

In the building, the high in plane rigidity of the floor slabs will have a considerable influence on the structural behavior, by ensuring that the out of plane deformation of the panels will be effectively restrained at each floor level. The main action of the panels will be effectively restrained at each floor level. Main action will then be in planes of the frame panels. As a result of the in plane rigidity, it may be assumed that the cross section of the building will undergo only rigid body displacements, translation and rotation, in the horizontal plane at each floor level. To obtain an accurate estimate of the structural behavior, it is essential to include this constraining action in the structural analysis of the three-dimensional formwork.

The constraining action can be achieved in a number of different ways. One way is using the option provided by the program, which provide an option for end constraints. If not a fictitious axially rigid joint pin-ended horizontal diagonal bracing members connecting nodes on the opposite corners at each floor level. The restrained corners then remain fixed relative to each other during any translation or rotation under applied loads. In addition, the axial stiffness of the beams at each story level can be assigned to be so large that any in plane axial deformations, or relative displacements between nodes, are negligible. The introduction of such diagonal bracing member has the disadvantage that it increases the bandwidth of the stiffness matrix and increases the solution time.

Reduction of Three-Dimensional Framed Tube to an Equivalent Plane Frame

In this section is presented a simplified yet accurate approximate method for the analysis of symmetrical framed-tube structures subjected to bending produced by lateral forces. The method is intuitively appealing to the engineer since, by recognizing the dominant mode of behavior of the structure, it is possible to reduce the analysis to that of an equivalent plane frame, with a consequently large reduction in the amount of computation required for a conventional full three-dimensional analysis.

Consider initially the framed tube of Fig.7 subjected to bending by lateral forces in the X direction. The lateral load is resisted primarily by the following actions.

1. The shearing actions in the web panels AD and BC parallel to the direction of the applied load.
2. The axial deformations of the normal frame panels AB and DC action effectively as flanges to the web panels.

Due to the symmetry of the structure about the XX axis, and the very high inplane stiffness of the floor slabs, it may be assumed that out-of-plane actions of the web frames are negligible, and the frames are subjected only to planar actions. It is also assumed that the torsion rigidities of the girders are negligible.

The axial displacements of the corner columns in the web frames are restrained by the vertical rigidity of the two normal frames. Consequently, the interaction between the flange and web panels consists mainly of vertical interactive forces through the common corner columns, A,B,C, and D. As a result of these interactive forces, the flange panels AB and DC are subjected primarily to axial

deformation, the uniformity of which across the panel will depend on the stiffness, that is, on the spans and flexural rigidities, of the connecting spandrel beams at each floor level.

Under the applied lateral loading, the shear forces will this be resisted mainly by the web frames, while the bending moments will be resisted by the moments and axial forces in the columns of the web frames and the axial forces in the columns of the flange frames. By virtue of the large lever arm that exists between these flange panels, the wind moments will be resisted most effectively if the maximum amount of axial force can be induced in the columns of frames AB and CD.

All other torsion and out-of-plane actions may be considered to be secondary, apart from the out-of-plane bending of the columns in the flange frames whose horizontal deflections will be the same as those of the web frames. This action may be of significance in the lower levels of the building since bending then occurs about the weaker axis of the columns.

In analyzing the primary mode of behavior, the fundamental compatibility condition that must be established is that of equal vertical displacements at the corner where the orthogonal panels meet. In the analytical model, a mechanism is required that will allow vertical shear forces, but not horizontal forces or bending moments, to be transmitted from the web panels to the flange panels through the corner columns.

In addition, for the web frames, the joints must be free rotate in the plane of the frame, to displace vertically, and to displace horizontally in unison in the plane of the frame at each floor level, sue to the in-plane rigidity of the floor slabs. For the flange frames, the joints must be free to rotate in the plane of the frame, and

to displace vertically. But flange joints on the line of symmetry must, and preferably all flange joints should, be constrained against horizontal displacement in the plane of the frame, as a result of the slab in-plane rigidity.

For example, consider the simple framed tube shown in plan in Fig.9. Since the structure is symmetrical about both center lines XX and YY, only one quarter, for example, EBH, need be considered in the analysis. The required boundary conditions at the lines of symmetry and skew symmetry are then introduced as described because of symmetry about the XX axis, the shear force in the beams, and the slope in the Y direction, of panel AB must be zero at the line of symmetry (E). Conditions of skew symmetry (H) must be zero. If, on the other hand, the web frame BC contains an even number of columns, so that no column is situated on the center line YY, the bending moment at the line of skew symmetry must also be zero. Appropriate support systems to simulate the required boundary conditions for the quadrant EBH of the structure of Fig.9 are shown in Fig10.

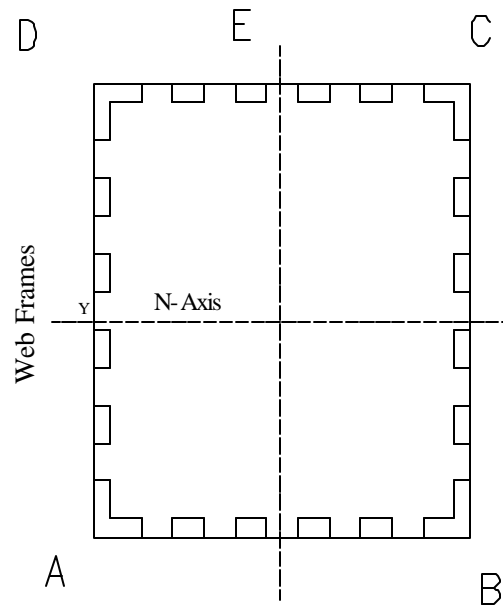


Fig 9
Frame tube structure. Structural plan

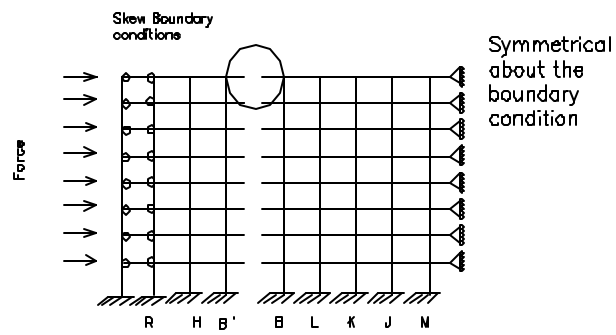


Fig 10
Connection Details

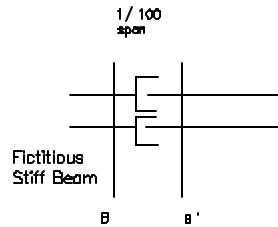


fig10

The detail of a connection

The equivalent planar system is obtained by “rotating” the normal half-panel EB through 90° into the plane of the web-half-panel BH. The in-plane stiffness of the floor slabs constrains all members of the two web bents to have the same horizontal deflection in the X direction; therefore it can be assumed that the one-quarter of the total lateral forces acting on the faces of the flange frames of the building can be applied in the plane of the half-web frame, as indicated in Fig.6. Since the beams are assumed axially rigid, the forces may be applied at any convenient nodes.

The desired vertical interaction between the web and flange panel may be achieved in various ways.

Most comprehensive modern general-purpose structural analysis programs include an inter nodal constraint option. This allows the displacement relating to specified degrees of freedom at two or more nodes in structure to be constrained

to be identical. The appropriate nodes at the intersections of the web and flange frames may then be specified directly in the analysis to have equal vertical displacements.

If this option is not available, some other device must be used to achieve the required vertical compatibility at the junctions.

One simple technique is to displace horizontally the intersection column of each flange frame by a small distance of, say, one-hundredth of the span of the adjacent beams, so that each common intersection joint is represented twice, once on each of the web and flange joints (B and B' in Fig.6). In numbering the nodes, the two nodes representing each intersection joint are numbered separately. The duplicate nodes are then joined by a fictitious stiff beam with a flexural rigidity of say 10,000 times that of the larger adjacent girder and with one end assigned to be released for moment and axial force (Fig.7). By this device, vertical compatibility is established and the required shear transmitted between the web and flange frames, while decoupling the rotation and horizontal displacement. However, it has been found that the results may be sensitive to the stiffness assumed for the fictitious beams.

An alternative technique has been devised to improve the disconnection of the rotations and lateral displacements between the frame panels, and to reduce the sensitivity of the results to the stiffness of the connecting fictitious members. The technique again involves the rotation of the flange frames into the plane of the web. The intersection line column of each frame is shown superimposed on the distance; say less than one hundredth of the story height. Thus each intersection joint is duplicated, once on the web column, and, immediately above, on the flange column. These common nodes should again be numbered separately. Each pair of intersection joint nodes is then connected by a fictitious

stiff vertical link with a large sectional area of, say, 10,000 times that of the intersection line column. The stiff links ensure vertical compatibility and transfer vertical shear between the web and flange frames while disconnecting rotations and vertical displacements.

In each model, the corner column is assigned its true inertia's in the corresponding planes of the web and flange frames, but its area should be assigned wholly to the column B' in the web frame with a zero area assigned to the column B in the flange frame. Horizontal and rotational constraints are applied to the flange frame nodes on the vertical line of symmetry and preferably, as a means of reducing the total number of degrees of freedom, horizontal constraints are also applied to all other flange frame nodes.

The resulting planar model does not include the out-of-plane of columns in the normal frames, which may be of significance in the lower levels. These columns suffer the same horizontal deflections about their weaker axis as the columns in the side frames do about their stronger axis. The effect of the out-of-plane bending may be included in the basic model by adding an equivalent column (RR in Fig.6), whose flexural rigidity is equal to the sum of the out-of-plane flexural rigidities of one-quarter of the flange columns. The additional column is connected by the pin-ended axially stiff links to the existing basic plane frame system. The links constrain the column to have the same horizontal deflection as the side panel members, and allow it to carry its share of the lateral forces. Once the total force and consequent moment has been determined for the effective column, it may be distributed to the individual column in proportion to their flexural rigidities.

Chapter 4

FINDINGS AND DISCUSSION

The frame program developed in this thesis is intended to focus on planar structures. It can handle both trusses and frames. Each element has 2 nodes and each node has 3 degrees of freedom. The program has been tested for many varieties of problems and is fairly stable and the results are very accurate. The program is based on the matrix methods of structural analysis and is developed using Matlab. Care should be taken during the input process as it has a great impact on the result. Input data for 40-story structure and its reduction model, both by program and Mastan2 are as follows,

Frame tube building reduction model:

Data:

Number of nodes = 108 nos.

Number of elements = 184 nos.

Width of the bay = 12 ft

Building dimension in plan = 132ft X 132ft

Story height = 60 ft

Total height = 480 ft

Wind load = 87.5 Kips/node

Floor load = 218.75 kips/column

Actually structure-Frame tube building 40 stories:

Data:

Number of bays = 11 nos

Width of the bay = 12 ft

Building dimension in plan = 132ft X 132ft

Story height = 12 ft

Total height = 480 ft

Column modulus of elasticity (E) = 3150 Ksi, 453600 ksf

Spandrel Beam (E) = 3150 Ksi

Wind load =17.4 Kips/node

Floor load =43.5 kips /column

Mastan2 model for 8 story building:

Data:

Number of Nodes = 108

Number of Elements = 184

Number of Sections = 6

Number of Materials = 1

Number of Supports = 12

Applied Loads

Lateral load=87.5 Kips/node

Floor Load=218.75 Kips/node

Mastan Model 40 story building:

Data:

Number of Nodes = 492

Number of Elements = 920

Number of Sections = 6

Number of Materials = 1

Number of Supports = 12

Applied Loads

Lateral load= 17.5 kips/node

Floor load= 43.5 Kips/node

Comparative result: [reduction model Vs actual model]

40 Story Model [Program Vs. Mastan]

	Program			Mastan Program		
	X Displacement (in)	Y Displacement (in)	θ Rotation (rad)	X Displacement (in)	Y Displacement (in)	θ Rotation (rad)
I order Analysis	2.8934	-1.93430	-0.00041	3.792	-2.2752	-0.00041
II Order Analysis	2.95		-0.00041	3.8676	-2.2836	-0.00054

Reduction Model [Program Vs. Mastan]

	Program			Mastan Program		
	X Displacement (in)	Y Displacement (in)	θ Rotation	X Displacement (in)	Y Displacement (in)	θ Rotation
I order Analysis	3.30	-2.18	-0.00054	3.30	-2.18	-0.00054
II Order Analysis	3.84		-0.00054	3.36	-2.19	-0.00055

From the values of comparative results table we can calculate the drift index of the building for both program and Mastan2. In case of matlab program drift index for first order analysis it is 0.000503 less than the allowable drift index of

0.002., while the drift index for second order analysis of the building we get 0.0006713, which is less than 0.002, the limit value of the drift index. The drift Index calculated for Matan2 model is, first order analysis it is 0.00066 and for second order analysis it is 0.00067, both less than the allowable limits. This confirms the stability of the building for deflection criterion. Comparing the results of actual model with reduction model we see the values are quite close to one another.

When first order analysis is done using the developed program, it is observed that the values when compared between actual and reduction model, a difference of -0.4" (actual model – reduction model) is got. While that for second order analysis it is -0.9". When First order analysis was done using Mastan2, we see that error between models are, +0.49" (actual model – reduction model) and that for second order analysis is +0.5". This is because in 40 story building, story shear causes shear deformations, which is not accounted for in the program developed. The story deflection we get is less than the values calculated using Mastan2 for 40 story building. When number of stories is less, shear deformation is also less, thereby solutions obtained from the program is close to Mastan2 solutions. It is observed that shear deformation causes error to accumulate as the number of stories is increased,

It is observed that the in reduction model, doing a first order analysis the solutions from the program and Mastan are exactly same up to 2 decimal places. The second order analysis results between the models differ by 0.48" this is because drift off error is not controlled in the program unlike the Mastan2.

The results from the buckling load analysis in the case of very tall buildings have to be sorted out and the least positive value is the buckling load and remaining values are the mode shapes. The buckling load ratio from the program is 61. Calculations using Mastan2 yielded approximately 50. The values differ because the program developed calculates buckling load by applying load in one step unlike Mastan2 where load is applied in increments.

For small structures with story height below 8 errors is the same as Mastan2 up to 2 decimal places. The member forces, member stress obtained from this program were very accurate and reliable. Nodal displacement and rotation are also accurate. Program run time is very small in fraction of seconds and the results obtained are accurate. For solving building over 50 stories good computer is a necessity as total degree of freedom of the whole structure increases and we encounter memory problem.

Recommendation for future research:

The program developed is modular program and therefore in future research appending dynamic analysis modules to this program would be a good start.

Iterative program employing Newton Raphsons method has been developed but could not be checked for numerical stability and so has not been included in this work so a future research to append the remaining different second order methods like the work control method, arc length control method and the displacement control method could be attempted.

Finite element method is the most widely used method to solve engineering problem, as matrix method is a back bone of FEM programs, this program would serve as a skeleton to build a advanced Finite element software.

The algorithm still needs to be fine tuned for numerical stability and checked for unique cases.

Finally, I think this program has all that one needs for the development of a basic and simple FEM program, so an effort in this direction is worth a try.

GLOSSARY

First order analysis. An analysis carried out for structures whose deformation is small and exhibit elastic behaviour.

Second order analysis. A iterative method which takes into account the material and geometric nonlinearity.

Elastic buckling load. The load at which the deformation of a slightly imperfect system increases without bound.

Stable equilibrium. A state of equilibrium where in a body is slightly displaced from its original position of equilibrium, will return back to that position subsequent to the removal of the disturbing force.

Unstable equilibrium: A state of equilibrium in which, when a body is displaced slightly from its equilibrium position of rest, does not return, but instead continues to move further away from original equilibrium position.

Neutral equilibrium: A state of equilibrium in which the body remains in the position to which the disturbing force has moved it.

Mastan2. A Matrix structural analysis commercial soft ware.

R e f e r e n c e s

1. Coull, A., and Subedi, N. K., "Framed-Tube Structures for High-rise Buildings" *Journal of the Structural Division, ASCE*, Vol. 97, No. ST8, Proc. Paper 8301, Aug., 1971, pp, 2097-2105.
2. Coull, A., and Bose,B., "Simplified Analysis of Framed-Tube structures," *Journal of the Structural Division, Proceedings of the American Society of Civil Engineers*, Vol.101, No. ST.11, Proc. Paper 11696, Nov 1975,pp.2223-2240
3. Coull, A., and Bose,B., "Torsion of Framed-Tube structures," *Journal of the Structural Division, ASCE*, Vol.102,No.ST.12,December, 1976, pp.2366-2370.
4. Coull, A., and Ahmed, A, K., "Deflections of Framed Tube Structures," *Journal of Structural Division, Proceedings of the American society of civil Engineers*, Vol. 104, No.St5, May, 1978, pp 857- 861
5. Khan F.R and Amin N.R., "Analysis and Design of Framed Tube structure for all tall buildings", *ACI Publication SP-36: Response of multistory Concrete structures to Lateral Forces*, 1973, pp39-60.
6. Khan, F, R., "Current Trends in Concrete High- Rise Buildings," *Proceedings, Symposium on Tall buildings, University of Southampton, Pergamon Press, London, England, Coull and Stafford Smith, eds., 1967, pp.571-590.*
7. J. J. Connor and C.C. Pouangare, "A simple model for the Analysis and Design of Frame Tube Structures."

8. Schwaighofer, J., and Ast, P.A., "Tables for the Analysis of Framed-Tube Buildings," Publication 72-01, Department of Civil Engineering, University of Toronto, Toronto, Canada, Mar., 1972
9. H. De Clercq, "Analysis and design of tube-type buildings structure," PhD dissertation, earth quake engineering research center, college of engineering research center, University of California at Berkeley, ca.
10. Taranath, B. S. "Structural analysis and design of tall buildings," New York, McGraw-Hill, c1988..
11. Alexander Chajes, "Principles of Structural Stability theory "Civil Engineering Mechanics series, 1974, Prentice-Hall, Inc., Engle Cliffs, New jersey.
12. B. S. Smith and A. Coull, Tall Building Structures Analysis and Design, Tall Buildings Structures-Analysis and Design, John Wiley & sons Publication.
13. W. McGuire, R. H. Gallagher, and R. D. Ziemian, Matrix Structural Analysis, Second Edition, John Wiley & Sons, Inc.
14. W. F. Chen, and E. M. Lui, Stability Design of Steel frames, CRC press, Boca Raton, Florida.
15. W. F. Chen, and E. M. Lui, Structural Stability - Theory and Implementation. Elsevier Science Publication Co., Inc.
16. H. Grandin, Jr., Fundamentals of the Finite Element Method, Waveland press, Illinois.

17. P. Jayachandran, M.ASCE, “Structural collapse of the World trade center Towers Subjected to Explosions caused by Aircraft”.
18. Carrie, P.F. “Approximate Analysis and optimization of tall framed tube buildings” Ms Thesis dissertation, Dept of Civil engineering, Worcester Polytechnic Institute, MA.

MANUAL

Matrix Program is simple to use and user-friendly. No prior knowledge of the language of the program is necessary to get started, but to develop complex models knowledge of Matlab is necessary. The programs have a modular structure and user adaptive. This dynamic nature of the algorithm is the start to more advanced programming using matrix methods.

The objective of this manual only to make the user an insight into the program developed and to understand the different variables he inputs into the program with the aid tutorial problems. Section 1 explains the program using the flow chart and algorithms. This manual presents two tutorials. Section 2 explains the input file of a simple 2D Truss, which can be quickly modeled using these programs. This tutorial is recommended as a quick start. New users should go through this tutorial first.

Section 3 presents an input file for frame tube building in two parts. In first part modified frame tube structures are presented. They are developed as RC Framed structure using the GUI-input program devolved which later runs to give the results. They are compared with other commercial software and errors have been presented. Lastly these tutorials shall help to understand the basis for the analysis of the 40 storey structure.

The above tutorial problems include only Analysis and the output are plotted and displayed on the screen as well as in an output file.

Algorithm:

Here is given a series of Algorithm to give a brief insight into the logic of the program.

ALGORITHM

1. Input Structure details: READ:
 - Coordinates
 - Applied Forces
 - Boundary Conditions
 - Element group and Material properties
 - Element Connectivity
2. Construct Stiffness Matrix and Force Vector
 - Construct Element stiffness matrix
 - Group to form global structure stiffness matrix $[\mathbf{K}_e]$
 - Construct Force Vector $\{ \mathbf{F} \}$
3. OPTION : a) Linear b)Nonlinear c)Critical buckling
 - a) LINEAR
 - Compute displacement Vector $\{ \mathbf{U} \} = [\mathbf{K}_e]^{-1} \{ \mathbf{F} \}$
 - Calculate Internal Stresses and Forces
 - Display results
 - STOP
 - b) NONLINEAR
 - Construct element Geometric stiffness matrix initially with zero internal stress
 - Group to form initial global structure geometric stiffness matrix $[\mathbf{K}_g]^i$
 - Total Structure Stiffness matrix $[\mathbf{K}_T] = [\mathbf{K}_e] + [\mathbf{K}_g]^i$
 - Construct $D\mathbf{F} = [\mathbf{F}] / \mathbf{n}$; \mathbf{n} = number of iterations
 - Loop \mathbf{i} to \mathbf{n}
 - Compute $\{ D\mathbf{U} \} = [\mathbf{K}_T]^{-1} \{ D\mathbf{F} \}$
 - Update coordinate displacement Vector $\{ \mathbf{U} \} = \{ \mathbf{U} \} + \{ D\mathbf{U} \}$
 - Compute internal stresses and forces in each element

- Construct new geometric stiffness matrix $[\mathbf{K}_g]^i$ with computed internal stresses
 - Update Total Structure stiffness matrix $[\mathbf{K}_T] = [\mathbf{K}_e] + [\mathbf{K}_g]^i$
- End loop
- Plot Load-displacement Curve for specified node and DOF
- STOP

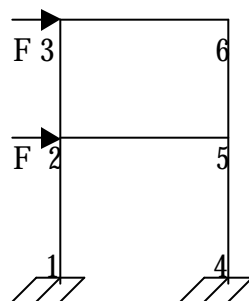
c) CRITICAL

- Construct Element stiffness matrix
- Group to form global structure stiffness matrix $[\mathbf{K}_e]$
- Construct Force Vector $\{\mathbf{F}\}$
- Compute internal stresses and Forces
- With these forces, construct element geometric stiffness matrix
- Construct Structure Geometric Stiffness matrix $[\mathbf{K}_g]$
- Solve $[\mathbf{K}_e + \lambda \mathbf{K}_g] \{\mathbf{D}\} = \mathbf{0}$ for λ and $\{\mathbf{D}\}$ which represent the eigenvalues (critical load factor corresponding to the specified force vector) and the corresponding mode shapes.
- Display results
- STOP

TUTORIAL 1

Description of the Tutorial Problem

The structure for this project is a single bay, two story RC frame that will be analyzed. A first order analysis and Elastic critical buckling load analysis is done and results are presented. Comparative results are also tabulated in the end. The figure below shows the structure.



An input file called "INTRO.txt" containing the input data for the above structure has been provided with the program. This file contains input data.

Basic Data

CONTROL INFORMATION

Number of Nodal Points	= 6
Number of Spatial Dimensions	= 2
Number of Degrees of Freedom	= 3
Number of Element Groups	= 1
Number of Material	= 1

NODAL COORDINATES:

Node	X	Y	Z
1	0.00000000	0.00000000	0.00000000
2	0.00000000	50.00000000	0.00000000
3	0.00000000	100.00000000	0.00000000
4	5.00000000	0.00000000	0.00000000
5	5.00000000	50.00000000	0.00000000
6	5.00000000	100.00000000	0.00000000

BOUNDARY CONDITIONS

Node	X	Y	Z
1	1	1	1
2	0	0	0
3	0	0	0
4	1	1	1
5	0	0	0
6	0	0	0

1= Fixed, 0=Free

APPLIED FORCES

Node DOFs

1	0.00000000	0.00000000	0.00000000
2	1000.00000000	0.00000000	0.00000000
3	100.00000000	-10000.00000000	0.00000000
4	0.00000000	0.00000000	0.00000000
5	1000.00000000	0.00000000	0.00000000
6	0.00000000	-10000.00000000	0.00000000

ELEMENT INFORMATION

Element Group Number 1

A	E	I	G
10	200000	100	0

CONNECTIVITY:

1	1	1	2	0	0	0	0	0	0
2	1	2	3	0	0	0	0	0	0
3	1	4	5	0	0	0	0	0	0
4	1	5	6	0	0	0	0	0	0
5	1	2	5	0	0	0	0	0	0
6	1	3	6	0	0	0	0	0	0

Refer the intro.txt file. The input data is explained below in the following paragraphs.

In the second line we 6 2 3 1 1 all separated by space represent number of elements, number spatial degree of freedom(2D planar), Degree of freedom per node, Element group type (possibilities are 2, i.e either beam-column or truss), type of analysis (1 = Linear Elastic analysis, 2 = Nonlinear analysis, 3 = critical buckling load analysis).

Next 7 lines is for coordinate positions. For example, 1 0 0 0 all separated by spaces represent node number, co ordinate position in x, coordinate position in y and lastly interpolation decision variable (1 invokes interpolation loop, if there are series of nodes then instead of entering one by one it can be done in 1 step). On the 9 line we have 0 which represents end of data for read_coord module program.

Lines 10 11 12 are for support condition. On line 10 we have 1 1 1 1 0 which represent node number, fixed x Dof, fixed y Dof, fixed rotation about z axis, no interpolation. Note 0 represent free and 1 represents fixed condition.

Lines 13-17 are for force. On line 13 we have 2 1000 0 0 0 represent node number, Force in the x-direction, Force in the Y -Direction, rotation about the z axis and no interpolation. Note -ve sign represent force towards left of node and downwards direction. On 17 line 0 represent termination of the respective program module.

On Line 18 we have 1 4 6 1 representing element group number, element type (3 represent truss type and 4 represents beam type. Don't think what happened to 1 and 2 that's not your problem), number of elements, number of materials used. On Line 19 we have 1 10 200000 100 0 representing Material number, area of cross section, Modulus of elasticity, moment of inertia and Shear modulus. The remaining input data is for node connectivity. On line 20 we have 1 1 1 2 0 representing element number, material number, i^{th} node, j^{th} node and no interpolation.

This completes the input file. Fem_main is the main program which needs to be invoked then type the input file name and then program run to finish. For more logical understanding refer the algorithm and flow chart.

OUTPUT

PHI OUTPUT

Node No.	Phi		
	X	Y	θ
1	0.00000	0.00000	0.00000
2	1.65683	-0.14009	-0.04440
3	3.94695	-0.38465	-0.04616
4	0.00000	0.00000	0.00000
5	1.65683	-0.35991	-0.04440
6	3.94683	-0.61535	-0.04616

Elastic Critical Buckling Load: (1)

0.78
5.79
19.01
39.71
82.95
163.89
200.15
200.00

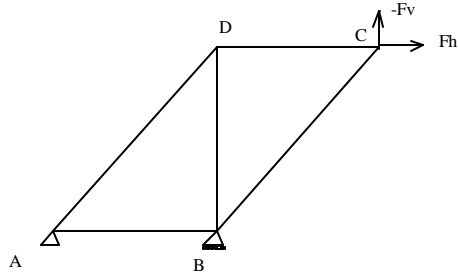
Comparative results:

<i>Program</i>			<i>Mastan</i>		
X	Y	q	X	Y	q
0.00000	0.00000	0.00000	0.0000	0.0000	0.0000
1.65683	-1.4009	-0.04440	1.6568	-1.4009	-0.044398
3.94695	-3.8465	-0.04616	3.9470	-3.8465	-0.046163
0.00000	0.00000	0.00000	0.0000	0.0000	0.0000
1.65683	-3.5991	-0.04440	1.6568	-3.5991	-0.044398
3.94683	-6.1535	-0.04616	3.9468	-6.1535	-0.046162

TUTORIAL 2

PROBLEM DICRIPTION:

The structure for this project is a 2D planar truss that will be analyzed. A first order analysis is done and results are presented. Comparative results are also tabulated in the end. The figure below shows the structure.



2D Planar Truss

CONTROL INFORMATION:

Number of Nodal Points = 4
 Number of Spatial Dimensions = 2
 Number of Degrees of Freedom = 2
 Number of Element Groups = 1

NODAL COORDINATES

Node	X	Y	Z
1	0.00000000	0.00000000	0.00000000
2	5000.00000	0.00000000	0.00000000
3	5000.00000	8670.00000	0.00000000
4	10000.0000	8670.00000	0.00000000

BOUNDARY CONDITIONS

Node DOFs

1	1	1
2	0	1
3	0	0
4	0	0

APPLIED FORCES AND DISPLACEMENTS

Node DOFs

1	0.00000000	0.00000000
2	0.00000000	0.00000000
3	0.00000000	0.00000000
4	282840.0000	-282840.0000

ELEMENT INFORMATION

Element Group Number 1

CONNECTIVITY

1	1	1	2	0	0	0	0	0	0
2	2	1	3	0	0	0	0	0	0
3	2	2	3	0	0	0	0	0	0
4	1	3	4	0	0	0	0	0	0
5	2	2	4	0	0	0	0	0	0

Referring to the input file truss.txt, Input basically is the same with changes as to Dof per node which is 2 in this case.

Output: Nodal displacement

PHI OUTPUT

Node No.	Phi	
	X	Y
1	0.00000	0.00000
2	-0.40779	0.00000
3	9.83626	-2.23479
4	10.95114	-7.80813

Comparative results:

<i>Program</i>			<i>Mastan</i>		
X	Y	q	X	Y	q
0.00000	0.00000	0.00000	0.0000	0.0000	0.0000
-0.40779	0.00000	0.00000	-0.40779	0.0000	0.0000
9.83626	-2.23479	0.00000	9.8363	-2.2348	0.0000
10.95114	-7.80813	0.00000	1.0951e+001	-7.8081	0.0000

A P P E N D I X - B

Frame Tube Reduction model of the building:

Data:

Number of bays = 11 nos.

Width of the bay = 12 ft

Building dimension in plan = 132ft X 132ft

Number story = 8 nos.

Story height = 60 ft

Total height = 480 ft

Column modulus of elasticity (E) = 3150 ksi, 453600 ksf

Spandrel Beam (E) = 3150 ksi

Wind load = 87.5Kips/node

Floor load = 218.75 kips /node

Strength of concrete = 3000 ksi

Frame tube building reduction modeling technique

Lumped girder Method:

In this case reduction modeling is done to reduce the structure from 40 story to 8 story for ease of analysis and data manipulation. The wind load is 87.5kips/ node and the floor load is 218.75.

1-20 STORIES,

For columns having $I_c = 182250\text{in}^4$ and area $A_c = 1080\text{in}^2$ and spandrel beam having $I_g = 182250\text{in}^4$

And area $A_g = 1080\text{in}^2$ the equivalent inertias are

$$I_{ce} = \frac{n^3}{\sum_1^n \left(\frac{1}{I_c} \right)} = \frac{5^3}{\left(\sum_1^5 \frac{1}{182250} \right)} = 4556250in^4$$

$$I_{ge} = \sum_1^n I_g = \sum_1^5 182250 = 911250in^4$$

20-30 STORIES,

For columns having $I_c=129654in^4$ and area $A_c= 882in^2$ and spandrel beam having $I_g=129654in^4$ And area $A_g=882in^2$ the equivalent inertias are

$$I_{ce} = \frac{n^3}{\sum_1^n \left(\frac{1}{I_c} \right)} = \frac{5^3}{\left(\sum_1^5 \frac{1}{129654} \right)} = 3241350in^4$$

$$I_{ge} = \sum_1^n I_g = \sum_1^5 129654 = 648270in^4$$

30-40 STORIES,

For columns having $I_c=111132in^4$ and area $A_c= 756in^2$ and spandrel beam having $I_g=111132in^4$ And area $A_g=756in^2$ the equivalent inertias are

$$I_{ce} = \frac{n^3}{\sum_1^n \left(\frac{1}{I_c}\right)} = \frac{5^3}{\left(\sum_1^5 \frac{1}{111132}\right)} = 2778300in^4$$

$$I_{ge} = \sum_1^n I_g = \sum_1^5 111132 = 555660in^4$$

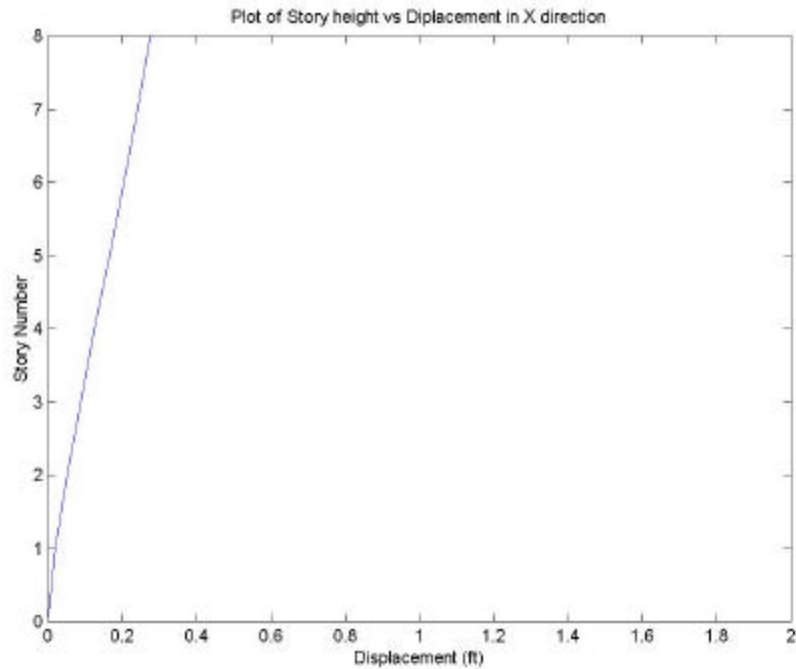
Reduction model section properties,					
Stories		I _{ge} (in ⁴)	I _{ce} (in ⁴)	Ag (in ²)	Ac (in ²)
1-20	1-4	911250	4556250	1080	1080
20-30	4-6	648270	3241350	882	882
30-40	6-8	555660	2778300	756	756

Reduction model section properties,					
Stories		I _{ge} (ft ⁴)	I _{ce} (ft ⁴)	Ag (ft ²)	Ac (ft ²)
1-20	1-4	43.9	219.7	7.5	7.5
20-30	4-6	31.26	156.31	6.125	6.125
30-40	6-8	26.79	133.98	5.25	756

The results of the reduction analysis are presented in tabulated form in end. The results of the program are compared with the Mastan results, and the comparative charts are presented in the succeeding pages to follows.

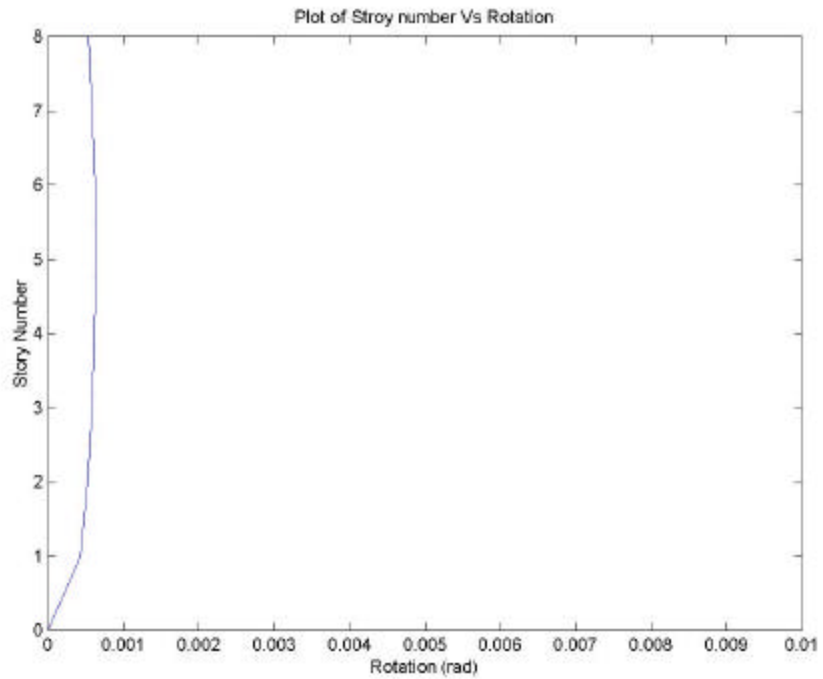
The plots of story height vs. displacement clearly show that the displacement at the top of the building is 0.2751 ft or 3.30in

and the result from the program and Mastan is 0.275ft or 3.30in.



Plot 1

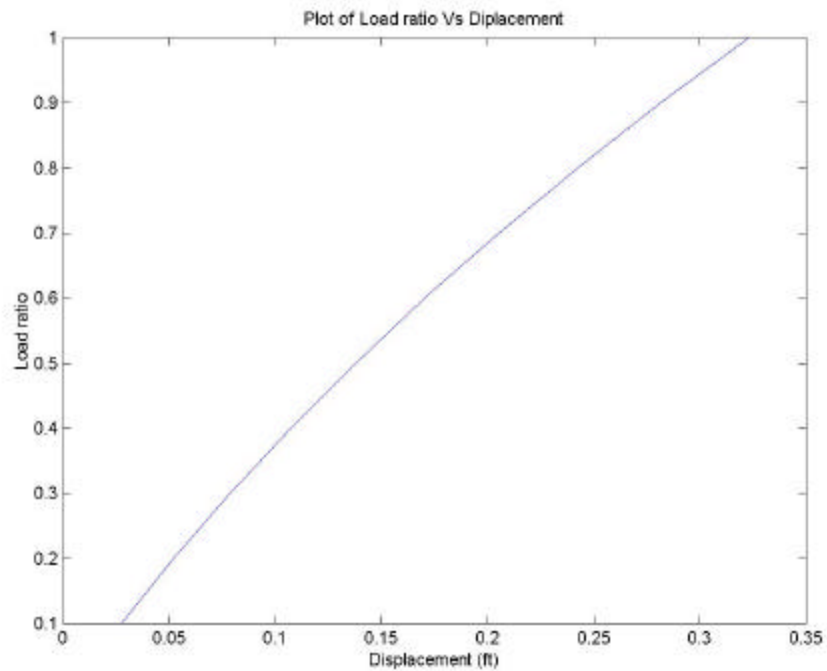
The plot of rotation Vs height of building is presented below. The plot of rotation gives the cumulative rotation along the height of the building. The plot below shows that the moment at the bottom is high and gradually at the top becomes zero, which is indicated by high values of rotation at the bottom and very low increase in the values of rotations in the middle portion and no rotation at the top portion of the building. The results match with Mastan results.



Plot 4

Second order analysis:

A simple step method is employed for the analysis. Max iteration specified is 10, greater the iteration greater will be the accuracy. Load increment is 0.1×200 (the reference load). The plot indicates a slight nonlinearity with max displacement being 0.32 ft or 3.84 inches. Mastan program gives the deflection value to be 0.28ft or 3.36.



Mastan2 model for 8 story building:

Input for Structural Analysis

- (i) Number of Nodes = 108
- (ii) Number of Elements = 184
- (iii) Number of Sections = 6
- (iv) Number of Materials = 1
- (v) Number of Supports = 12
- (vi) Applied Loads

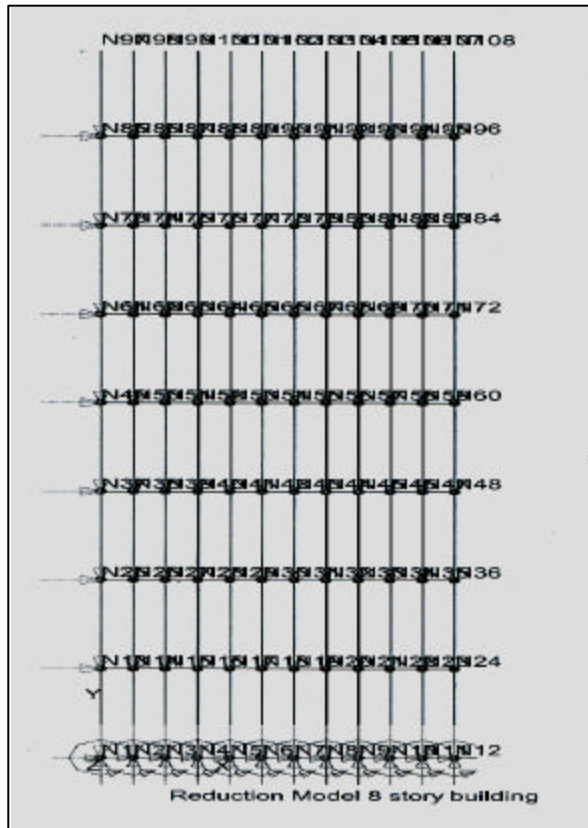
lateral load=87.5 Kips/node

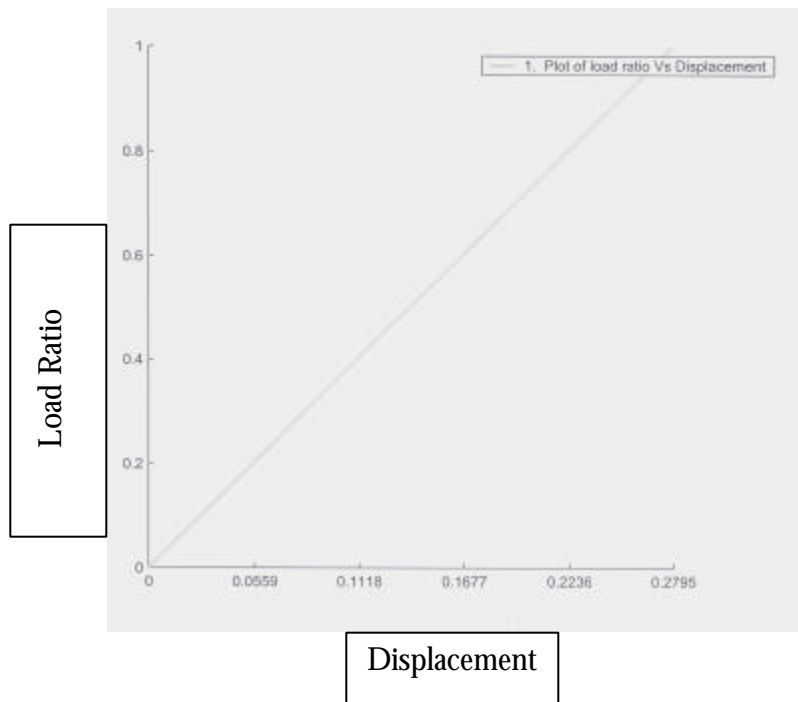
Floor Load=218.75 Kips/node

The Mastan model diagram is given below and comparative solution is also presented.

Comparative results:

	Program			Mastan Program		
	X Displacement (ft)	Y Displacement (ft)	θ Rotation	X Displacement (ft)	Y Displacement (ft)	θ Rotation
I order Analysis	0.275	-0.1815	-0.00054	0.275	-0.1815	-0.00054
II Order Analysis	0.320		-0.00054	0.280	-0.1822	-0.00055





Mastan Plot of Displacement Vs Load Ratio

Actually structure-Frame tube building 40 stories:

Data:

Number of bays = 11 nos

Width of the bay = 12 ft

Building dimension in plan = 132ft X 132ft

Story height = 12 ft

Total height = 480 ft

Column modulus of elasticity (E) = 3150 Ksi, 453600 ksf

Spandrel Beam (E) = 3150 Ksi

Wind load =17.4 Kips/node

Floor load =43.5 kips /column

Floor	Column dimension (in)			Spandrel beam (in)			Concrete (Ksi)
	B	D	H	B	D	H	F _c
1-19	24	45	144	24	45	144	3000
20-29	21	42	144	21	42	144	3000
30-40	18	42	144	18	36	144	3000

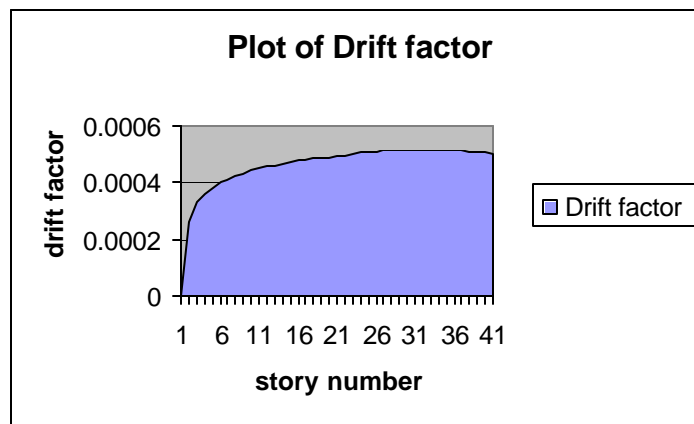
Floor	Column dimension (in)			Spandrel Beam (in)			Concrete
	B	D	H	B	D	H	F _c (ksi)
1-19	24	45	144	24	45	144	3000
20-29	21	42	144	21	42	144	3000
30-40	18	42	144	18	42	144	3000

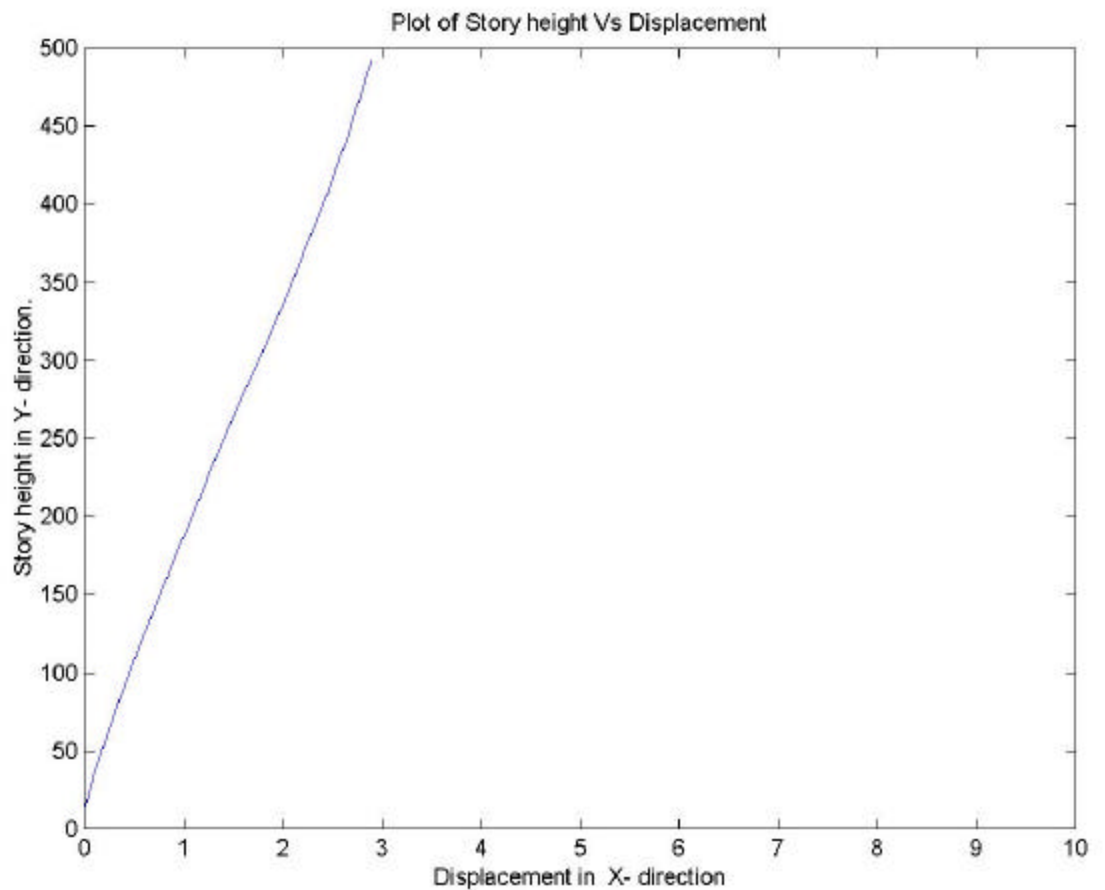
Flexural strength:

Story	Columns and spandrel beam Moment of inertia and Area		
	Strong axis I (in ⁴)	Weak Axis I (in ⁴)	Area (in ²)
1-19	182248.7	51840	1008
20-29	129662.21	332410.368	882
30-40	111186.43	20404.224	756

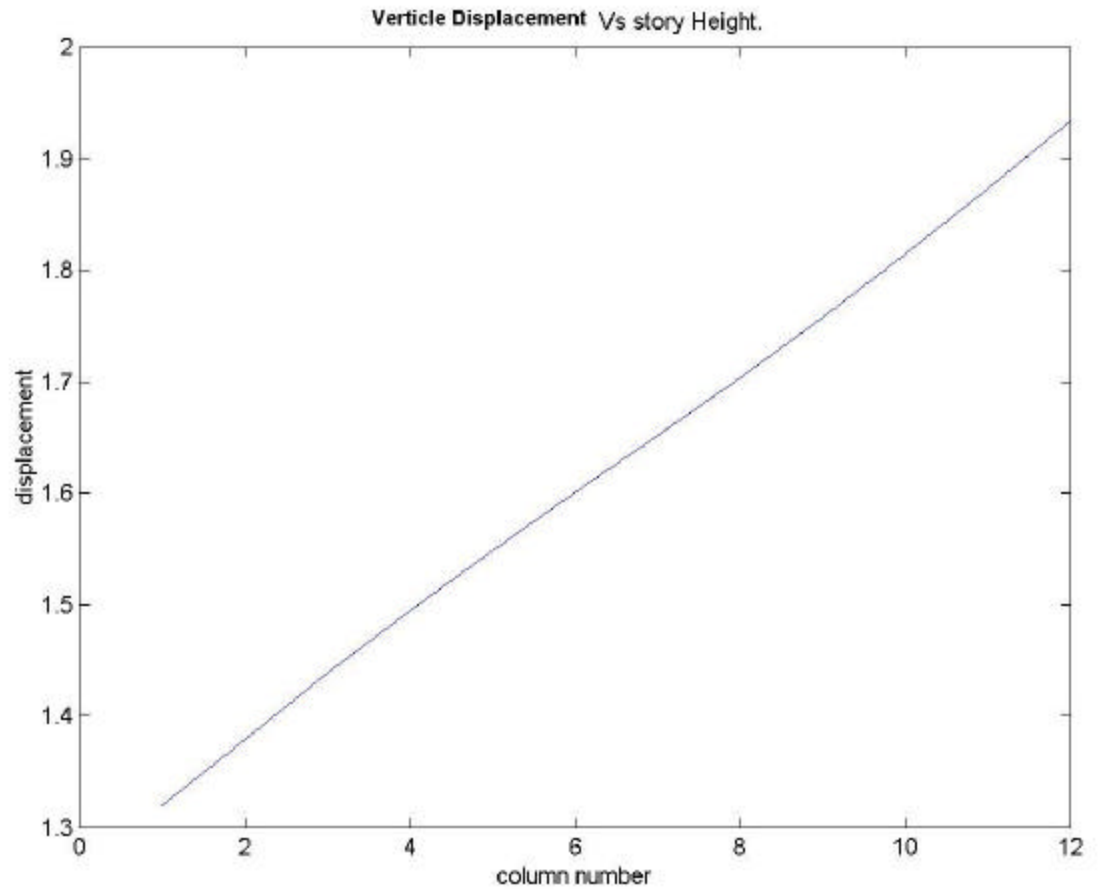
The results of the reduction analysis are presented in tabulated form in end. The results of the program are compared with the Mastan results, and the comparative charts are presented in the succeeding pages to follows.

The plots of story height vs. displacement clearly show that the displacement at the top of the building is 2.89348in and the result from the program and Mastan is exactly the same up to 4th decimal place. The drift factor of the buildings is 0.0005017. The drift factor for 40 stories

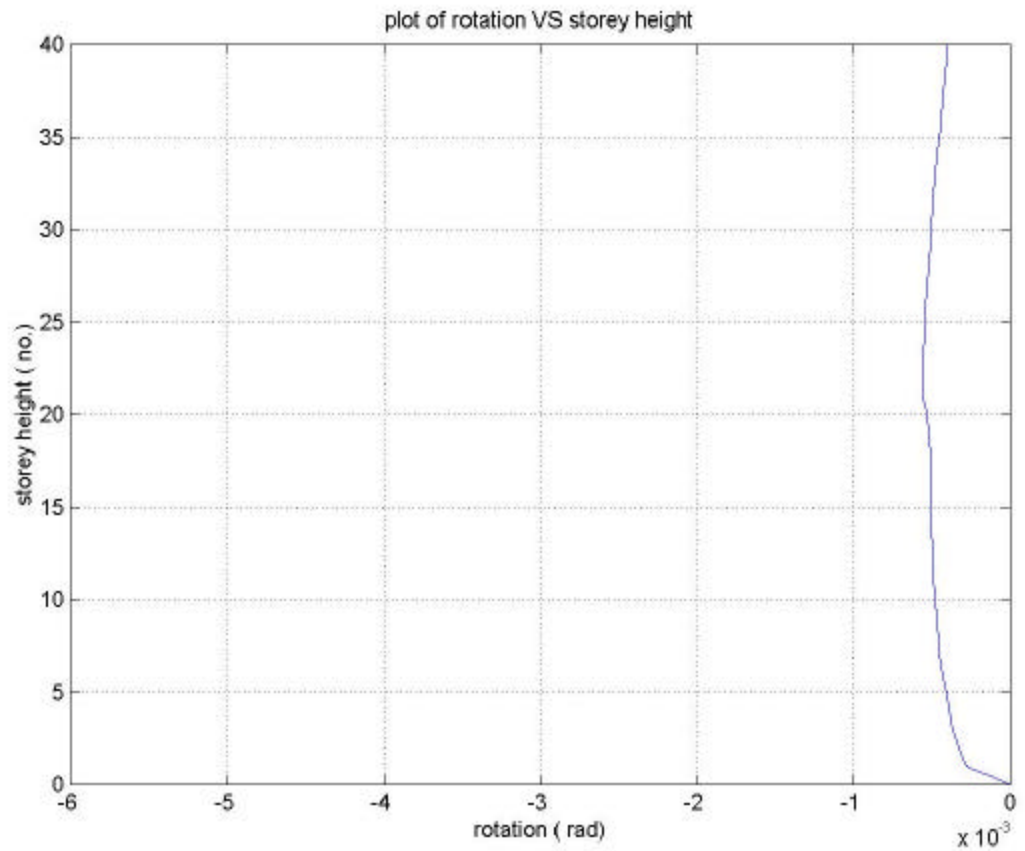




The plot of Vertical displacement Vs distance along the perimeter is presented below. The vertical displacement of the columns along the perimeter in the plan is considered. The results obtained from Mastan are compared with the program and plot shows that the results match with one another.



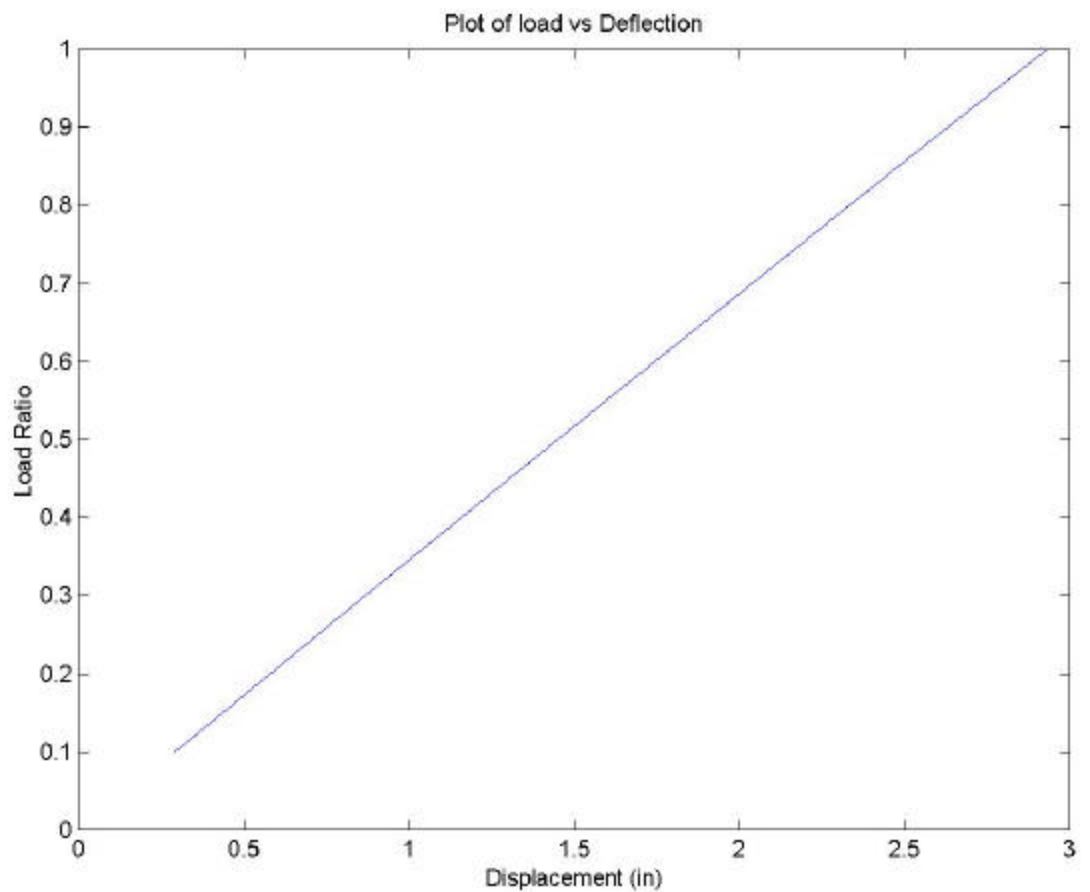
The plot below is plot of rotation along the story height. The rotations are in radians and for complete rotation of all the nodal points refer the out put file.



Second order analysis:

Results from second order analysis compared are presented in the form of a graph of Load ratio versus displacement. The graph indicates that the deflection is around 2.95. The linear nature of the curve and the proximity of the result close to 2.89 obtained from Ist order analysis clearly indicates that there is not much of a second order effect taking place. The structure there is stable. The curve obtained from Mastan-2 is also presented and from the graph and it is clear that the deflection is 3.8 in. The drift factor of the building is

0.0005121. The limit value for drift factor is 0.002. The buckling load factor for the building calculated using the program is 61.6. The elastic critical load ratio from Mastan is 47.



Mastan Model 40 story building:

General Information Categories:

- (i) Number of Nodes = 492
- (ii) Number of Elements = 920

(iii) Number of Sections = 6

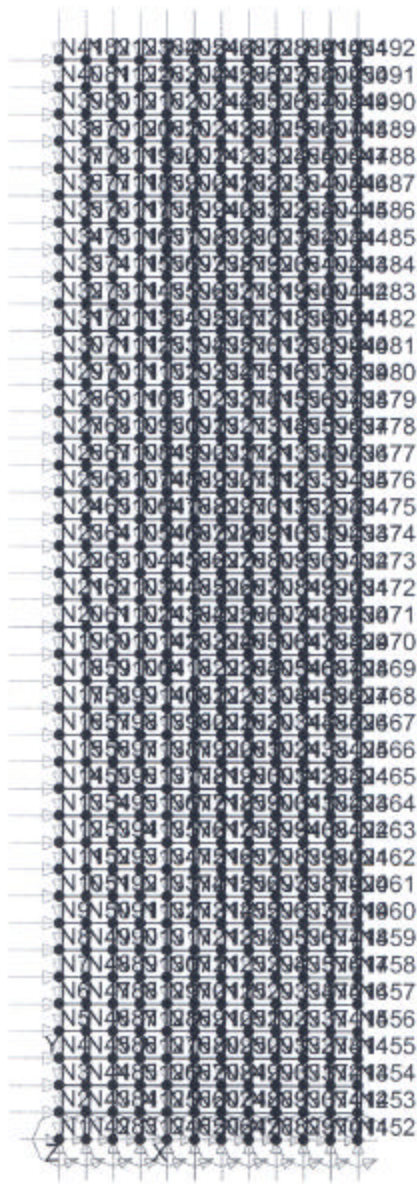
(iv) Number of Materials = 1

(v) Number of Supports = 12

(vi) Applied Loads

Lateral load= 17.5 kips/node

Floor load= 43.5 Kips/node



40 story Building

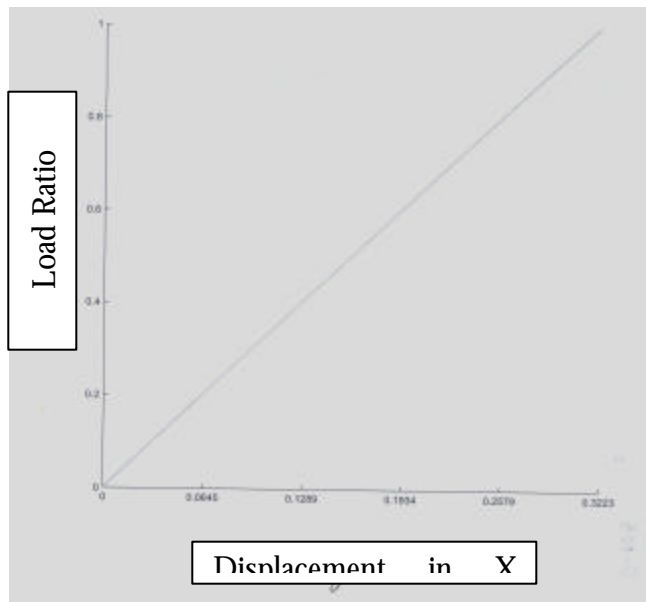


Fig Mastan Load Ratio Vs Displacement

Comparative result: [reduction model Vs actual model]

40 Story Model [Program Vs. Mastan]

	Program			Mastan Program		
	X Displacement	Y Displacement	θ Rotation	X Displacement	Y Displacement	θ Rotation
I order Analysis	2.8934	-1.93430	-0.00041	3.792	-2.2752	-0.00041
II Order Analysis	2.95		-0.00041	3.8676	-2.2836	-0.00054

Reduction Model [Program Vs. Mastan]

	Program			Mastan Program		
	X Displacement (in)	Y Displacement (in)	θ Rotation	X Displacement (in)	Y Displacement (in)	θ Rotation
I order Analysis	3.30	-2.18	-0.00054	3.30	-2.18	-0.00054
II Order Analysis	3.84		-0.00054	3.36	-2.19	-0.00055