EVALUATION OF ISDS SOFTWARE

by

Nadella Navin Kumar

Project submitted to the Faculty of the
Virginia Polytechnic Institute and State University
in partial fulfillment of the requirements for the degree of
Master of Engineering
in
Civil Engineering

APPROVED:

Richard M. Barker
R. M. Barker, Chairman

Siegfried Holzer
S. M. Holzer,

Don A. Garst
D. A. Garst,

July, 1991
Blacksburg, Virginia

Acknowledgements

The author would like to take this opportunity to offer his sincere thanks to Dr. R. M. Barker, his major advisor, for guidance, suggestions and patience and for providing necessary materials, information and constructive criticism throughout this project.

The author is also grateful to Dr. S. M. Holzer and Professor D. A. Garst for being on his committee.

Thanks also goes to the author's roommates, who had been a constant source of encouragement and moral support during the entire project.

Finally the author is extremely thankful to his parents without whose moral and financial support his graduate studies at VPI & SU would not have been possible.

Table of Contents

List of Illustrations

List of Tables

Chapter 1

INTRODUCTION

1.1 GENERAL

Developments in the recent past in computer technology have resulted in personal computers with capabilities which equal or even exceed the computing power of many minicomputers, and even some mainframe systems, of only a few years ago. It is now possible to handle problems, on an inexpensive computer sitting on the individual users desk, which previously required the use of a much larger system costing several hundred thousand dollars or more.

Of course, for computers to handle problems, suitable software must be available. Although the personal computer (PC) has a tremendous potential in the field of structural analysis and design, this potential can only be realized if suitable software is available to the structural engineer which will perform the required operations. Although a majority of the initial programs which were developed for the PC were of the entertainment variety, a few users began to experiment with programs aimed at business and engineering applications. Before long programs which would perform many business functions began to appear.

At the same time, specialized structural analysis procedures were being developed which were ideally suited for use on computers, and a number of very powerful computer programs such as STRESS, STRUDL, and NASTRAN were being written which used these procedures. Unfortunately these programs were available only on mainframe or minicomputers, and many small engineering companies, who cannot afford to own or lease this type of equipment, find that it is very expensive and inconvenient to use these analysis programs through a computer service company. A similar situation exists in many colleges and universities where the students find it is very inconvenient to use the mainframe computers in their structural analysis and design courses, either due to restrictions on the use of computer or to the selection of suitable software. Thus, there is a need at present time for structural analysis software which will run on a low cost PC sitting on the users desk. Such software should be able to handle most of the day-to-day analysis problems which would be encountered in a structural design office or in a typical structural engineering analysis or design course. It should also be convenient to use and it should not require extensive knowledge in computer programming or a complete understanding of the theory governing the analysis procedures which are used. Finally it should be inexpensive to obtain and easy to operate [Fleming, 1986].

The integrated structural design system (ISDS) software package marketed by Research Engineers Incorporation, Marlton, NJ, satisfies all the above requirements and hence is being used extensively in many engineering offices and schools. The ISDS program is composed of a number of modules with the prefix STAAD and is often identified by this name as well.

1.2 OBJECTIVE

The main objective of the project will be to evaluate ISDS's ability to analyse and design a multistory reinforced concrete frame subjected to gravity, wind, and earthquake loads and various combinations of the same. Analysis results will be checked by another frame analysis program (FAP) which was written by Holzer (1985). The results from ISDS will also be compared with hand calculations made using conventional reinforced concrete design methods.

1.3 OUTLINE OF THE CHAPTERS

Chapter Two discusses the background knowledge that the user must have in order to use the package effectively. Chapter Three presents the capabilities of the ISDS package and discusses the theoretical approach adopted and the assumptions made in analyzing and designing steel and concrete structures. Chapter Four gives general description of ISDS. It discusses the theoretical approach adopted by ISDS in solving the steel and concrete structural problems. It gives a brief description of the sign conventions used in the coordinate system of the ISDS package. Chapter Five gives a general description of the example problems considered for the purpose of establishing ISDS's ability to analyze and design structures subjected to gravity, wind, and earthquake loadings and their different combinations. It discusses the portal method for analyzing frames subjected to wind load and the assumptions made therein. Chapter Six presents the results obtained from the study and why the results from ISDS, FAP, and conventional design methods differ and also discusses the significance of the same. Finally in Chapter Seven the advantages and disadvantages of the package are discussed and further improvements that can be made in the package to make it more efficient are suggested.

Chapter 2

USER BACKGROUND

Since the objective of the project is to establish capabilities of ISDS package, it will be appropriate to discuss the background knowledge one must possess in order to use the package effectively.

In the writer's opinion, it is not necessary for the user of the ISDS package to be an expert in computer programming or to be familiar with the theory behind the structural analysis procedures which were used in developing these programs. The user must be familiar with the basic behavior of structural systems under applied loads and should understand the fundamental concepts of equilibrium and stability. A background equal to that which would be presented in most undergraduate introductory courses in structural analysis should be sufficient. It is assumed that the user of this package has some experience in development of mathematical models of structural systems. To use the package effectively, the user must be able to make sound engineering judgments concerning the properties of the mathematical model and the loadings to which it will be subjected so that the results of the analysis accurately represents the behavior of the structure. The ability to develop the mathematical models for complex structural systems can only be achieved through experience. Therefore the ISDS package is not intended for use by complete novices in the field of structural analysis unless it is being used as a learning tool to gain knowledge concerning the behavior of structural systems. The program will perform a correct analysis of the mathematical model as it is specified by the user; however, the user must make correct choices, based upon sound engineering principles, in the development of the mathematical model. For example, the following are several example situations which might be encountered :

1. Is a support joint for a particular frame structure totally fixed against translation and rotation, or is it free to rotate? In the first case, the support would have to supply a moment restraint to the joint, while in the second case the moment at the joint would be zero. The stress induced in the structure and the displacements caused by the loads would be different for each case [Fleming, 1986].

2. Are the members of the structure rigidly attached to the joints at their ends, or can they be considered to be pinned for analysis purposes? For the first case the structure would be analyzed as a frame, while for the second case a truss analysis would be correct. The results of these two analyses may not be the same in all cases [Fleming, 1986].

3. Can a particular structure be assumed to behave as a plane system for analysis purposes, or must three-dimensional action be considered? A two-dimensional analysis is much simpler and less time consuming than a three-dimensional analysis. Many structural systems behave essentially as a two-dimensional system and can be analyzed as a plane structure. However for many structures a three-dimensional analysis is required to accurately represent the behavior of the system. The user must decide whether the structure is to be analyzed as a two-dimensional or as a space structure [Fleming, 1986].

4. What preliminary member sizes are to be chosen? ISDS is not capable of selecting member sizes on its own for a given set of loadings. The initial member sizes have to be given by the user and the ISDS, after analyzing and designing, only indicates if the member prescribed can take the given loading without failure. Therefore the user must know some approximate

methods of analysis, so that he can arrive at a preliminary section and thus reduce the number of runs on the computer for getting an economical design.

5. What preliminary material stresses are to be chosen? An experienced user may choose an appropriate material with certain stress properties on a first trial, based upon the given set of loadings and dimensions of members already chosen. This may not be the case when an inexperienced person is using the package.

6. Decide whether the least amount of steel was prescribed for structural elements for a given set of loading, material stresses, and member properties. For example, in the design of columns, ISDS package gives the number of longitudinal bars in a column in multiples of four. Also the minimum diameter of the bar that can be used as a stirrup in a beam or as a tie in a column is 0.5 inches. But if the user knows some preliminary design procedures and specifications for minimum requirements of steel in concrete members, he can make a quick calculation as to the actual amount of steel required, thus dispensing with the overly conservative amount of steel prescribed by ISDS.

In each of the cases listed above, and in many other situations which might be encountered, the analyst must understand the basic behavior of structural systems and must be able to translate the actual structure into a mathematical model which can be analyzed using a computer program. During this process, if the properties of the mathematical model are not chosen correctly, the results of the analysis can be completely unrealistic, and in some cases can lead to erroneous and dangerous conclusions concerning the response of the structure to applied loads. Thus an inexperienced user can obtain unrealistic results from a structural analysis program due to incorrect assumptions when developing the input data for the program. It is not sufficient to accept the results of a structural analysis merely because they were obtained from the computer [Fleming, 1986]. The user must verify that the results are realistic and are representative of the actual behavior of the structure.

The major advantage of the computer is that it frees the user from the tedious, time-consuming, and error-prone calculations which are required to perform the analysis of a complex structural system. Therefore the user can spend more time considering the behavior of the structure, since the analysis time and effort are greatly reduced. If there are any questions in the user's mind as to the properties which should be used for the mathematical model of the stucture, several different cases may be considered. The results of these various analyses can then be compared and the results which appear to be most realistic, based on the experience of the user, can be used. A PC is particularly useful for this purpose since the user has a dedicated computer sitting on his or her desk and is not limited by the budget which is available for renting computer time.

ISDS is a very good example of a package that is easy to use and one that can be run on a PC and hence manifests all the usefulness described above.

Chapter 3

ISDS - CAPABILITIES

Integrated structural design system (ISDS) is a fully integrated structural engineering software package capable of structural analysis, design and drafting - all within the same program. As the name suggests, ISDS integrates these different yet related activities into a single continuous process. ISDS is made up of several functional modules, each with a prefix STAAD (Structural Analysis and Design), namely STAAD-III, STAAD-PL, STAAD-DRAFT, STAAD-UTIL, and each fully capable of performing its own task [ISDS - program user's manual, 1989].

The STAAD-III structural analysis module is capable of performing both steel and concrete design working within the guidelines of the standard codes selected by the user. Available codes for such design include American Institute of Steel Construction (AISC), working strength, and Load and Resistance Factor Design (LRFD) specifications, American Concrete Institute (ACI), American Association of State and Highway Transportation Officials (AASHTO), and British Standard Institution (BSI) codes. A number of other foreign codes of practice are also available to choose from which are well maintained to reflect new modifications.

STAAD-PL is a complementary graphics program to STAAD-III, serving as both a preprocessor and postprocessor. It can be used for model verification. Once the input file has been created, STAAD-PL can interpret it and display the structure (See Figure 1). The display can then be manipulated, using suitable commands, to show portions of the structure in greater detail. In addition, a hardcopy output on plotters and printers can be generated.

The STAAD-III analysis and design results can be graphically verified using STAAD-PL's post processing features. For this purpose, the necessary data files must be created during the STAAD-III run, by inserting suitable commands in the input file. STAAD-PL can then display bending moment and shear force diagrams, deflected shapes, section displacements, stress contours and failure diagrams of the structure, based on the analysis results (See Figure 2).

The STAAD-DRAFT drafting module uses information held in a central database to generate full scale design drawings complete with member designations and dimensions. It can also be used independently as a drafting package (See Figure 3).

The STAAD-UTIL utility module helps in managing files, plotting drawings, setting up device configurations, communicating with peripherals, and interacting with the operating system. This module has submodules like STAAD-VIEW, STAAD-ED, STAADPRINT etc. The STAAD-VIEW submodule allows the user to view the results of analysis and design of a structure. He can then take printouts of only a selected portion of analysis results that he wants (See Figure 4). The STAAD-ED module is where the user writes down his program, i.e., a set of commands which leads to the analysis and design of the structure (See Appendix A).

A recent trend in the structural software market has been to provide interfaces (links) between available application software and popular CAD packages. But from a everyday user's point of view it can be cumbersome and inefficient to introduce CAD capabilities to an available application software since the process involves purchasing, maintaining and learning more than one program [ISDS - program user's manual, 1989]. But ISDS obviates such a situation for structural engineers

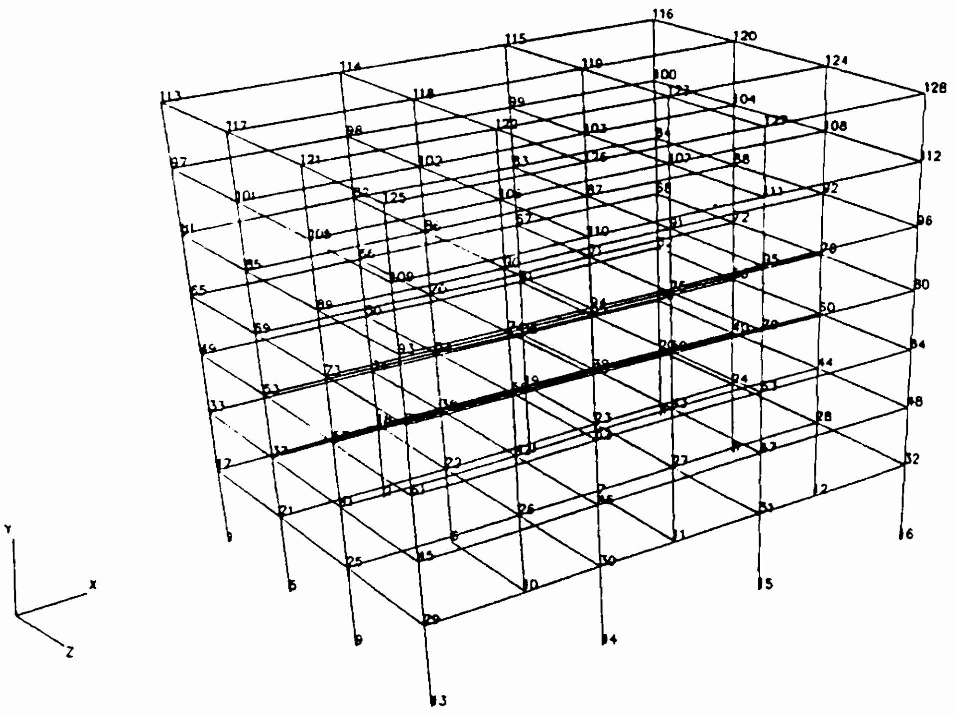
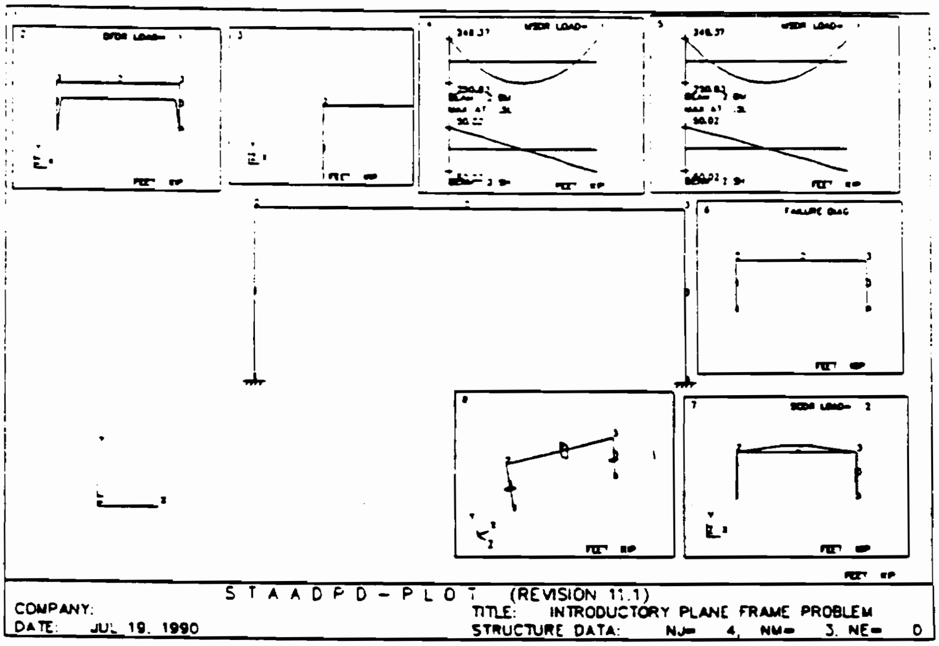
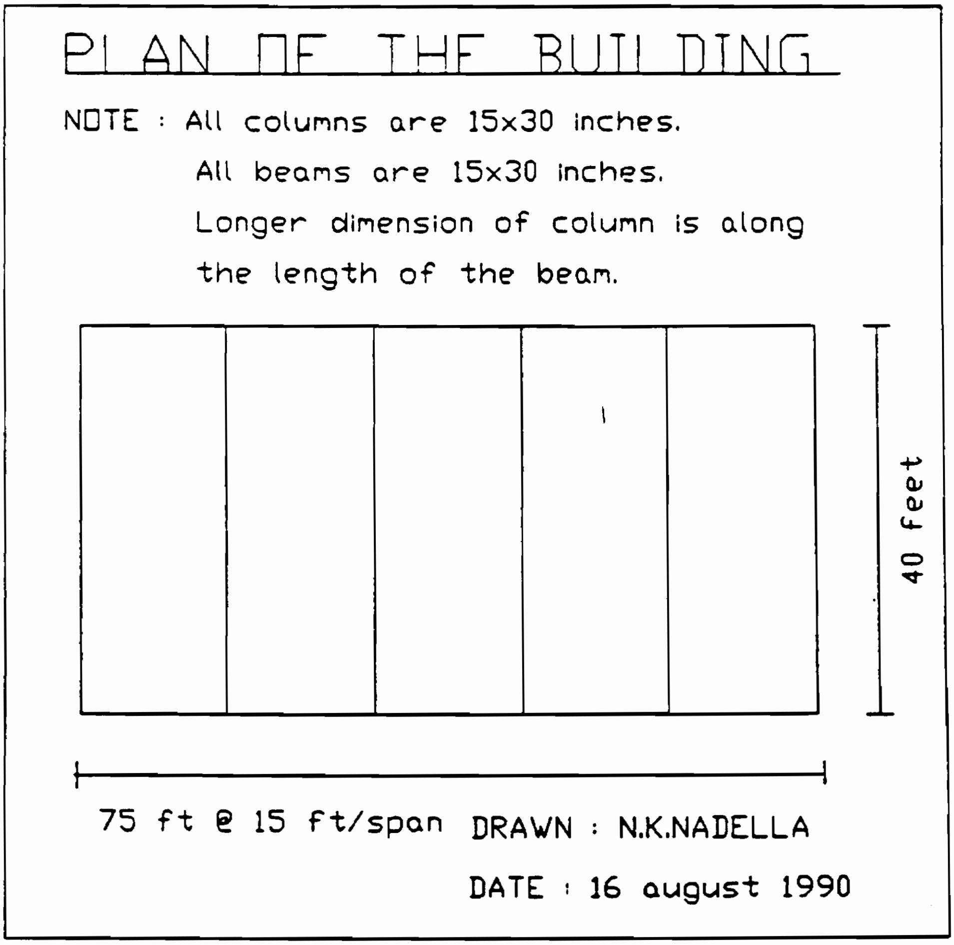


Figure 1. Preprocessing feature of STAAD-PL.



STAADPD-PLOT (REVISION 11.1)
COMPANY:
DATE: JUL 19. 1990
TITLE: INTRODUCTORY PLANE FRAME PROBLEM
STRUCTURE DATA: NJ= 4, NM= 3. NE= 0

Figure 2. Post-processing features available in STAAD-PL.



PLAN OF THE BUILDING

NOTE : All columns are 15x30 inches.
All beams are 15x30 inches.
Longer dimension of column is along
the length of the beam.

DRAWN : N.K.NADELLA
DATE : 16 august 1990

Figure 3. Output from STAAD-DRAFT.

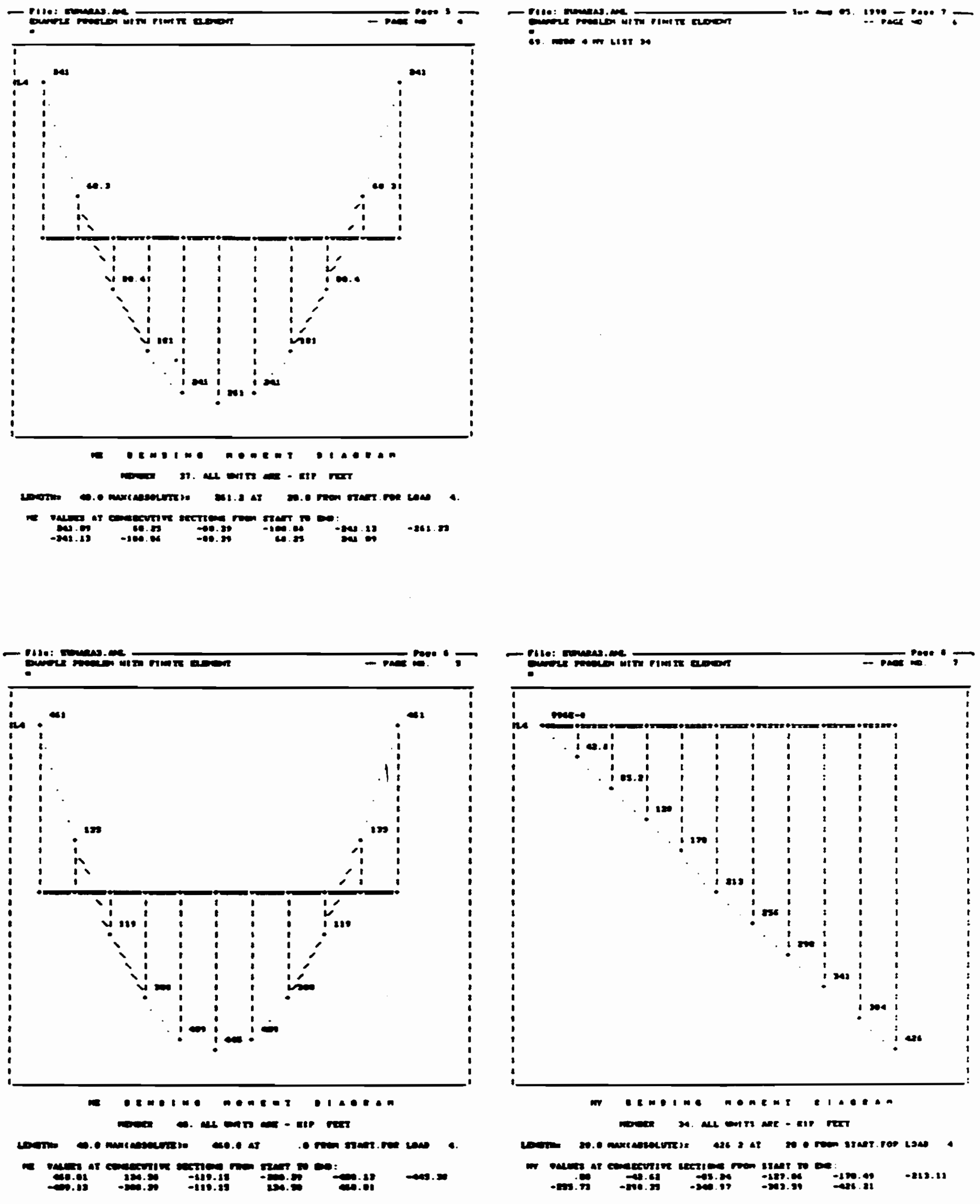


Figure 4. A page from STAAD-VIEW module.

by combining into a single environment the tasks of structural modelling, verification, analysis and design with drawing generation. Hence the high level of integration found in ISDS represents a significant breakthrough in application software for structural engineers and it would be appropriate to say that ISDS is a welcomed partner for the structural engineer.

Chapter 4

GENERAL DESCRIPTION OF ISDS

4.1 THEORETICAL APPROACH

A widely used method for analyzing buildings is the matrix method of structural analysis [Fleming, 1986]. In the matrix analysis of structures by the displacement method, the structure is first idealized into an assembly of discrete structural elements. Each element has an assumed form of displacement and the elements are connected together by discrete joints. The solution is obtained by combining these individual displacements in a manner that satisfies the force equilibrium and displacement compatibility at the joints [ISDS - program user's manual, 1989].

Such a discrete element analysis of structures, having elements which are continuously attached to one another, resolves to element forces at nodal points. This idealized solution has no physical counterpart in the actual structure. Therefore, these forces must be considered as a set of equivalent discrete forces replacing the continuously varying stress field. Displacements of the joints, however, can be interpreted as the actual displacements of the corresponding joints on the structure and linear relationships with element forces are established. All the joints of the structure are included in this exercise, whether they are free to displace or are restrained by supports. Those components of joint displacements that are free to move are called degrees of freedom. The total number of degrees of freedom represent the number of unknowns in the analysis.

One method which is used to solve the unknowns from a series of simultaneous equations and which is particularly suited for structural analysis is the method of decomposition. This method has been selected for use in ISDS. A very accurate and cost effective decomposition method is the Modified Cholesky's method which takes advantage of the symmetric stiffness matrices of linearly elastic structures [ISDS - program user's manual, 1989]. Stiffness matrices are also banded about the main diagonal and the narrower the band the fewer the calculations required. Cholesky's Method takes full advantage of the symmetric banded stiffness matrix during solution. ISDS offers features by which the program can internally rearrange the joint numbers to provide a least bandwidth which gives the most efficient solution.

For a complete analysis of the structure, the necessary matrices are generated on the basis of the following assumptions [ISDS - program user's manual, 1989] :

1. The structure is visualized as an assemblage of beam and plate type elements joined together at their nodes and is assumed to be loaded and reacted by concentrated loads acting at the nodes. The loads may either be forces or moments and may act in any specified direction.

2. A beam is a longitudinal structural member having a constant section along its length. In a three dimensional space frame, the beam will have six degrees of freedom at each node. They always carry axial forces and may also be subjected to shear and bending in two arbitrary perpendicular planes. They may also be subjected to torsion.

3. A plate element is a two-dimensional element having constant thickness. In planform, it can be a triangle or a general quadrilateral.

4. Internal and external loads acting on each node are in equilibrium. If torsional or bending properties are defined for any member, six (i.e., three translational and three rotational) degrees

of freedom are considered at each node in the generation of relevant matrices. If a member is defined as truss member (i.e., carrying only axial forces) then only three degrees (translational) of freedom are considered at each node.

The member end forces considered when different structure types are specified are shown in Figure 5.

4.2 COORDINATE SYSTEMS

Two types of coordinate systems, viz; local and global coordinates are adopted by ISDS program to define the positive values for a structure and its loading pattern.

1. Global coordinates - A global coordinate system is an arbitrary system in space which follows the orthogonal right handed rule. This system is used to define joint locations and to provide joint loading directions (See Figure 6).

2. Local coordinates - A local coordinate system is associated with each member and it also follows the orthogonal right handed rule (See Appendix B).

Figure 7 shows a beam member from joint i on the longitudinal axis of the beam in a positive x-direction from joint i to joint j. The local y and z axes coincide with the axes of the two principal moments of inertia.

Relationship between local and global coordinates - Inputs for joint restraints and loadings can be provided in the global coordinate system whereas output for member end forces is printed in the local coordinate system. Therefore it is important to know the relationship between the local and global coordinate systems (See Figure 8). The relationship is defined by the Beta angle.

When the local x-axis is not parallel to the global Y-axis, the Beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position of having the

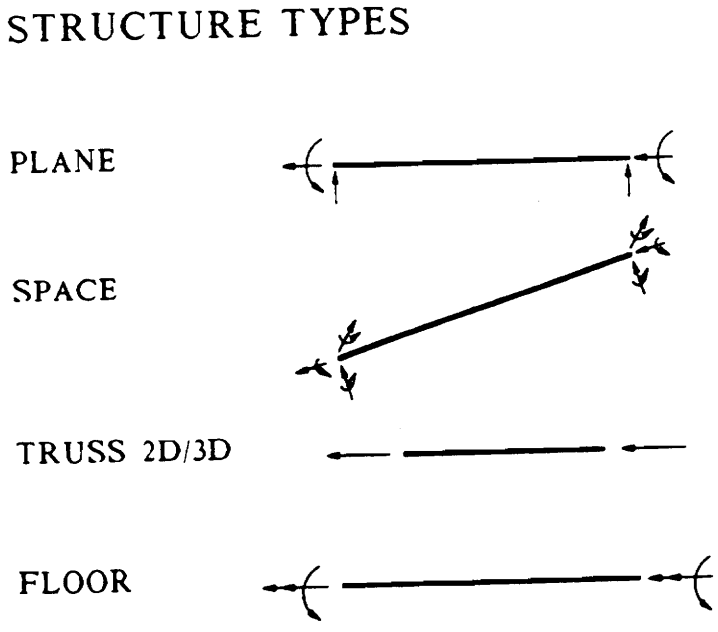


Figure 5. Member end forces for specified structure type.

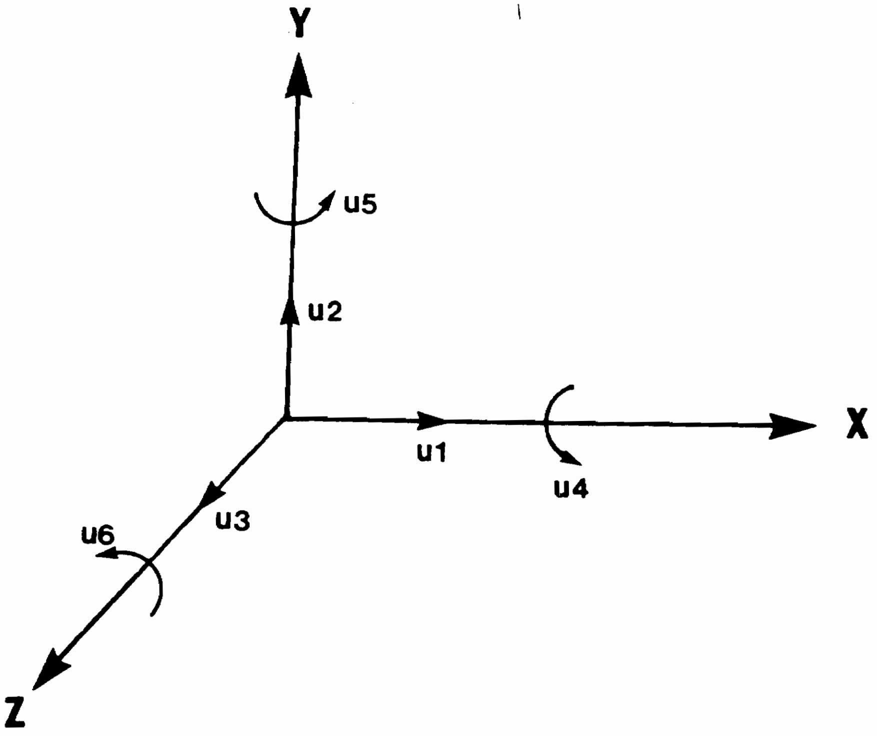


Figure 6. Global coordinate system with directions of diplacement.

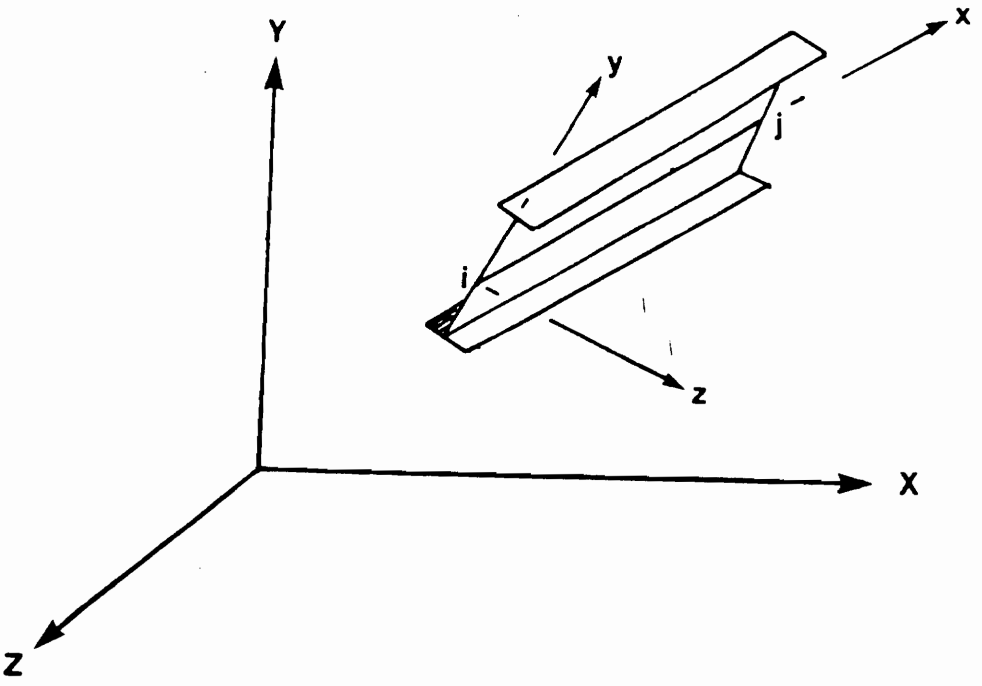


Figure 7. Local coordinates drawn in global frame.

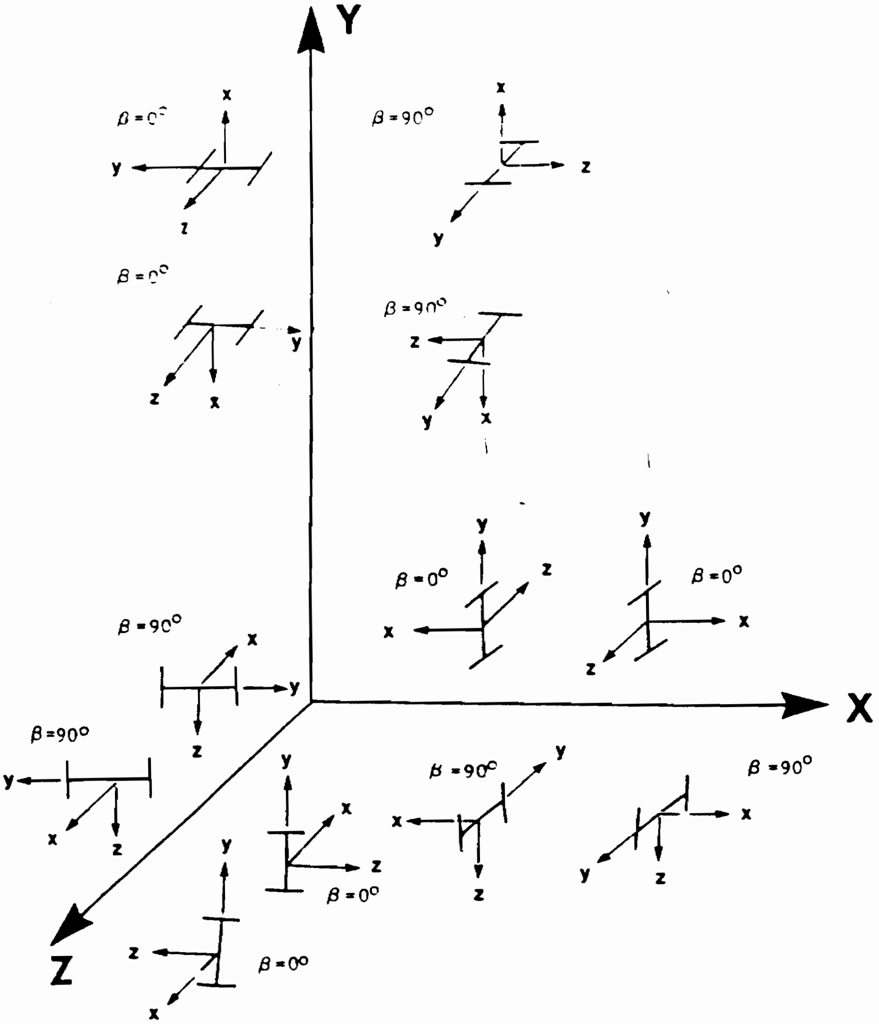


Figure 8. Relationship between global and local axis.

local z-axis parallel to the global XZ plane and so that the local y-axis is in the same positive direction as the global Y-axis [ISDS - program user's manual, 1989].

When the local x-axis is parallel to the global Y-axis, as in the case of a column in a structure, the Beta angle is the angle through which the local z-axis has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis [ISDS - program user's manual, 1989].

Chapter 5

DESCRIPTION OF EXAMPLE PROBLEMS

The objective, as stated earlier, is to evaluate ISDS's ability to analyze and design a multistory reinforced concrete frame subjected to gravity, wind, and earthquake loads and various combinations of the same.

It will be achieved by first considering a simple one-story frame with multiple bays subjected to gravity and wind loads and comparing the results from analyses using ISDS, FAP, and a conventional method. Also, the results of design will be compared using ISDS and a conventional reinforced concrete design method.

Secondly, a multistory frame with multiple bays in both directions subjected to gravity, wind, and earthquake loads and their combinations will be analyzed and their results compared using ISDS, FAP, and a conventional method. Also, the results of design will be compared using ISDS and a conventional reinforced concrete design method.

5.1 FIRST EXAMPLE : ONE STORY FRAME

The plan and elevations of the building are shown in Figures 9 through 11. The base of the first story columns are assumed fixed. The design data are as follows :

1. Gravity service loads
    a. Total average superimposed dead load = 143.75 psf
    b. Total average superimposed live load = 40 psf
2. Wind loads : 20 psf, in direction BC (See Figure 10)
3. Material properties
    a. Compressive strength of concrete = 4000 psi
    b. Yield strength of steel = 60,000 psi
4. Dimensions of structural members
    a. All columns 15 in. x 30 in.
    b. 40 feet span beams 15 in.(wide) x 30 in.(deep).
    c. 15 feet span beams 15 in.(wide) x 8 in.(deep).
    d. All slabs are 7.50 in. thick.
5. Design specifications
    a. Concrete : ACI 318-83
    b. Lateral loads : UBC 1976

5.2 APPROXIMATE ANALYSIS FOR LATERAL LOADS

There are two common approximate methods of analyzing building frames for lateral loads, namely; Portal method and Cantilever method. The portal method is more popular and more commonly used for determining wind forces in building frames because of its simplicity. This method is said to be satisfactory for most buildings up to 25 stories in height [Nawy, 1986]. In the Portal method, the frame is theoretically divided into independent portals and the following three assumptions are made [Nawy, 1986] :

1. Columns bend in such a manner that there is a point of inflection at midheight.
2. The girders bend in such a manner that there is a point of inflection at midspan.

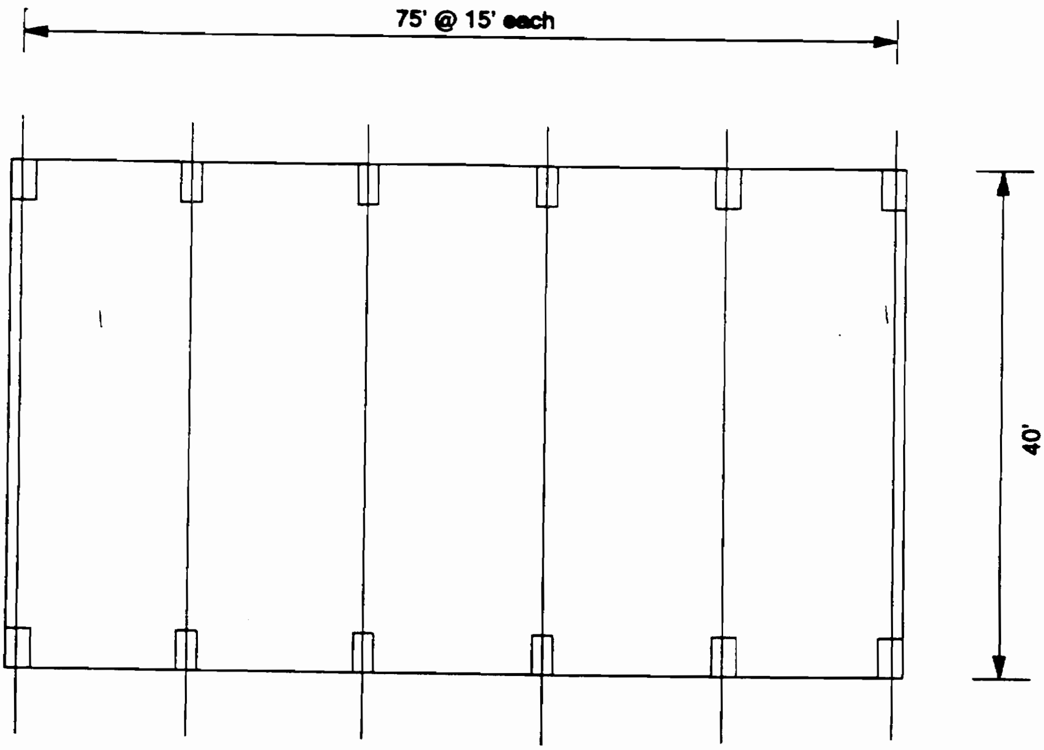


Figure 9. Plan of the building, Example #1.

3. The horizontal shears on each level are arbitrarily distributed between the columns.

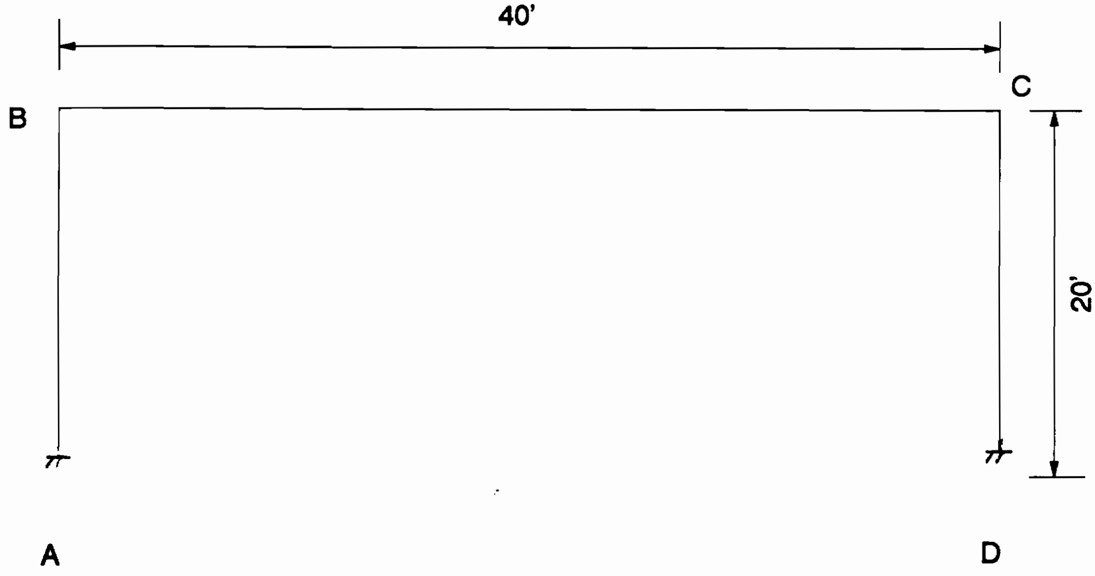


Figure 10. Transverse section of the building, Example #1.

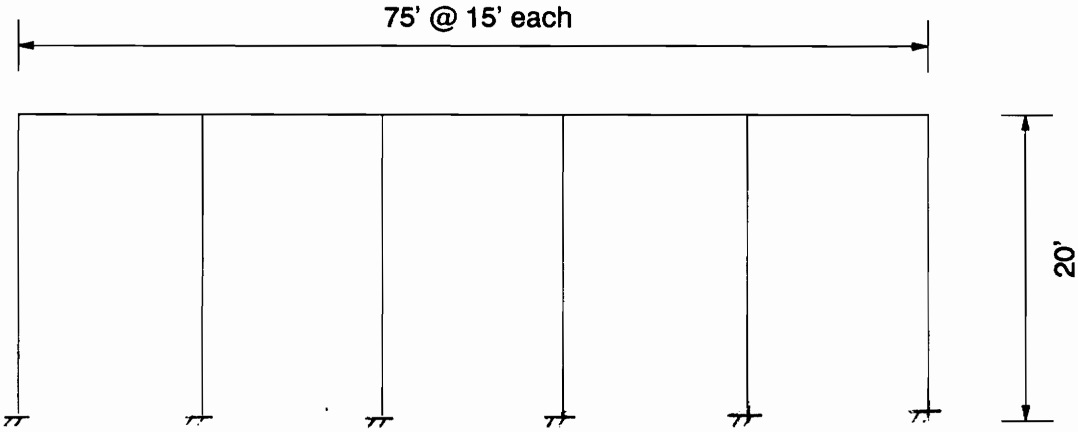


Figure 11. Longitudinal section of the building, Example #1.

One commonly used distribution is to assume the shear divides among the columns in the ratio of one part to exterior columns and two parts to the interior columns.

Briefly the calculations are made as follows :

1. Column shears

   The shears in each column on the various levels are first obtained. Because there are two exterior columns (See Figure 10), the following expression may be written :

   V + V = 0.020(20 x 15) = 6 kips.

   Shear in column AB = 3 + 1.5 = 4.5 kips.

   Shear in column BC = 1.5 kips.

2. Column moments

   The columns are assumed to have points of inflection at their midheight, therefore their moments, top and bottom, equal the column shear times half the column heights. Moment (top of column) in column AB = Moment in column CD = M

   M = 3 x 20/2 = 30 kip-feet.

3. Girder moments and shears

   At any joint in the frame the sum of the moments in the girders equals the sum of the moments in the columns. By beginning at the upper left hand corner of the frame and working across from left to right, adding or subtracting the moments as the case may be, the girder moments are found. The girder shear equals the girder moments divided by half girder lengths.

   Girder moment = 30 kip-feet.

   Girder shear = 30/20 = 1.50 kips

4. Column axial forces

   The axial forces in columns may be directly obtained from the girder shears. The column axial force in AB (See Figure 10) is numerically equal to the shear in the girder BC.

   Axial force in column AB = 1.50 kips.

The results of the analysis by the portal method are compared with more exact analyses in Chapter 6.

5.3 SECOND EXAMPLE : MULTISTORY FRAME

The plan and elevations of the building are shown in Figures 12 through 14. The columns and structural walls have constant cross- section throughout the height of the building. The beams and slabs also have the same dimensions at all floor levels. The base of the first story columns are assumed fixed. The design data are as follows :

1. Gravity service loads
   - a. Total average superimposed dead load = 30 psf
   - b. Total avearge superimposed live load = 75 psf
2. Earthquake loads : UBC 1976 (basis of STAAD-III analysis and design)
   - a. Seismic zone coefficient factor = Z = 1
   - b. Horizontal force factor = 0.80
   - c. Importance factor = 1.50 (essential facilities) [11]
3. Material properties
   - a. Compressive strength of concrete = 4000 psi
   - b. Yield strength of steel = 60,000 psi
4. Dimensions of structural members
   - a. All columns are 15 in. x 30 in.
   - b. All beams in transverse direction are 15 in.(wide) x 30 in.(deep).
   - c. All beams in longitudinal direction are 15 in.(wide) x 33 in.(deep).
   - d. The shear wall has a constant thickness of 12 in.
   - e. All slabs have a thickness of 7.5 in.
5. Design specifications
   - a. Concrete : ACI 318-83
   - b. Lateral loads : UBC 1976

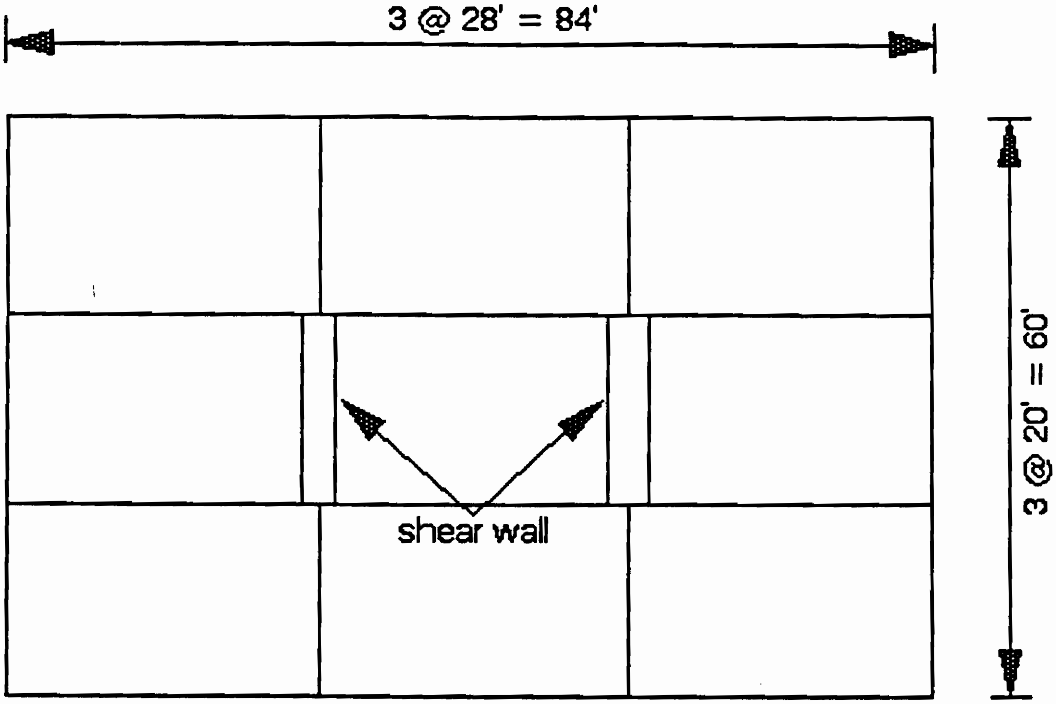


Figure 12. Plan of the building, Example #2.

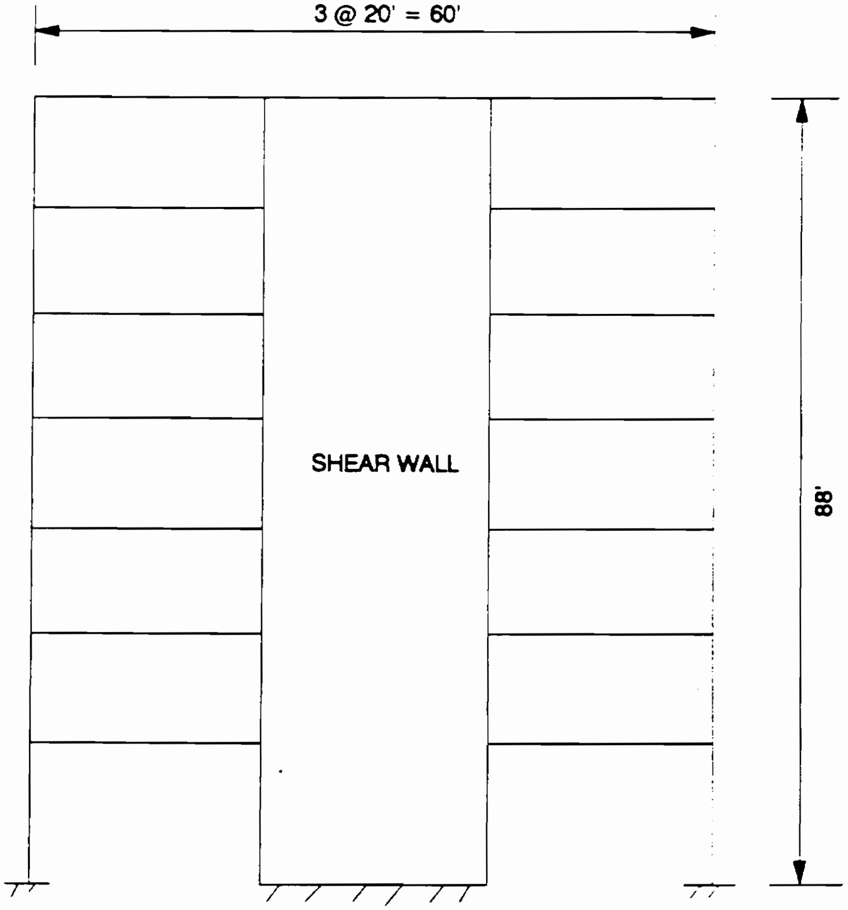


Figure 13. Transverse section of the building, Example #2.

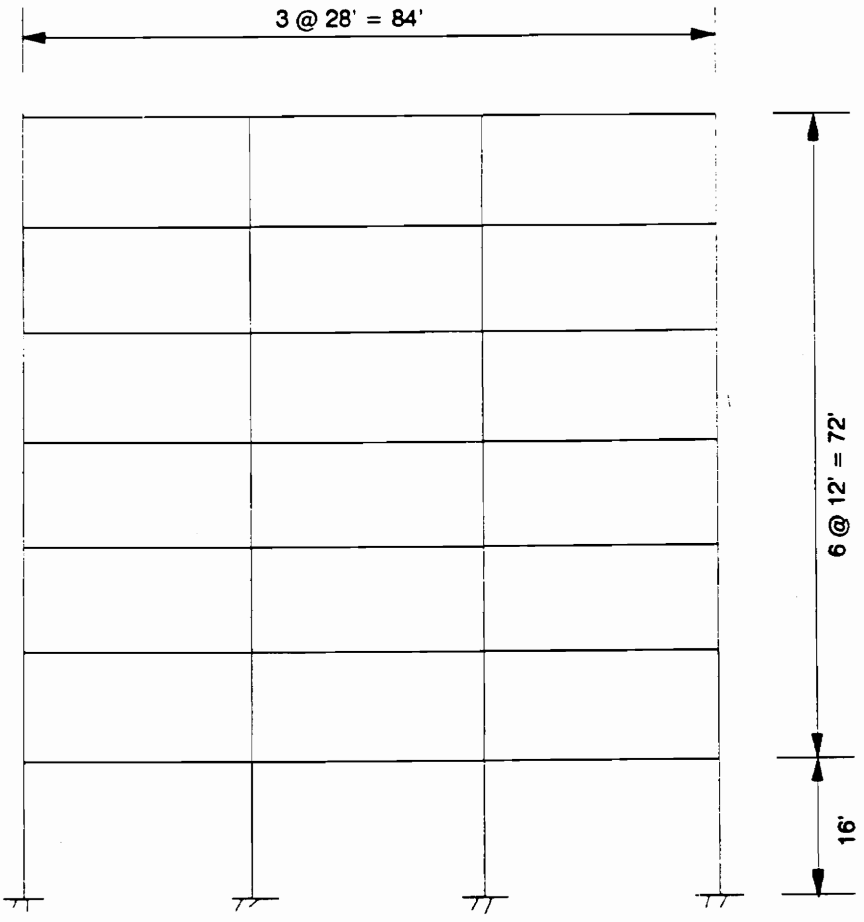


Figure 14. Longitudinal section of the building, Example #2.

Chapter 6

DISCUSSION OF RESULTS

The first section of this chapter discusses some general observations made while working on this project. The subsequent sections discuss comparison of different analyses and design methods for example #1 and #2 problems described in Chapter 5.

6.1 OBSERVATIONS MADE DURING THE STUDY

6.1.1 EFFECT OF CONSIDERING PLATE ELEMENTS

The main purpose of this exercise was to observe how the frame was affected by the placing of horizontal plates (slabs) between columns and beams. The fixed base space frame is shown in Figure 15.

First, only the frame, without the plates was analyzed and the shear and moments were determined from analysis. Secondly, ten plates were added which stiffened the frame. The thickness of the plates was 7.50 inches. A uniformly distributed load was applied on the plates in the negative global Y-direction. Shear and moments were determined for this case. Table 1 shows the moments at nodes 19, 13 and 76 which are nodes of an external bay of the space frame. It also shows moments at nodes 20, 14, 77 and 21, 15, 78 which are nodes of the first and second interior bays of the space frame respectively.

It can be seen from Table 1 that, when the plates are not included, the middle section of the frame gets a larger amount of moment than the external bays. But the presence of plates stiffens the frame and causes the frame to move uniformly and therefore moments at exterior and interior bays get closer to one another.

Again as seen from the table, the presence of plates does not mean an obvious decrease in moments. The inclusion of plates will only cause a more uniform distribution of moments among members.

6.1.2 EFFECT OF PLATES ON TORSION IN SPANDREL BEAMS

The loading and the boundary conditions are as explained in Section 6.1.1 The values of torsion in the spandrel beams are presented in Table 2.

It can be seen from the table that, when the plates are not included, the exterior spandrel beam gets more torsion than when the plates are included. When the plates are included there is a more uniform distribution of torsion among the spandrel beams, i.e. the values of torsion in the exterior and interior spandrel beams come closer to one another. The central beam (45) has no torsion.

6.1.3 P-DELTA ANALYSIS

To study the effect of P-Delta analysis (as compared to ordinary linear analysis), a number of computer runs were made on a portal frame (See Figure 16) using STAAD. During the trial, the width of the bay was kept constant, but the heights of the columns were gradually increased. Axial loads of 1000 kips acting in negative global y- direction were placed at joints 2 and 3 as shown in Figure 16. An axial load of 25 kips acts in global x - direction at joint 2. Table 3 indicates values of moments in members 1 and 2 of the frame for three different heights of the frame.

It is seen from the table that there is a considerable difference in values when the analysis is performed by the two methods mentioned. The P-Delta analysis method accommodates the require-

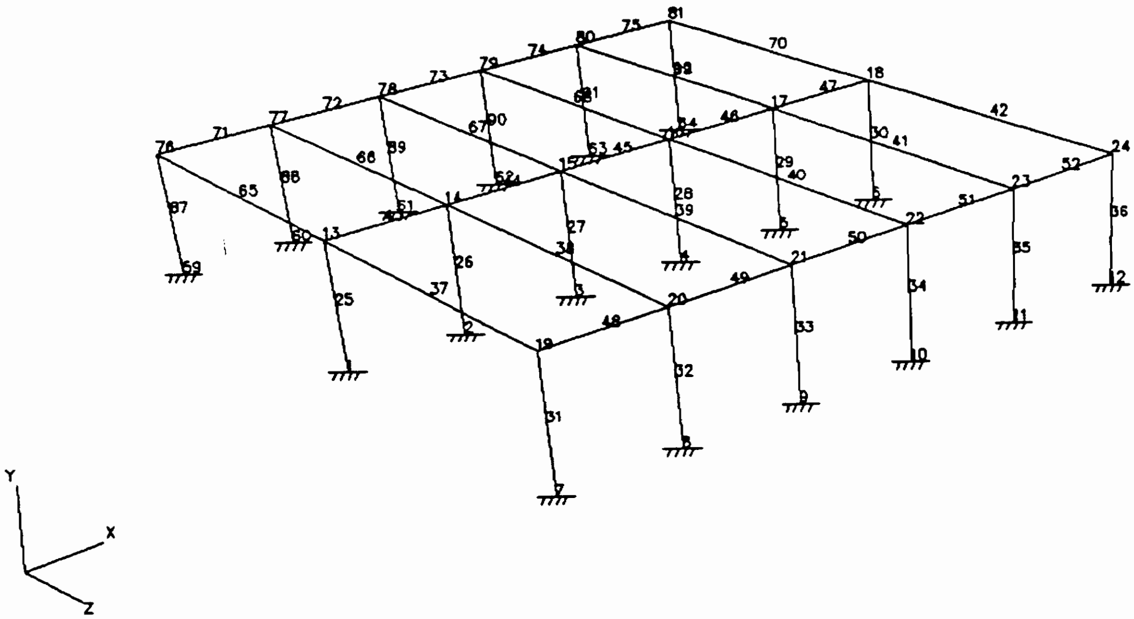


Figure 15. Space frame used to study effect of plates on frames.

Table 1. Effect of plates on moments (kip-ft) in a frame.

NODE	PLATES INCLUDED	PLATES NOT INCLUDED
19 .EXT. FRAME	218.55	221.17
13 EXT. FRAME	391.47	389.7
16 EXT. FRAME	219.02	221.17
20 1st INT. FRAME	425.68	400.88
14 1st INT. FRAME	690.6	703
77 1st INT. FRAME	425.16	400.88
21 2nd INT. FRAME	417.16	400.36
15 2nd INT. FRAME	694.67	703.26
78 2nd INT. FRAME	417.54	400.36

Table 2. Effect of plates on torsion (kip-ft) in spandrel beams.

MEMBER	PLATES NOT INCLUDED	PLATES INCLUDED
EXTERNAL SPANDREL BEAM	2.23	1.37
1st INTERNAL SPANDREL BEAM	0.02	0.29
CENTRAL SPANDREL BEAM	0	0
1st INTERNAL SPANDREL BEAM	0.02	0.29
EXTERNAL SPANDREL BEAM	2.23	1.37

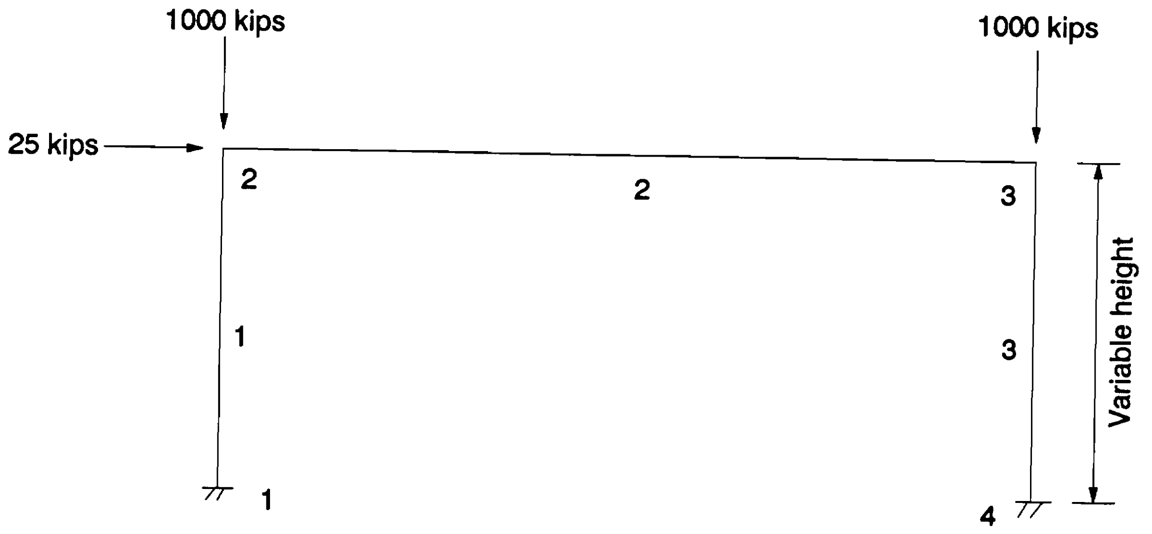


Figure 16. Frame used for study of P-Delta effect.

Table 3. P-Delta effect, values of moments (kip-ft).

			HEIGHT OF COLUMN		
ANALYSIS	MEMBER	END	20 FT	35 FT	50 FT
LINEAR ANALYSIS	1	END 1	161	337	509
		END 2	63	117	295
	2	END 1	63	117	295
		END 2	380	505	639
SECOND ORDER OR P-DELTA ANALYSIS	1	END 1	179	413	707
		END 2	51	173	451
	2	END 1	51	173	451
		END 2	392	561	795

ments as specified in Section 10.10 of ACI 318-83 code. Therefore it is the most appropriate for the design of concrete members.

6.1.4 EFFECT OF JOINT SIZE

Consider the portal frame shown in Figure 10 (Chapter 5). Beam BC was subjected to a uniformly distributed load of 4.5214 kips per feet (factored). Analysis of this frame resulted in a negative bending moment of 450.82 kip feet at ends of the beam. But the moment at the face of the column AB will be a smaller value since the column has a depth in the direction of the beam. One could design the beam for this lesser value of moment. In fact one can find the values of bending moments and shear forces at any section in the beam or column by specifying a joint at that point and providing the member incidences.

Therefore a joint was specified on the beam BC at a distance of 1.25 feet from joint B. The joint, the member numbers and the bending moment diagram for the new frame are shown in Figures 17 and 18. The moment of inertia of members 10, 11, 13, and 14 were made approximately 15 times greater than moment of inertia of other members in the frame. From analysis it was seen that the moment at the face of the column was 341.23 kip feet. It can also be verified as follows :

Moment at face of column

$$= 450.82 - 90.43 \times 1.25 + 4.5214 \times 1.25 \times 1.25/2 = 341.31 \text{ kip feet}$$

It can also be verified by solving the parabola :

Moment at face of column

$$= 450.82 - 4 \times (453.56 + 450.82) \times 1.25 \times (40 - 1.25)/2$$

$$= 341.31 \text{ kip feet.}$$

6.1.5 ANALYSIS OF SLAB BY FINITE ELEMENTS USING STAAD

The space frame as shown in Figure 19 was considered. It is made up of 4 columns and beams which support a two-way slab that is discontinuous on all its edges. Column bases are assumed to be fixed. The columns are 20 feet high and the slab (element) is 21 feet 'square' in plan. The

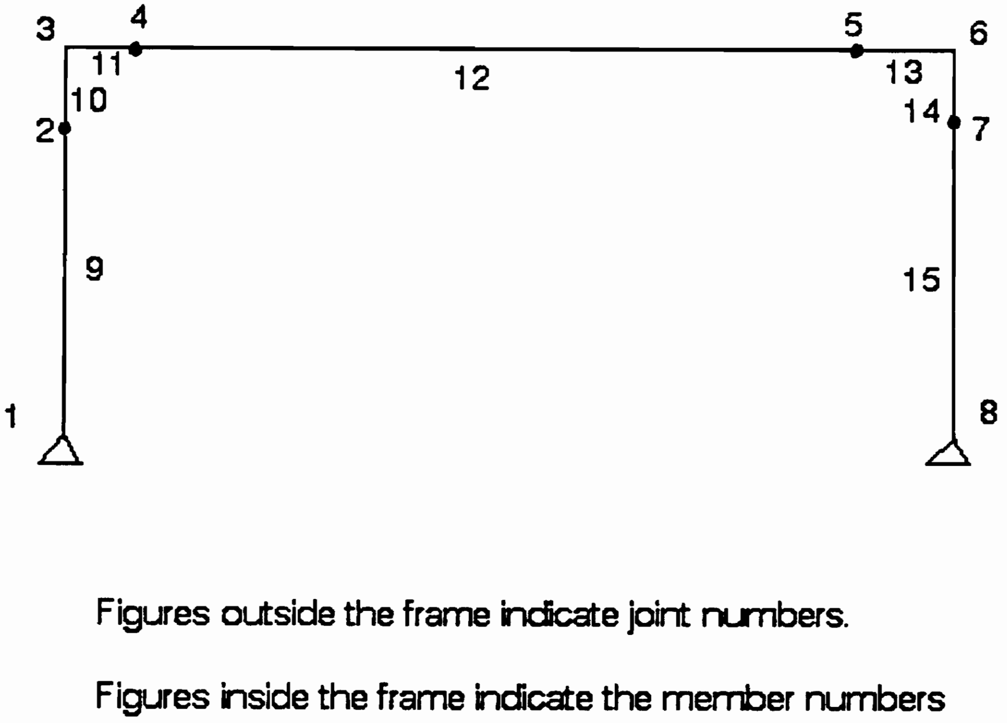


Figure 17. Effect of joint size.

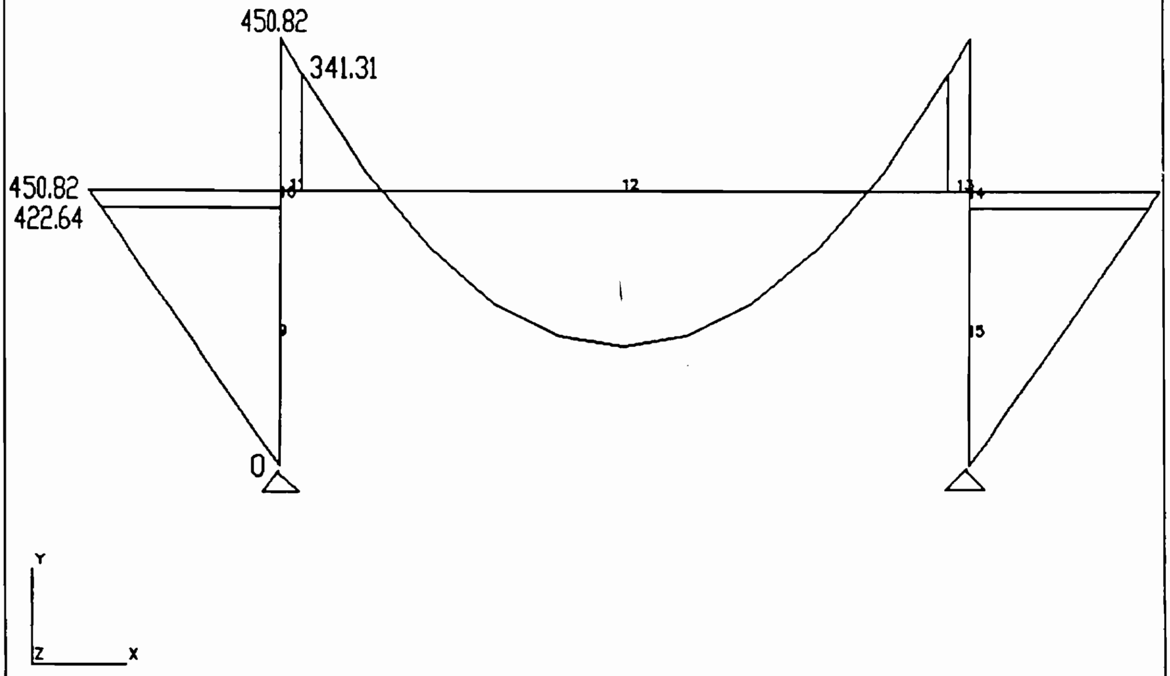


Figure 18. Bending moment diagram for new frame.

columns and the beams have a 30in. X 30in. cross-section. The slab is subjected to a uniformly distributed load of 1.0 kip per square feet, acting in the negative global Y-direction.

In the first trial, the slab was divided into 9 finite elements, 3 in each direction. In the second trial, the mesh was refined a bit further by providing a total of 25 finite elements. In the third trial the slab was divided into 7 finite elements in each direction for a total of 49 finite elements. The results of the analyses using the above 3 cases are presented in Table 4.

It is evident from the table that as the refinement of the mesh increased, the value of the 'positive moment' at center of slab started increasing and approached the value of 19.50 kip-feet - value calculated using conventional reinforced concrete design method. This method was first introduced in the 1963 ACI code, and is the simplest for designing two-way slabs supported on beams. This method is still considered useful in estimating what should be done with all those slabs not really large enough to justify a slab system approach. This method uses coefficients (values of coefficients depend upon the ratio of short span to long span). According to this method [Ferguson, 1979], the positive moment at midspan :

M_{pos} = coefficient x w x $(l_x)^2$ where,
w = Uniformly distributed load on the slab.
l_x = Length (c/c) of short span of the slab = 19.75 feet.
Coefficient = 0.05 [Ferguson, 1979]
Therefore, moment = 0.05 x 1 x 19.75^2 = 19.50 kip-feet.

(Timoshenko gives a coefficient of 0.0479 for the case of uniformly distributed load on a simply supported slab).

A few more refinements of the mesh would therefore mean a closer value.

6.2 COMPARISON OF ANALYSES

The pinned base space frame as shown in Figure 20 is chosen as a simple example for comparison of different analyses. First, the space frame was subjected to wind loading in the negative global Z - direction. The wind load of 20 psf was assumed to act on only one longitudinal face of the

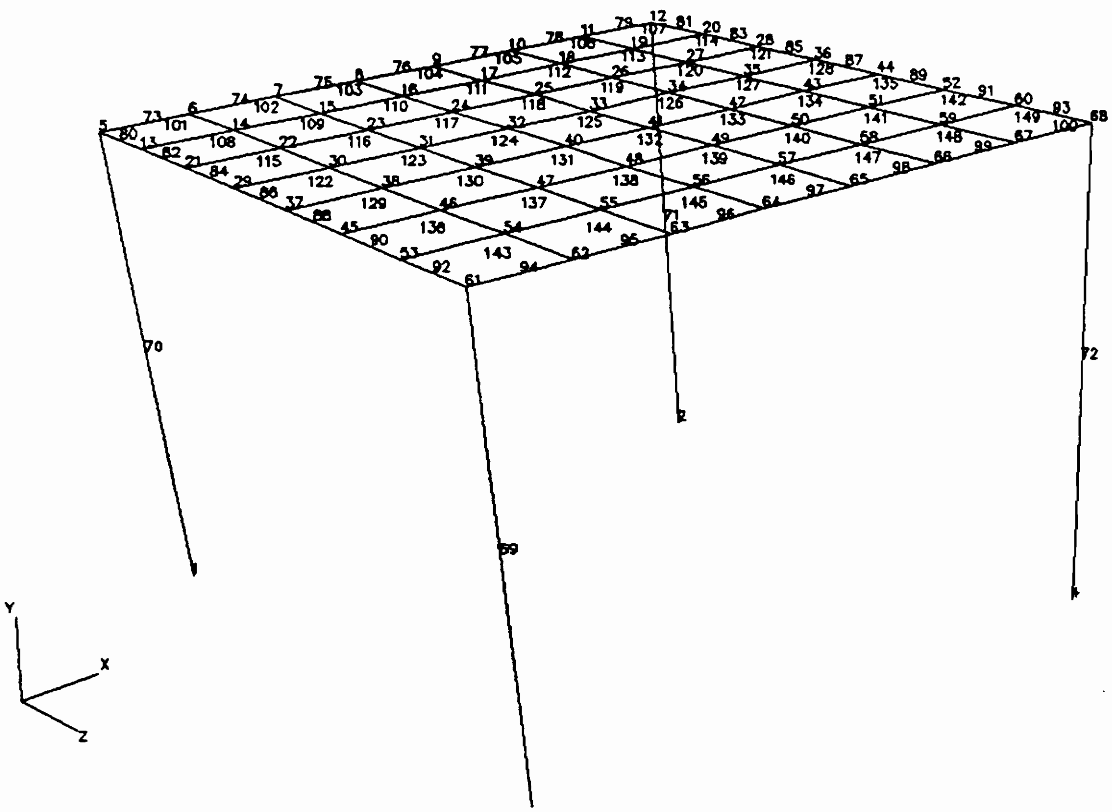


Figure 19. Analysis of slab by dividing it into finite elements.

Table 4. Positive moment values (kip-ft) at center of slab.

NO. OF ELEMENTS IN EACH DIRECTION	POS. MOMENT AT CENTER OF SLAB	COMPUTED VALUE FERGUSON, 1986
3	15.62	19.5
5	16.97	
7	18.33	

frame. This frame was then analyzed using ISDS, FAP and the portal method of hand calculations. The results are tabulated in Table 5. The percentage difference in results from ISDS and FAP are negligible, the maximum percentage difference being 0.20 %. The values of moments obtained from the portal method of hand calculation agree reasonably well with those obtained from ISDS and FAP, with the maximum percentage difference being 30%. The space frame analyses of ISDS and FAP indicate larger moments for the exterior portals and smaller moments for the interior portals than those obtained from the simplified two-dimensional portal method.

Secondly, the space frame was subjected to a gravity load in the negative Y-direction. When results from hand calculations (using the moment distribution method) are compared in Table 6 with those of ISDS and FAP, a maximum percentage difference of 6.80% was found. This difference is not unexpected since the moment distribution method neglects the axial load effect.

The values of shear in beams and axial force in columns, due to wind loading shown in Tables 7 and 8, compared very well between STAAD and FAP analyses. There was no difference in the results. But there was considerable difference in values of column shear when compared with the portal method values. This was because of the assumptions in portal method of analysis. This may not be the case always.

6.3 COMPARISON OF MEMBER DESIGN

For the purpose of comparison of design between ISDS and ordinary reinforced concrete design method, the pinned base space frame as shown in Figure 20 was chosen. The five horizontal plates between the column and the beams were subjected to a uniform pressure intensity of 0.26925 ksf (kips per square feet) acting in the negative global Y-direction.

6.3.1 DESIGN OF BEAM (#40)

1. DESIGN OF BEAM FOR NEGATIVE MOMENT

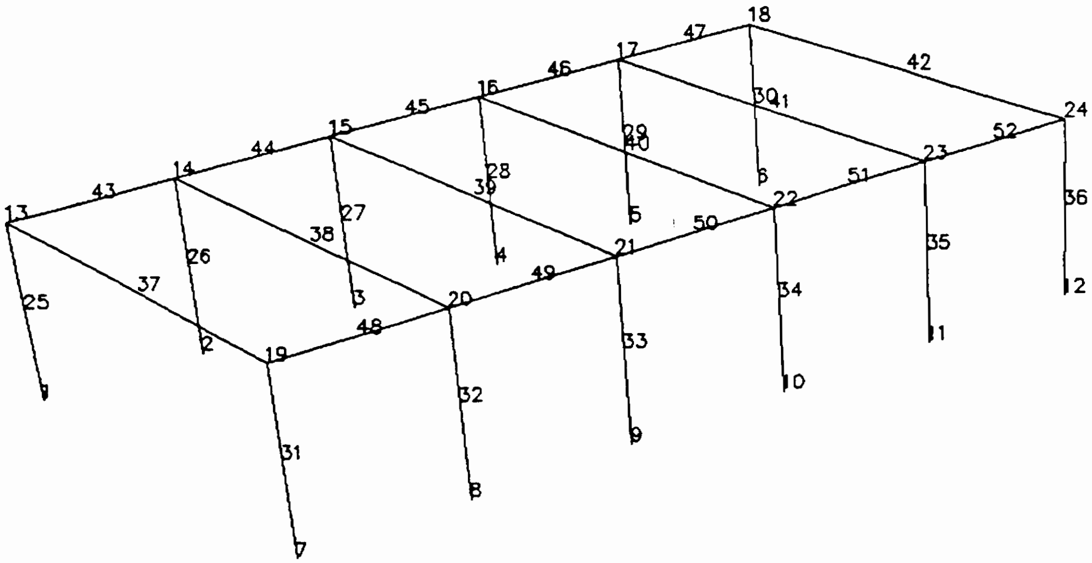


Figure 20. Space frame used for comparison of analyses.

Table 5. Moments (kip-ft) for wind loading.

MEMBER	END	STAAD	FAP	PORTAL METHOD
40 Int. portal beam	END 1	26.96	26.96	30
	END 2	30.64	30.64	30
37 Ext. portal beam	END 1	19.44	19.44	15
	END 2	21.29	21.3	15
34 Int. portal column	END 1	0	0	0
	END 2	26.99	26.99	30
31 Ext. portal column	END 1	0	0	0
	END 2	19.4	19.41	15

Table 6. Moments (kip-ft) for gravity loading.

MEMBER	END	STAAD	FAP	MOMENT DISTRIBUTION
40 Int. portal beam	END 1	249.37	249.86	266.85
	END 2	249.37	249.86	266.85
37 Ext. portal beam	END 1	249.37	249.86	266.85
	END 2	0	0	0
34 Int. portal column	END 1	249.36	249.86	266.85
	END 2	0	0	0
31 Ext. portal column	END 1	249.37	249.86	266.85
	END 2	0	0	0

Table 7. Shear (kips) for wind loading.

MEMBER	END	STAAD	FAP	PORTAL METHOD
40 INT. PORTAL BEAM	END 1	1.44	1.44	1.5
	END 2	1.44	1.44	1.5
37 EXT. PORTAL BEAM	END 1	1.02	1.02	0.75
	END 2	1.02	1.02	0.75
34 INT. PORTAL COLUMN	END 1	4.35	4.35	4.5
	END 2	1.65	1.65	1.5
31 EXT. PORTAL COLUMN	END 1	2.47	2.47	2.25
	END 2	0.43	0.43	0.75

Table 8. Axial force (kips) in column for wind loading.

MEMBER	END	STAAD	FAP	PORTAL METHOD
34 Int. portal column	END 1	1.44	1.44	1.5
	END 2	1.44	1.44	1.5
31 Ext. portal column	END 1	1.02	1.02	0.75
	END 2	1.02	1.02	0.75

Beam is designed as a rectangular beam ignoring the presence of flange, since the flange is present on the tension side at the ends of the beam.

K_n = 1041 psi [Ferguson, 1979]

compressive strength of concrete = 4000 psi

yield strength of steel = 60000 psi.

M_d = 255.02 kip-feet.

M_l = 60.93 kip-feet.

$M_u = 1.4\,M_d + 1.7\,M_l$ = 460.61 kip-ft

$M_n = M_u/\Phi$ = 460.61/0.9 = 511.8 kip-feet.

$M_n = K_n b d^2$

$511.8\text{x}12000 = 1041\text{x}15\text{x}d^2$

Therefore required d = 19.83 inches.

Assuming #8 bars

$d_{(available)}$ = 30 - 0.50 - 1.50 - 1.0/2 = 27.5 inches.

Clear cover to bottom bar = 1.50 inches

Diameter of bar = 1.0 inch.

a/d = 0.38 [Ferguson, 1979]

Therefore a = 0.38 x 27.5 = 10.45 inches.

z = d - a/2 [Ferguson, 1979]

z = 27.5 - 0.50 x 10.45 = 22.27 inches

$M_n = A_s f_y z$

Therefore, trial A_s = 511.8 x 12/(60 x 22.27) = 4.59 sq in.

$a = A_s f_y/(0.85 f_c b)$

= 4.59 x 60 / (0.85 x 4 x 15) = 5.39 inches.

New z = 27.5 - 0.50 x 5.39 = 24.80 inches.

A_s = 511.80 x 12/(60 x 24.80) = 4.12 sq in.

Iterating, we get A_s = 4.08 sq.in.

Provide 6 #8 bars (A_s = 4.72 sq. in.), which is the same as provided by ISDS.

2. DESIGN OF BEAM FOR POSITIVE MOMENT

Beam is designed as a T-beam to consider the effect of flange, since the flange is present on the compression side of the beam when designing the beam for positive moment [McCormac, 1988].

Effective flange width is the smallest of :

1. Span/4 = 40 x 12 / 4 = 120 inches. (governs)
2. 8 x thickness of slab + overhang = 8 x 7.5 x 2 + 15 = 135 inches.
3. Beam spacing = 15 x 12 = 180 inches.

M_d = 238.70 kip-feet.

M_l = 65.52 kip-feet.

M_u = 1.4 M_d + 1.7 M_l = 445.57 kip-ft

M_n = 445.57 / 0.9 = 495 kip feet.

Assume #9 bars

d_e = 30 - 0.50 - 1.50 - 1.128/2 = 27.44 inches.

Approximate lever arm is the larger of :

1. d = 0.9 x 27.44 = 24.96 inches (governs)
2. $d - H_f/2$ = 27.44 - 7.5/2 = 23.69 inches.

Trial A_s = 495.08 x 12 / (24.96 x 60) = 3.96 inches.

It was found from calculations that the neutral axis is in the flange.

Therefore the design is done as for a rectangular beam 120 inches wide.

$a = A_s f_y / (0.85 f_c b)$

= 3.96 x 60 / (0.85 x 4 x 120) = 0.58 inches.

$z = 27.44 - 0.5 \times 0.58 = 27.15$ inches

$A_s = 495.08 \times 12 / (60 \times 27.15) = 3.65$ sq in.

New $a = 0.58 \times 3.65/3.96 = 0.53$ inches

New $z = 27.17$ inches

$A_s = 3.64$ sq in.

Number of #9 bars = 3.64/1 = 3.64

Therefore provide 4 #9 bars (A_s = 4 sq.in.), which is the same as provided by ISDS.

3. CALCULATION OF SHEAR REINFORCEMENT

For members subjected to shear and flexure only

Shear taken by concrete $= 2 \times (f_c)^{0.5} \times b \times d$

$= 2 \times 4000^{0.5} \times 15 \times 27.44$ [Ferguson, 1979]

$= 52.06$ kips.

At a distance of 'effective depth of beam' from support,

$V_u = 80.06$ kips

$V_n = 80.06/0.85 = 94.20$ kips. $> V_c$

$V_s = 94.2 - 52.06 = 42.14$ kips.

$< 4 f_c b_w d$

$V_s = A_v f_y d / s$

Assume #4 bar for stirrup

Spacing of stirrup, $s = 2 \times 0.2 \times 60 \times 27.44/42.14 = 15.63$ inches c/c.

Maximum spacing $= d/2 = 27.44/2 = 13.72$ inches.

Therefore provide #4 stirrups at 13.7 inches c/c.

ISDS provides #4 stirrups at 13.8 inches c/c.

To provide minimum stirrup reinforcement up to a place where :

$$V_n > 0.5V_c$$

i.e. up to 14.15 feet (14 feet provided by ISDS). See Figure 21.

4. DEVELOPMENT OF POSITIVE MOMENT REINFORCEMENT

The points of inflection for the beam under consideration were determined from statics to be 6 feet from the supports (See Figure 21).

At point of inflection, V_u = 90.62 - 4.5312 x 6 = 63.43 kips

For 4 #9 bars, M_n = 553.6 kip-feet.

$$L_d <= M_n/V_u + L_a \text{ [Ferguson, 1979]}$$

L_a is the larger of :

1. 12 d_b = 12 x 1.128 = 13.54 inches.
2. d = 27.44 inches (governs)

$$L_d = 553.6 \text{ x } 12/63.43 + 27.44 = 132.17 \text{ inches.}$$

For #9 bars L_d= 38 inches [Portland Cement Association, 1983] < 132.67 inches (OK). One third of positive moment reinforcement must extend along the same face of the member into the support for at least 6 inches [American Concrete Institute, 1983]. Cut 2 #9 bars at 1 feet from face of support (See Figure 22).

5. DEVELOPMENT OF NEGATIVE MOMENT REINFORCEMENT

At least one-third the total tension reinforcement provided for negative moment at a support shall have an embedment length beyond the point of inflection not less than effective depth of member, 12 x diameter of bar or one-sixteenth the clear span, whichever is greater [American concrete Institute, 1983].

1. d_e = 27.44 inches.
2. 12d_b = 12 x 1.128 = 13.53 inches.
3. 1/16 of span = 40 x 12/16 = 30 inches (governs).

Extend the 6 #8 bars to a distance of 6 feet from the face of the support.

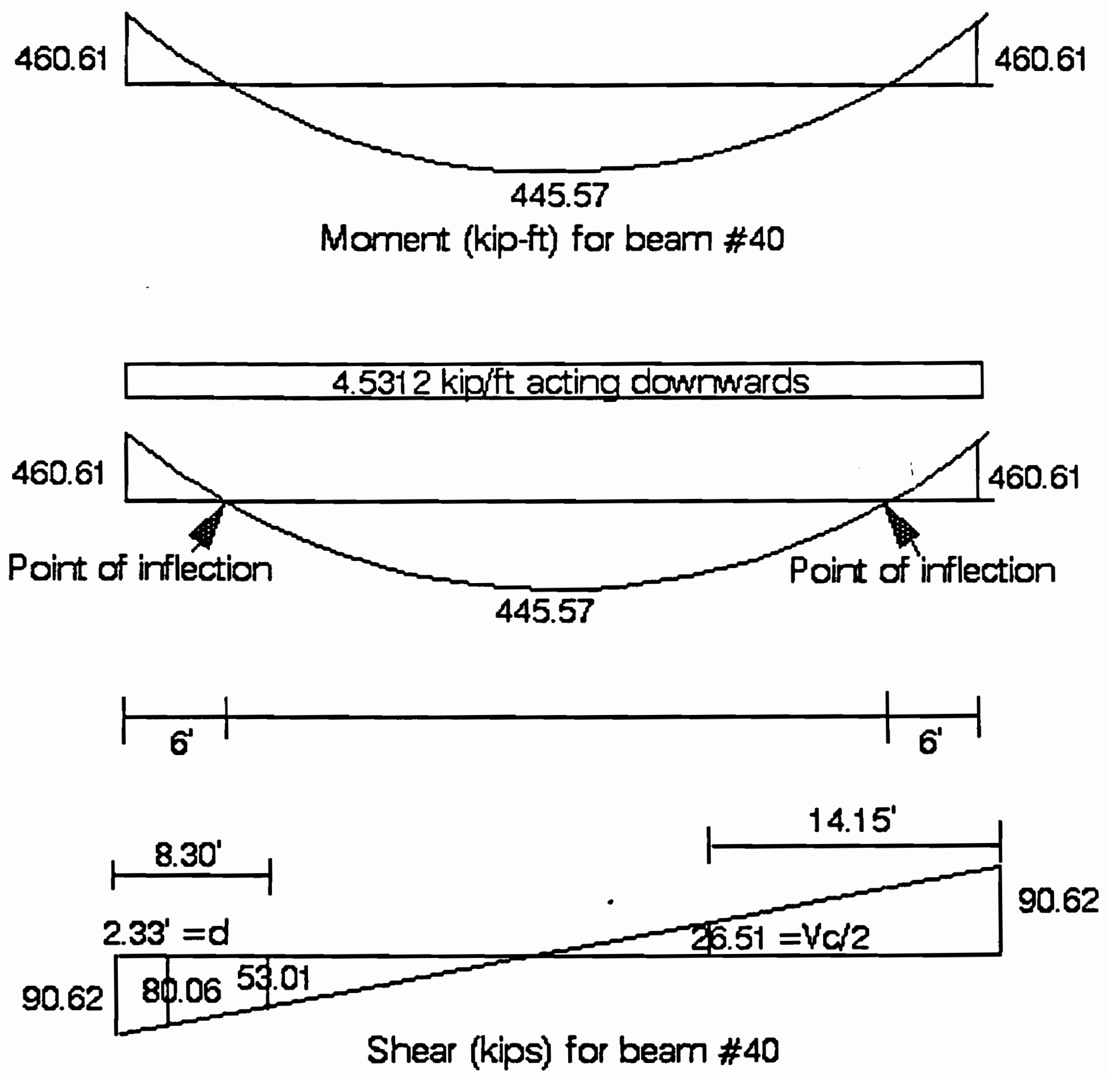


Figure 21. Beam #40

Also provide a 90 degree hook with side cover of 2.5 inches and end cover of > = 2 inches requires a total embedment length L_{dh} of 14 inches [Portland Cement Association, 1983] for a #8 bar. Overall depth of column required would be 16 inches whereas available depth is 30 inches (See Figure 22).

6.3.2 DESIGN OF COLUMN [Ferguson, 1979]

To design a column for an axial load of 105.54 kips and moment of 426.79 kip-feet (factored), for a given compressive strength of concrete = 4000 psi, and yield strength of steel = 60000 psi.

$$M_u/(A_g h) = 426.79 \text{ x } 12 \ / \ (15 \text{ x } 30 \text{ x } 30) = 0.38$$

$$P_u/A_g = 105.54/(15 \text{ x } 30) = 0.2345$$

Referring to "load moment strength interaction diagram for R4-60.75 columns" we get $\rho_g = 0.017$ [SP 86,1984].

$$A_s = \rho_g A_g = 0.017 \text{ x } 15 \text{ x } 30 = 7.65 \text{ sq in.}$$

Provide 24 #5 bars (equal number of bars on all sides - same provided by ISDS).

6.4 COMPARISON OF FUNDAMENTAL PERIODS

The purpose of this exercise was to find out how floor diaphragm action in a structure affects it's fundamental period of vibration. When floor diaphragms were modeled in Example #2, an error message was displayed indicating that the device (PC) did not have enough memory for calculation. Therefore, a smaller structure with one-bay in one direction and two-bays in the other, as shown in Figure 23 was chosen to compare the periods calculated using UBC 1976 and the ISDS. Plan dimensions of the structure are 60 feet x 40 feet. The structure is 144 feet tall. The cross-section of all beams and columns are 15 in x 30 in. and all slabs are 9 in thick. The 12 story structure was subjected to earthquake loading in global X and Z directions.

Five different structural configurations were investigated. First, the structure without floor slabs was analyzed. Secondly, all joints in a single floor level were slaved together, i.e., the relative dis-

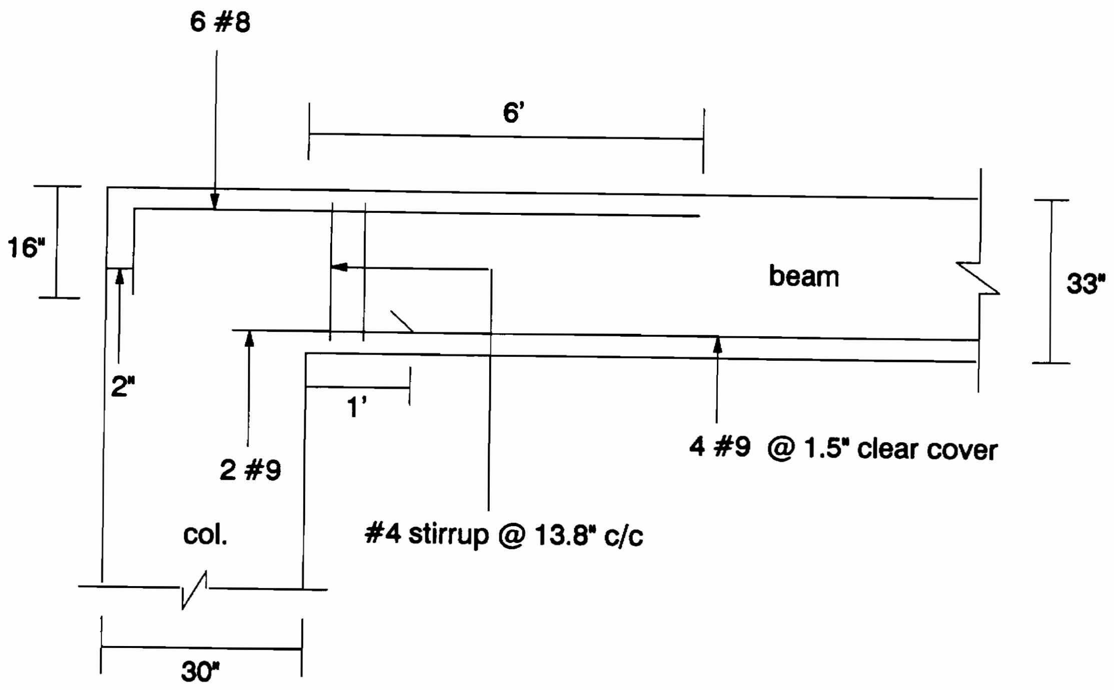


Figure 22. Reinforcement detail at joint of beam #40.

placements of all joints in a floor were made the same. Thirdly, plate elements were included but joints were not slaved together. Fourthly, plate elements were included and a second set of joints was provided on each column just below the slab level and these joints were slaved together. Fifthly, two sets of joints were provided on each column one each below and above the slab level and these joints were slaved together. The results of the analyses are presented in Table 9.

Calculation of the fundamental period of the structure using UBC 1976.

Fundamental period of structure in transverse direction :

$T = 0.05h_n/(D)^{0.5}$[Portland Cement Association, 1983]

$= 0.05 \text{x} 144/(40)^{0.5}$

$=$ 1.138 seconds.

Fundamental period of structure in longitudinal direction :

$0.03 \text{x} 144^{0.75}$ [Portland Cement Association, 1983]

$=$ 1.247 seconds.

It is seen from the table that the values of fundamental periods obtained from empirical formulae differ considerably from those obtained from computer run. It is seen from the table that when plate elements are considered, the period increases considerably because of the increase in mass. It is also observed that when floor diaphragm action is provided, i.e., all joints are made to move the same distance (relatively) in all directions, the value of the period of the structure comes down since stiffness of the structure increases. The calculation of the period using empirical formulae does not include the effect of mass and stiffness (unless where a shear wall is provided) and hence the difference in results.

What the writer feels from this study is that, although the empirical expressions to estimate period that are available in seismic design codes like UBC may be used for preliminary design, once a preliminary design is available, the structure's period should be evaluated from its stiffness and mass characteristics for more accurate values.

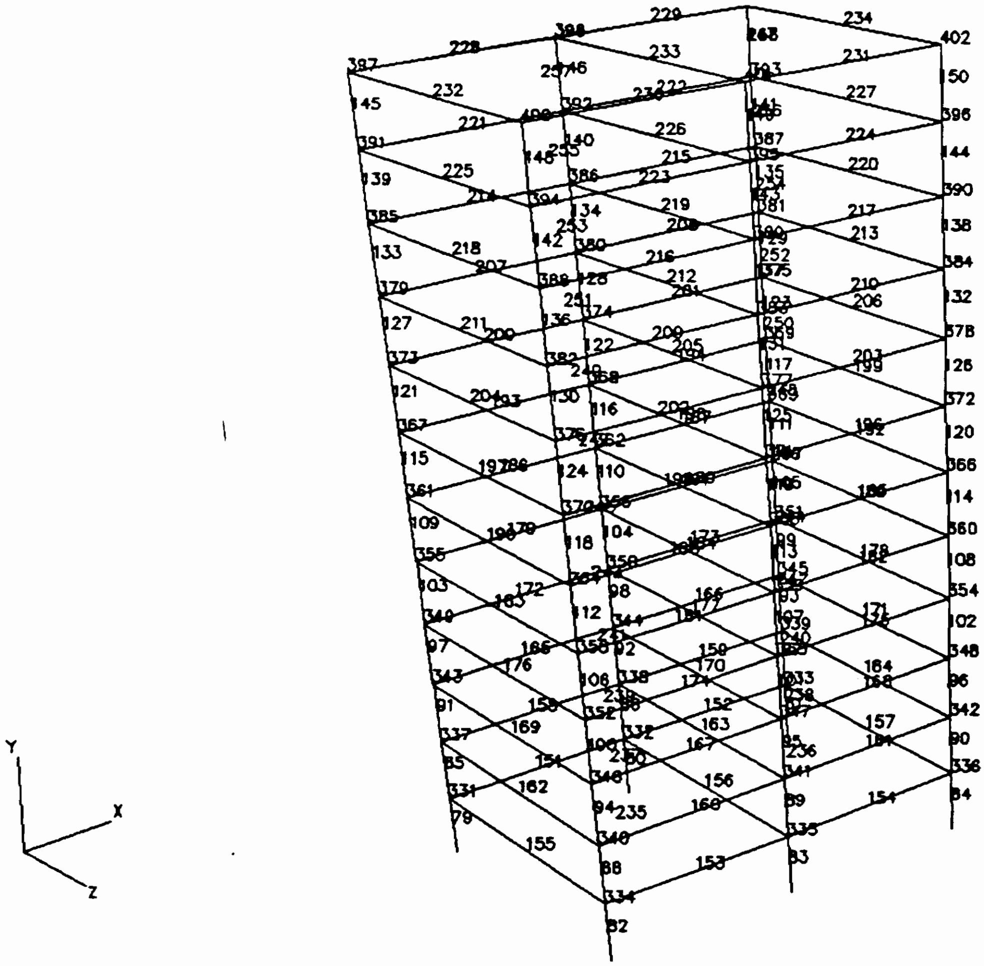


Figure 23. Structure for comparison of time periods.

Table 9. Time period (seconds) of the structure for different cases.

	FUNDAMENTAL PERIOD	
CONDITION	LONGITUDINAL	TRANSVERSE
ONLY FRAME	1.58763	1.49618
FRAME WITH SLAVED JOINTS	1.55884	1.44972
FRAME WITH PLATE ELEMENTS	2.47505	2.13655
FRAME WITH PLATE ELEMENTS AND SLAVED JOINTS JUST BELOW SLAB LEVEL	2.47463	2.13659
FRAME WITH PLATE ELEMENTS AND SLAVED JOINTS JUST ABOVE AND BELOW SLAB LEVEL	2.46338	2.13659
UBC 1976	1.247	1.138

The plan and the elevation of the building are shown in Figures 12 through 14 (Chapter 5). The design data and the other details of the building are given in Section 5.3. Figure 24 shows the computer model for the building frame. Plate elements are used for the shear wall and floor slabs.

6.5.2 DESIGN LATERAL FORCES FOR TRANSVERSE DIRECTION [UBC, 1988]

Period, $T = C_t(H_n)^{0.75}$

where h_n = 88 feet (Height of the building)

$C_t = 0.1/(A_c)^{0.5}$

$A_c = \Sigma A_e[0.20 + (D_e/H_n)^2]$

D_e = 20 feet.

A_e = (12/20) x 20 sq ft.

A_c = 2 x 20 x [0.20 + $(20/88)^2$] = 10.06 sq ft.

C_t = 0.10/$(10.06)^{0.5}$ = .0315

Period, T = 0.0315 (88) 0.75 = 0.905 Seconds.

(1.204 calculated by ISDS)

Base shear, $V = (ZIC/R_w)W$

Where R_w = 12 in zones 3 and 4, SMRSF rigid

Z = 0.40 (for zone 4)

I = 1.50 (for essential facilities)

C = 1.25/$(T)^{2/3}$ = 1.25 x 1/$(0.905)^{2/3}$

= 1.336 < 2.75 (OK)

V = (0.40 X 1.50 X 1.336/12) W = 0.0668W

6.5.3 DESIGN LATERAL FORCES FOR LONGITUDINAL DIRECTION [UBC, 1988]

Period, $T = C_t(H_n)^{0.75}$ = :ƒ.0.03x$(88)^{0.75}$ = 0.862 Sec.

The value calculated by ISDS is 0.438 seconds.

C = 1.25 x 1/$(0.862)^{2/3}$ = 1.38 < 2.75 (OK)

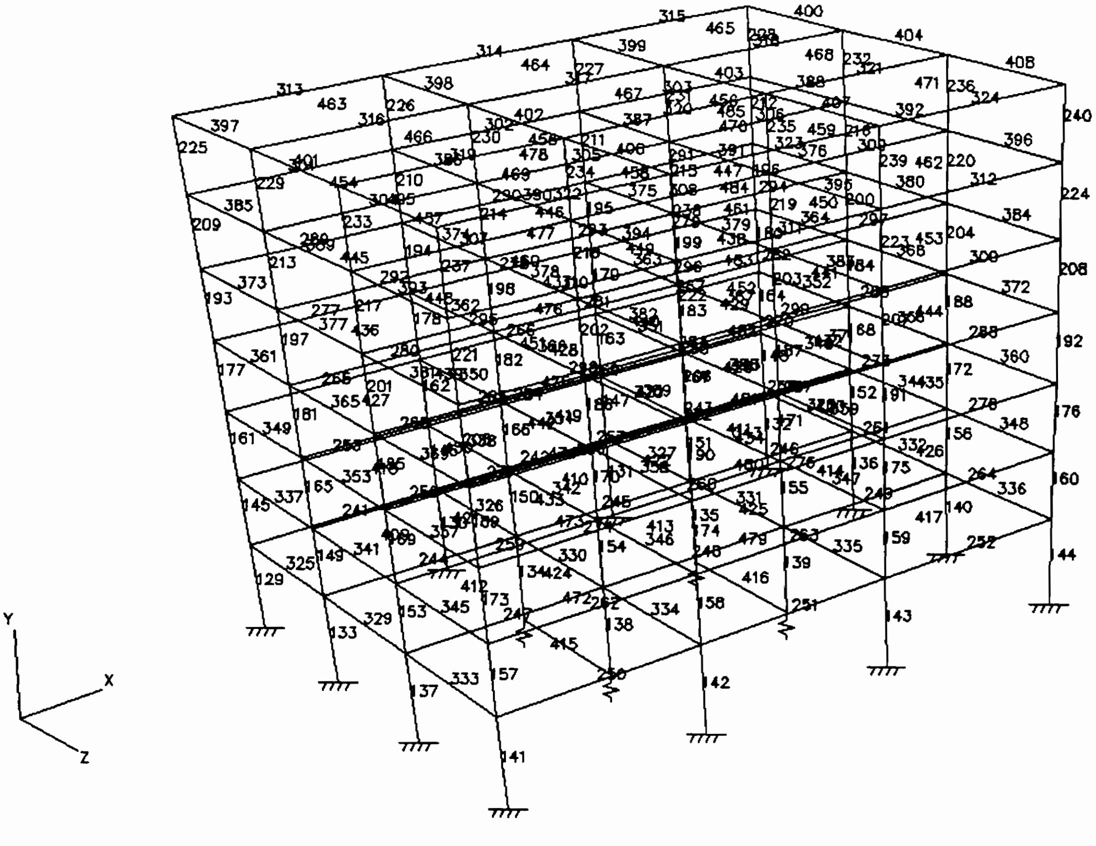


Figure 24. Computer model for Example #2 frame.

$R_w = 12$

$C/R_w = 0.115 > 0.075$

Base shear = (0.40 x 1.50 x 1.38/12) W = 0.069 W

6.5.4 CALCULATION OF DEAD LOAD OF STRUCTURE (W)

1. Dead load from slab = (7.50/12) x 85.25 x 62.5 x 150 = 499511.72 lb
2. Dead load from beams = 1.25 x 1.8333 x 250 x 150 = 87890 lb
3. Dead load from beams = 1.25 x 2.75 x 301 x 150 = 119930 lb
4. Dead load from columns = 1.25 x 2.5 x 16 x 11 x 150 = 75937.50 lb
5. Dead load from shear wall = 1 x 2 x 17.5 x 10.125 x 150 = 53156.25 lb
6. Superimposed dead load = 30 x 82.75 x 60 = 148950 lb

Therefore total dead load = 985.37 kips say 1000 kips

Tables 10 and 11 show design lateral forces in transverse and longitudinal direction (for the entire structure).

F_t = 0.07 x 0.905 x 464.26 = 29.41 kips (transverse).

F_t = 0.07 x 0.862 x 479.55 = 28.94 kips (longitudinal).

V = 0.0668 x 6950 = 464.26 kips

F_r = Lateral force at roof level.

$= F_t + (V - F_t)W_xH_x/\Sigma(W_iH_i)$

Lateral force at roof level (transverse) =

29.41 + (464.26 - 29.41) x 88 x 950/359600 = 130.50 kips

F_7 = Lateral force at 7th story level =

(464.26 - 29.41) x 76 x 1000/359600 = 91.90 kips

Similarly lateral forces at all other levels have been calculated and presented in Tables 10 and 11.

F_r = Lateral force at roof level (longitudinal) =

28.94 + (479.55 - 28.94) x 88 x 950/359600

= 133.70 kips

F_7 = Lateral force at 7th story level =

(479.55 -28.94) x 76 x 1000/359600 = 95.23 kips.

Similarly lateral forces at all other levels are calculated and presented in Tables 10 and 11. Also shown in Tables 10 and 11 are the story shears corresponding to the distributed sesimic forces.

For comparison, the wind forces and story shears corresponding to a basic wind speed of 90 mph and exposure B (Urban and Suburban areas), computed as prescribed by UBC 1988, are shown for each direction in Tables 10 and 11.

6.5.5 CALCULATION OF WIND PRESSURE [UBC, 1988]

Design wind pressure for structures or elements of structures are determined for any height in accordance with the following formula :

$$p = C_q C_e Q_s I$$

p = Design wind pressure

C_e = 1.1 (exposure B)

$C_q = 0.80$

Q_s = 21 psf.

I = Importance factor = 1.15 (essential facilities)

Design wind pressure at roof level

p = 1.1 x 0.80 x 1.15 x 21 = 21.26 psf

Similarly design wind pressure at all other levels are calculated and presented in Tables 10 and 11. Design wind pressures at various levels are shown in Figure 25. The lateral displacements due to seismic and wind forces presented in Table 10 and 11 are plotted in Figure 26.

Table 10. Design lateral forces in transverse direction.

		SEISMIC FORCES				WIND FORCES		
FLOOR LEVEL	HEIGHT Hx, Ft	STORY WEIGHT Wx, kips	Wx Hx Kip-Ft	LATERAL FORCE Fx, Kips	STORY SHEAR SUM(Fx) Kips	WIND PRESSURE psf	LATERAL FORCE	STORY SHEAR SUM(Fx) Kips
R	88	950	83600	130.5		21.26	10.87	
7	76	1000	76000	91.9	130.5	21.26	21.75	10.87
6	64	1000	64000	77.39	222.4	19.96	20.42	32.62
5	52	1000	52000	62.88	299.79	19.64	20.1	53.04
4	40	1000	40000	48.37	362.67	17.39	17.8	73.14
3	28	1000	28000	33.86	411.04	15.14	15.5	90.94
2	16	1000	16000	19.35	444.9	14.08	16.8	106.44
		6950	359600		464.25			123.24

6.5.6 OBSERVATIONS MADE DURING STUDY OF EXAMPLE #2

Table 11. Design lateral forces in longitudinal direction.

		SEISMIC FORCES				WIND FORCES		
FLOOR LEVEL	HEIGHT Hx, Ft	STORY WEIGHT Wx Kips	Wx Hx Kip-Ft	LATERAL FORCE Fx Kips	STORY SHEAR SUM(Fx) Kips	WIND PRESSURE psf	LATERAL FORCE	STORY SHEAR SUM(Fx) Kips
R	88	950	83600	133.7		21.26	7.97	
7	76	1000	76000	95.23	133.7	21.26	15.95	7.97
6	64	1000	64000	80.2	228.93	19.96	14.97	23.92
5	52	1000	52000	65.16	309.13	19.64	14.73	38.39
4	40	1000	40000	50.12	374.29	17.39	13.04	53.62
3	28	1000	28000	35.08	424.41	15.14	11.36	66.66
2	16	1000	16000	20.05	459.49	14.08	12.32	78.02
		6950	359600		479.54			90.34

6.5.6.1 GENERAL

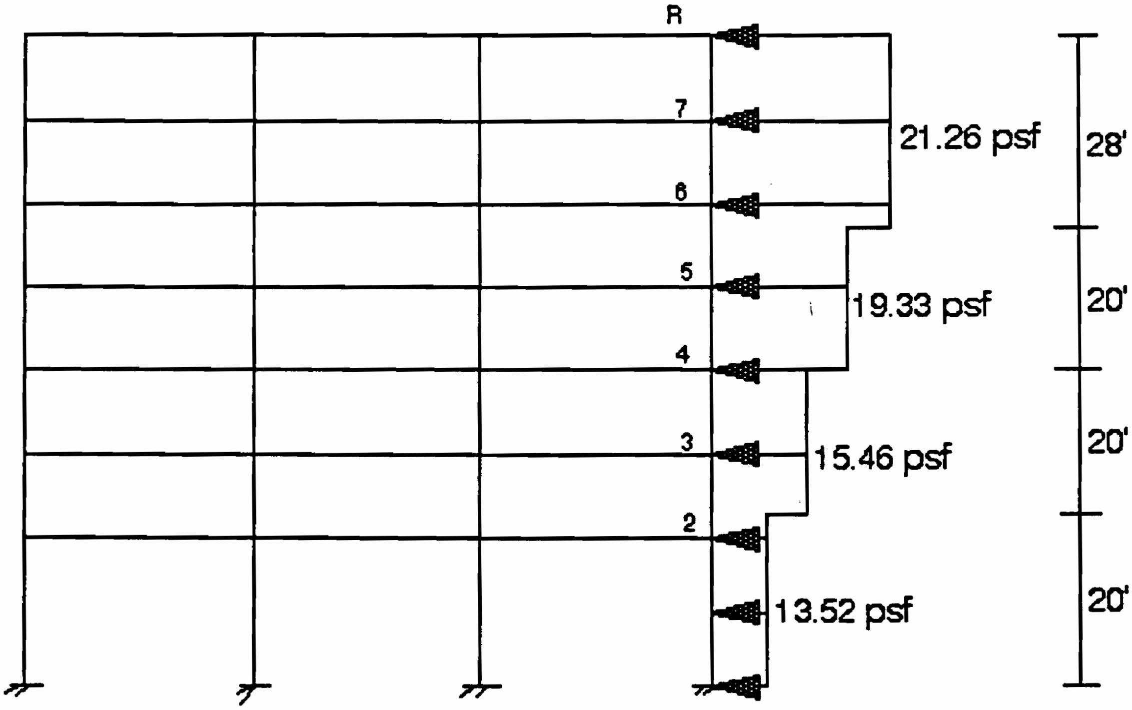


Figure 25. Design wind pressures at various levels.

The ratio of total wind force in the transverse direction to that in the longitudinal direction (See Tables 10 and 11) is 1.36. The corresponding ratio for seismic forces is 0.968. The ratio of the wind forces is directly proportional to ratio of the length to breadth of the structure. The structure has a smaller stiffness in the longitudinal direction, hence the displacements due to seismic forces in the longitudinal direction are greater than that in the transverse direction. Displacements due to wind in transverse direction are less than in the longitudinal direction because of differences in surface area and stiffness of structure in those directions.

It is evident from the analyses that the wind forces are external forces, the magnitudes of which are proportional to the exposed surface, while the seismic forces are internal forces depending primarily on the mass and stiffness properties of the structure.

Figure 26 shows lateral displacements of the structure in the transverse and longitudinal directions under seismic and wind loads in a graphical form. One can observe from the figure, the difference in the shapes of the curves plotted. For frames in the longitudinal direction (whether subjected to lateral earthquake or wind forces), the whole structure deflects as a shear-type building as shown by curves EQL and WIL. This occurs because the floor diaphragm action dominates the response. The shapes of curves EQT and WIT suggest a cantilever behavior for the structure when subjected to lateral forces in the transverse direction. This indicates that the shear wall dominates the applied lateral force response in the transverse direction.

6.5.6.2 DISTRIBUTION OF SHEAR IN TRANSVERSE DIRECTION

It was observed from the analyses of example #2 frame that in the transverse direction most of the shear at the base of the structure was taken by the shear wall. The ratio of the shear taken by the frame to the shear taken by the shear wall increases as we go to the top of the structure.

Table 12 shows the percentage of lateral force (transverse) taken by the two exterior frames and the two interior frames with the shear walls. It is seen that, at the base, 90% of the lateral force is taken by the frames having the shear walls. At the roof level, the frame takes more of the lateral force, i.e., the frame ″supports″ the wall at the top.

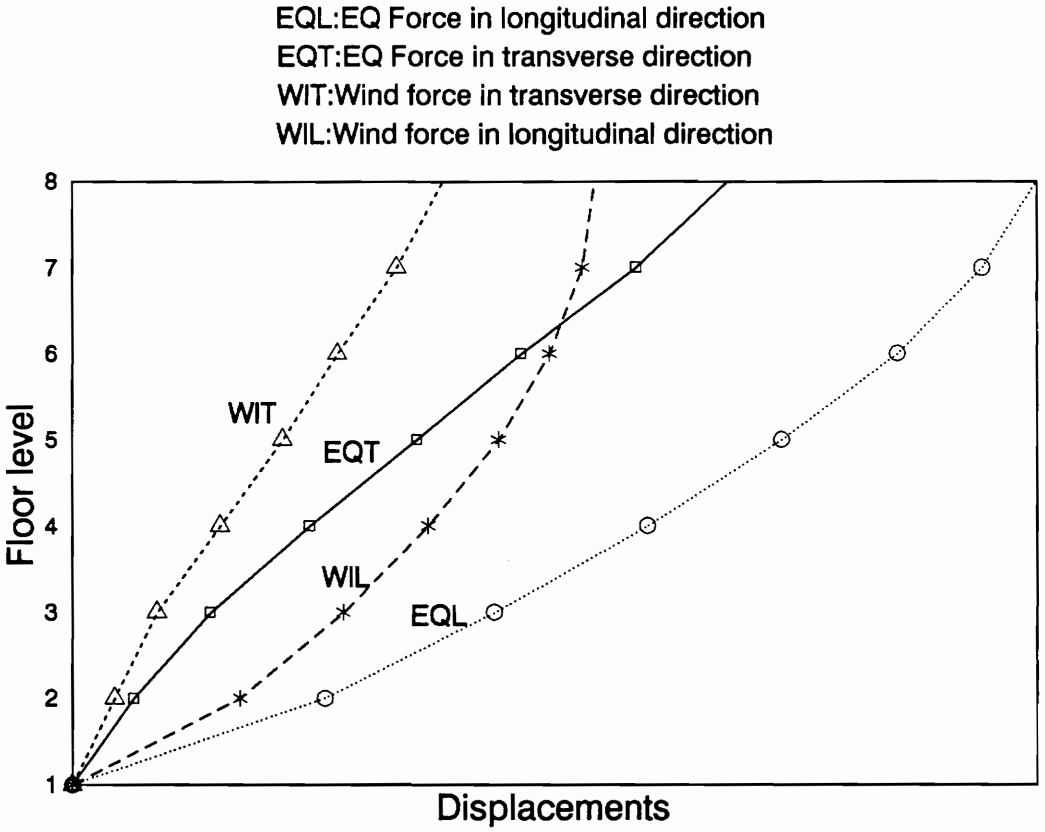


Figure 26. Lateral displacements under seismic and wind loads.

Figure 27 shows results tabulated in Table 12 and observation made in Section 6.5.6.3 in a graphical form.

6.5.6.3 DISTRIBUTION OF SHEAR IN LONGITUDINAL DIRECTION

In the longitudinal direction (where the presence of shear wall does not greatly affect the stiffness of the structure) the two exterior frames were observed to take 40% of total shear while the two interior frames took the remaining shear and it was not seen to vary along the height of the structure.

6.6 COMPARISON OF MOMENTS IN AN INTERIOR LONGITUDINAL FRAME OF EXAMPLE #2

Table 13 compares values of moments in various members due to earthquake force in the longitudinal direction as calculated by the portal method and ISDS. Figure 28 shows the values of shear, moment and axial force in various members of the interior longitudinal frame due to earthquake force in longitudinal direction. The values of lateral forces (and hence moments) at different levels in Figure 28 were arrived at by using the observation regarding distribution of shear in longitudinal direction described in Section 6.5.6.3. The general procedure, and the steps involved in the calculation of the forces and moments in a frame by portal method, described in Section 5.2, were used in arriving at the values shown in Figure 28. These values are also presented in Table 13. Table 13 also presents values of moments due to dead and live loads (factored) calculated using the moment distribution method.

It is observed from Table 13 that there is a maximum percentage difference of 30.25% and an average percentage difference of 16.28% which is quite reasonable. The values using the portal method were based on the results of the ISDS analysis showing that the interior frames in longitudinal direction take approximately 1.50 times the lateral force in the external frames.

The values of the dead load and live load moments from hand computation (using moment distribution method on a reduced frame which includes a single floor and its attached columns) and

Table 12. Distribution of shear in transverse direction.

FLOOR	R	7	6	5	4	3	2
TWO EXTERNAL FRAMES (% OF TOTAL SHEAR)	80	50	45	30	24	20	10
TWO INTERNAL FRAMES (% OF TOTAL SHEAR)	20	50	55	70	76	80	90

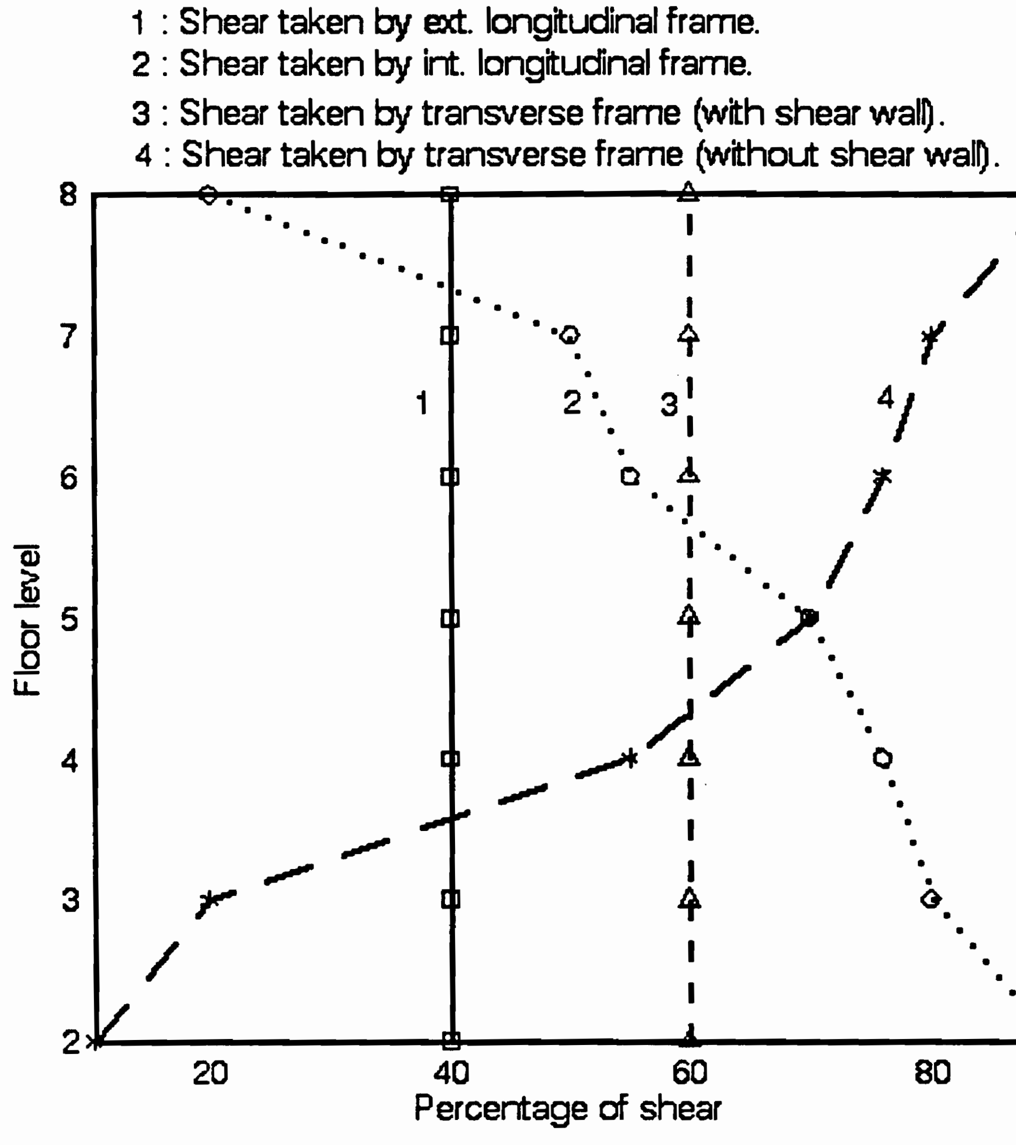


Figure 27. Distribution of shear between frame and shear walls.

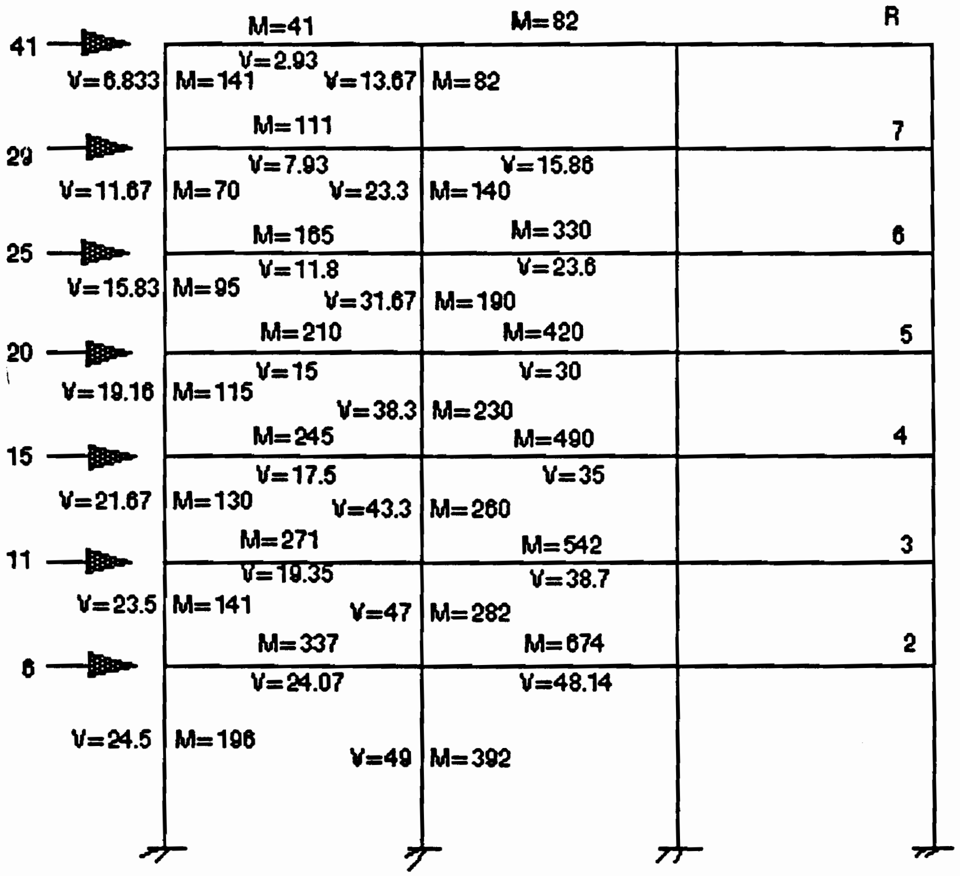


Figure 28. Moments (kip-ft) calculated by portal method.

Table 13. Comparison of moments (kip-ft).

	HAND COMP.		STAAD	
MEMBER	EQL PORTAL	LL+DL MOM. DIST.	EQL	LL+DL
133 Ist STORY EXT. COL.	196	56	224	51
149 2nd STORY EXT. COL.	141	74	184	87
213 6th STORY EXT. COL.	70	74	88	68
229 7th STORY EXT. COL.	41	108	51	91
244 1st STORY EXT. BEAM	337	150 392	348	191 442
256 2nd STORY EXT. BEAM	271	158 390	297	205 450
304 6th STORY EXT. BEAM	111	158 390	119	188 476
316 7th STORY EXT. BEAM	41	108 405	47	143 472

Values of moments for columns : Value at the top of the column
Value of moments for beams : Value at LHS (END 1)

the ISDS agree reasonably well. The maximum percentage difference is 26% which is not unexpected since the moment distribution method does not take into account the effect axial loads.

All this goes to show that the distribution of lateral forces among different stories using UBC and that obtained by ISDS agree reasonably well and that they corroborate each other.

6.7 COMPARISON OF DESIGN FOR EXAMPLE #2

6.7.1 DESIGN OF BEAM (#244, See Figure 24)

1. DESIGN OF BEAM FOR NEGATIVE MOMENT

The beam is designed as a rectangular beam.

k_n = 1041 psi. [Ferguson, 1979]

Compressive strength of concrete = 4000 psi.

yield strength of steel = 60000 psi.

$M_n = M_u/\phi$ = 587.70/0.9 = 653 kip-feet.

$M_n = k_n b d^2$

653 x 12000 = 1041 x 15 x d^2

Therefore, required d = 22.40 inches.

Assuming #8 bars,

$d_{(available)}$ = 33 - 0.5 - 1.50 - 1.0/2 = 30.5 inches.

Clear cover to bottom bar = 1.50 inches.

a/d = 0.38 [Ferguson, 1979]

Therefore a = 0.38 x 30.5 = 11.59 inches.

z = 30.5 - 0.50 x 11.59 = 24.71 inches.

$M_n = A_s f_y z$

Therefore, trial A_s = 653 x 12/(60 x 24.71) = 5.29 inches.

New a = 5.29 x 60/(0.85 x 4 x 15) = 6.22 inches.

New z = 30.5 - 0.50 x 6.22 = 27.39 inches.

A_s = 653 x 12/(60 x 27.39) = 4.77 sq inches.

Iterating further, we arrive at a value of A_s = 4.72 sq in.

Provide 6 #8 bars (4.74 sq in.)

(ISDS provided 5 #9, A_s = 5 sq.in.)

Negative moment at the other end = 862.84 kip-feet.

Following the same procedure as above we arrive at :

A_s = 7.50 sq inches.

Provide 10 #8 bars in two rows (same as provided by ISDS).

2. DESIGN OF BEAM FOR POSITIVE MOMENT

The beam is designed as a T-beam to consider the effect of flange.

Effective flange width is the smallest of :

1. span/4 = 28 x 12/4 = 84 inches. (governs)
2. 8 x thickness of slab + overhang = 8 x 7.50 x 2 + 15 = 135 inches.
3. Beam spacing = 20 x 12 = 240 inches.

M_n = 377.66/0.9 = 419.62 kip-feet.

Assume #9 bars,

$d_{(available)}$ = 33 - 0.5 - 1.50 - 1.128/2 = 30.44 inches.

Approximate lever arm is the larger of :

1. d = 0.9 x 30.44 = 27.40 inches. (governs)
2. d - h_f /2 = 30.44 - 7.50/2 = 26.69 inches.

Trial A_s = 419.62 x 12/(26.69 x 60) = 3.14 sq inches.

The beam is designed as a rectangular beam 84 inches wide, since from calculations, the neutral axis was found to lie within the flange.

$a = A_s f_y/(0.85 f_c b)$

= 3.14 x 60/(0.85 x 4 x 84) = 0.66 inches.

$z = 30.44 - 0.50 \times 0.66 = 30.11$ inches.

$A_s = 419.62 \times 12/(60 \times 30.11) = 2.79$ sq inches.

New $a = 0.66 \times 2.79/3.14 = 0.58$ inches.

New $z = 30.14$ inches.

$A_s = 2.78$ sq inches.

No. of #9 bars = 2.78/1 = 2.78

Therefore provide 3 #9 bars (same as provided by ISDS).

3. CALCULATION OF SHEAR REINFORCEMENT

For members subjected to shear and flexure only,

Shear taken by concrete = $2 \times 4000^{0.5} \times 15 \times 30.44 = 57.7$ kip

At a distance of 'effective depth of beam' from support,

$V_u = 85.52$ kips

$V_n = 85.52/0.85 = 100.62$ kips. $> V_c$

$V_s = 100.62 - 57.75 = 42.8$ kips.

$V_s = A_v f_y d/s$

Assume #4 stirrups

Spacing of stirrups, $s = 2 \times 0.20 \times 60 \times 30.44/42.86 = 17.04$ inches c/c.

Maximum spacing = d/2 = 15.22 inches c/c (ISDS provides #4 at 15.3 in. c/c.

To provide minimum stirrup reinforcement up to a distance where

$V_n > V_c$

At start : up to 13 feet 3 in. (15′ 8″ provided by ISDS)

At end : up to 16 feet (18′ 2″ provided by ISDS)

According to ACI code 318-89 the maximum spacing of the hoops shall not exceed :

1. d/4 (governs).

2. 8 x d_b (Diameter of smallest longitudinal bar).

3. 24 x diameter of hoop bar.

4. 12 inches.

Therefore, provide #4 bars at 7.50 inches c/c, but beyond the region where $V_n < 0.50\ V_c$, maximum spacing of hoop reinforcement could be d/2, i.e., 15.3 inches c/c [American Concrete Institute, 1983].

4. DEVELOPMENT OF POSITIVE AND NEGATIVE MOMENT REINFORCEMENT

Positive moment strength at joint face shall be not less than half of the negative moment strength provided at that face of the joint [American Concrete Institute, 1983]. This criteria is not satisfied at one end of the beam #244 in the design done by ISDS.

Development of negative moment reinforcement is satisfied and is as explained in Section 6.3.1. It is seen that positive and negative moment development provided by ISDS is slightly more conservative.

6.7.2 DESIGN OF COLUMN

The column must be designed for an axial load of 798.25 kips and a moment of 353.5 kip-feet.

Compressive strength of concrete = 4000 psi.

Yield strength of steel = 60000 psi.

$$M_u/(A_g h) = 353.5 \times 12 /(15 \times 30 \times 15) = 0.628$$

$$P_u/A_g = 798.25/(15 \times 30) = 1.774$$

Referring to 'load moment strength interaction' diagram for R4-60.75 columns, we get :

$$\rho_g = 0.033$$

Therefore, $A_s = 0.033 \times 15 \times 30 = 14.85$ sq inches.

Providing #14 bars, number of bars required = 8

Therefore provide 8 #14 (same as provided by ISDS).

6.7.3 DESIGN OF SHEAR WALL [Portland Cement Association, 1983]

ISDS program assumed length of shear wall to be 17.25 feet (in the transverse direction of the structure). Since the moments were found to be nominal, minimum amount of steel required as specified by ACI code was provided. Two curtains of reinforcements were provided. Required area of reinforcement in each direction per foot of wall

$$\text{Vertical } A_s = 0.0015 \times 12 \times 12 = 0.216 \text{ sq in/feet.}$$

$$\text{Horizontal } A_s = 0.0025 \times 12 \times 12 = 0.36 \text{ sq in/feet.}$$

Area of reinforcement given by ISDS in each curtain / feet = 0.259 sq in/feet.

Limitations as to the design specifications and reinforcement detailing are discussed in Chapter 7.

Chapter 7

REMARKS AND CONCLUSIONS

7.1 COMMENTS ON COMPARISON

When we compare the results of the analyses from ISDS and FAP, we find that the results compare very well, with almost negligible percentage difference.

The results from the reinforced concrete design method and the design using ISDS also compare very well. But, ISDS does not specify a few details such as :

1. How much positive moment steel in a beam or a slab must be extended into the supports and up to what distance?
2. Anchorage requirements (in the form of hooks) for negative moment reinforcement in a beam, slab or a shear wall. It simply mentions if anchorage is required or not.
3. What amount of minimum shear reinforcement is required for various structural elements such as beams, columns, and slabs etc?
4. Size and spacing of ties in a column.
5. Cover for main and secondary (ties) reinforcing bars in various structural elements.
6. Class of splice when required.

It was also observed that :

1. The cut-off points at ends of beam for positive and negative moment reinforcements was not symmetrical, even though the loading on the structure is symmetrical.

2. The minimum size of secondary reinforcing bar (stirrup) in a beam is taken as 0.50 inches, even though smaller sized bars could be used.

3. The column is always designed as a member with equal number of bars along its four faces, i.e., the number of bars in the cross-section of the column are always in multiples of four.

The ISDS version used by the writer in this project did not reflect the new provisions for seismic design given in ACI 318-89 code. This was made evident from the comparison of design for Example #2 problem since :

1. ISDS provided maximum spacing of shear reinforcement at 15.3 inches c/c (d/2). But, calculating according to ACI code provisions, maximum spacing came to 7.50 inches c/c (d/4).

2. Positive moment strength at joint face equal to at least half of the negative moment strength was not provided in the beam #244 (at one end).

3. b/d ratio according to ACI 318-89 code should not be less than 0.30. But, when beam size of 12 in. x 45 in. was used (b/d ratio of 0.266), no error message was displayed.

7.2 ADVANTAGES AND DISADVANTAGES

7.2.1 ADVANTAGES

1. In many colleges and universities students find it is very inconvenient to use the mainframe computers in their structural analysis and design courses, either due to restrictions on the use of the computer or the complexity of software. But the ISDS software which runs on a low cost PC obviates such difficulties.

2. A number of powerful computer programs such as STRESS, STRUDL, and NASTRAN are available only on mainframe or minicomputers and many small engineering companies cannot

afford to own or lease this type of equipment. They also find it very expensive and inconvenient to use these analysis programs through a computer service company. But the ISDS which is a versatile and relatively low cost package obviates such a problem.

3. ISDS is capable of handling most of the day-to-day analysis problems which would be encountered in a structural design office or in a typical structural engineering analysis or a design course.

4. It is convenient to use.

5. It does not require any knowledge in computer programming or a complete understanding of the theory governing the analysis procedures which are used, to use the package effectively.

6. It can be used as a learning tool to gain knowledge concerning the behaviour of structural systems.

7. The user can spend more time considering the behaviour of the structure since ISDS greatly reduces analysis time and effort by freeing the user from tedious, time-consuming, and error-prone calculations which are required to perform the analysis of a complex structural system.

8. ISDS combines into a single environment the tools of structural modeling, verification, analysis, and design with drawing generation.

7.2.2 DISADVANTAGES

1. ISDS requires a PC with all its memory free and available. Therefore one has to 'boot' the system before every single use.

2. ISDS is 'case-sensitive'. It accepts only upper-case letters.

3. The programming is not of 'free-format' type. Even a slight change in the sequence of commands results in display of error on the screen.

4. Error messages are not clearly stated. There is no listing of meaning of error messages in the program user's manual.

5. Reinforcement detailing for various structural elements is not done properly. There is no output of diagrams of reinforcement detailing. Other limitations regarding design are discussed in Section 7.1.

6. Very large structures cannot be designed using ISDS which runs on a PC.

7.3 SUGGESTIONS FOR IMPROVEMENTS IN THE PACKAGE

1. Make ISDS 'case-insensitive'.

2. Make programming of 'free-format' type. Whatever the order of the commands given in the input file, ISDS should be able to analyse and design the structure specified.

3. Make a listing of the meaning of the error messages in the program user's manual, since most of the error messages are not clearly stated in the output display.

4. Introduce a structural detailing module in STAAD-PL which would produce drawings of reinforcement details for various structural elements.

5. Make necessary changes in this ISDS version to reflect the special provisions for seismic design given in ACI 318-89.

Bibliography

1. American Concrete Institute, ″Building Code Requirements For Reinforced Concrete (ACI 318-83)″, 1983.

2. American Concrete Institute, ″Deflections of concrete structures (SP 86)″, 1984.

3. Nawy, Edward G., Simplified Reinforced Concrete, Prentice- Hall, Inc., 1986.

4. Ferguson, Phil. M., Reinforced Concrete Fundamentals, John Wiley and Sons, 1979.

5. Fleming, John. F., Structural Engineering Analysis on Personal Computers, McGraw-Hill Book Company, 1986.

6. Holzer, S.M., Computer Analysis of Structures : Matrix Structural Analysis Structured Programming, New York, Elsevier, 1985.

7. Fintel,Mark, ″Handbook of Concrete Engineering″, Van Nostrand Reinhold Ltd., 1974.

8. McCormac, Jack., and Elling, Rudolf E., Structural Analysis - A Classical and Matrix Approach, Harper and Row, Publishers, 1988.

9. Portland Cement Association, "Notes on ACI 318-83 - Building Code Requirements for Reinforced Concrete", 1983.

10. Research Engineers, Inc., "STAAD-III/ISDS - Program User's Manual, 1989.

11. International Conference of Building Officials, Uniform Building Code, 1976 edition.

12. Uniform Building code, 1988 edition.

13. Winter, George., and Nilson, Arthur. H., Design of Concrete Structures, McGraw-Hill Book Company, 1973.

Appendix A. PROGRAM FOR SOLVING EXAMPLE #2

```
STAAD SPACE EXAMPLE PROBLEM WITH FINITE ELEMENT
*
* This program is written by Nadella Navin Kumar. It analyses and designs a seven storyed *
structure subjected to gravity, wind and earthquake loading and different combinations of the same.
*
UNIT KIP FEET
*---------------------------------STRUCTURE GEOMETRY INPUT
JOINT COORDINATE
1 0 0 0 4 84 0 0
REPEAT 3 0 0 20
17 0 16 0 20 84 16 0
REPEAT 3 0 0 20
33 0 28 0 36 84 28 0
REPEAT 3 0 0 20
49 0 40 0 52 84 40 0
```

```
REPEAT 3 0 0 20

65 0 52 0 68 84 52 0

REPEAT 3 0 0 20

81 0 64 0 84 84 64 0

REPEAT 3 0 0 20

97 0 76 0 100 84 76 0

REPEAT 3 0 0 20

113 0 88 0 116 84 88 0

REPEAT 3 0 0 20

MEMB INCI

*

* FOR COLUMNS

129 1 17 240

*

* FOR BEAMS

241 17 18 243 ; 244 21 22 246 ; 247 25 26 249 ; 250 29 30 252

253 33 34 255 ; 256 37 38 258 ; 259 41 42 261 ; 262 45 46 264

265 49 50 267 ; 268 53 54 270 ; 271 57 58 273 ; 274 61 62 276

277 65 66 279 ; 280 69 70 282 ; 283 73 74 285 ; 286 77 78 288

289 81 82 291 ; 292 85 86 294 ; 295 89 90 297 ; 298 93 94 300

301 97 98 303 ; 304 101 102 306 ; 307 105 106 309

310 109 110 312 ; 313 113 114 315 ; 316 117 118 318

319 121 122 321 ; 322 125 126 324 ; 325 17 21 336

337 33 37 348 ; 349 49 53 360 ; 361 65 69 372 ; 373 81 85 384

385 97 101 396 ; 397 113 117 408

*

ELEMENT INCIDENCE

409 17 21 22 18 TO 411 ; 412 21 25 26 22 TO 414
```

```
415 25 29 30 26 TO 417 ; 418 33 37 38 34 TO 420
421 37 41 42 38 TO 423 ; 424 41 45 46 42 TO 426
427 49 53 54 50 TO 429 ; 430 53 57 58 54 TO 432
433 57 61 62 58 TO 435 ; 436 65 69 70 66 TO 438
439 69 73 74 70 TO 441 ; 442 73 77 78 74 TO 444
445 81 85 86 82 TO 447 ; 448 85 89 90 86 TO 450
451 89 93 94 90 TO 453 ; 454 97 101 102 98 TO 456
457 101 105 106 102 TO 459 ; 460 105 109 110 106 TO 462
463 113 117 118 114 TO 465 ; 466 117 121 122 118 TO 468
469 121 125 126 122 TO 471 ; 472 6 10 26 22 ; 473 22 26 42 38
474 38 42 58 54 ; 475 54 58 74 70 ; 476 70 74 90 86
477 86 90 106 102 ; 478 102 106 122 118 ; 479 7 11 27 23
480 23 27 43 39 ; 481 39 43 59 55 ; 482 55 59 75 71
483 71 75 91 87 ; 484 87 91 107 103 ; 485 103 107 123 119
*
MEMB PROP
129 TO 240 PR YD 1.25 ZD 2.50
241 TO 324 PR YD 2.75 ZD 1.25
325 TO 408 PR ZD 1.25 YD 2.50
*
ELEMENT PROP
409 TO 471 TH 0.625
472 TO 485 TH 1.00
*
UNIT FEET
*
CONSTANT ; E 524757.0 ALL
DEN 0.15 ALL
```

```
SUPPORT
1 TO 16 FIXED
UNIT KIP
*--------------------------------LOADINGS
DEFINE UBC LOAD
ZONE 4 K 0.80 I 1.50
SELFWEIGHT
LOAD 1 UBC IN X DIRECTION
UBC LOAD X
CALCULATE NATURAL FREQUENCY
LOAD 2 UBC IN Z DIRECTION
UBC LOAD Z
CALCULATE NATURAL FREQUENCY
*
LOADING 3 DEAD LOAD FROM FLOOR
MEMBER LOAD
241 TO 324 UNI Y -0.3984375
325 TO 408 UNI Y 0.35156
129 TO 240 UNI X -0.46875
330 342 354 366 378 390 402 UNI Y -1.51875
331 343 355 367 379 391 403 UNI Y -1.51875
241 242 243 253 254 255 265 266 267 277 278 279 UNI Y -1.027
289 290 291 301 302 303 313 314 315 250 251 252 UNI Y -1.027
262 263 264 274 275 276 286 287 288 299 300 UNI Y -1.027
310 311 312 322 323 324 UNI Y -1.027
244 TO 249 256 TO 261 268 TO 273  280 TO 285 UNI Y -2.054
292 TO 297 304 TO 309 316 TO 321 UNI Y -2.054
325 328 329 332 333 336 337 340 341 344 345 348 UNI Y -0.825
```

```
349 352 353 356 357 360 361 364 365 368 369 372 UNI Y -0.825

373 376 377 380 381 384 385 388 389 392 393 396 UNI Y -0.825

397 400 401 404 405 408 UNI Y -0.825

326 327 330 331 334 335 338 339 342 343 UNI Y -1.65

346 347 350 351 354 355 358 359 362 363 UNI Y -1.65

366 367 370 371 374 375 378 379 382 383 UNI Y -1.65

386 387 390 391 394 395 398 399 402 403 406 407 UNI Y -1.65

*

LOADING 4 WIND LOAD ON SMALLER FACE (LONGITUDINAL)

* WINDWARD

MEMBER LOAD

397 401 405 UNI Z -0.12756

385 389 393 UNI Z -0.25512

373 377 381 UNI Z -0.23952

361 365 369 UNI Z -0.23568

349 353 357 UNI Z -0.20868

337 341 345 UNI Z -0.18168

325 329 333 UNI Z -0.34496

* TOP

AREA LOAD

313 TO 324 ALOAD 0.0186025

397 TO 408 ALOAD 0.0186025

* LEEWARD SIDE

MEMBER LOAD

400 404 408 UNI Z -0.08

388 392 396 UNI Z -0.15945

376 380 384 UNI Z -0.1497

364 368 372 UNI Z -0.1473
```

```
352 356 360 UNI Z -0.11355
240 244 248 UNI Z -0.2156
*
LOADING 5 LIVE LOAD
MEMBER LOAD
241 242 243 253 254 255 266 267 277 278 279 UNI Y -0.622
289 290 291 301 302 303 313 314 315 250 251 252 UNI -0.622
262 263 264 274 275 276 286 287 288 298 299 300 UNI Y -0.622
310 311 312 322 323 324 UNI Y -0.622
244 TO 249 256 TO 261 268 TO 273 280 TO 285 UNI Y -1.244
292 TO 297 304 TO 309 316 TO 321 UNI Y -1.244
325 328 329 332 333 336 337 341 344 345 348 UNI Y -0.50
349 352 353 356 357 360 361 364 365 368 369 372 UNI Y -0.50
373 376 377 380 381 384 385 388 389 392 393 396 UNI Y -0.50
397 400 401 404 405 408 UNI Y -0.50
326 327 330 331 334 335 338 339 342 343 UNI Y -1.0
346 347 350 351 354 355 358 359 362 363 UNI Y -1.0
366 367 370 371 374 375 378 379 382 383 UNI Y -1.0
386 387 390 391 394 395 398 399 402 403 406 407 UNI Y -1.0
*--------------------------------LOAD COMBINATIONS
LOAD COMB 6 DL+LL
3 1.4 5 1.7
LOAD COMB 7 DL+LL+EL
3 1.05 5 1.275 1 1.275
LOAD COMB 8 DL+LL-EL
3 1.05 5 1.275 1 -1.4025
LOAD COMB 9 DL + EL
3 0.90 1 -1.43
```

```
LOAD COMB 10 DL - EL
3 0.90 1 -1.43 *
PRINT PROBLEM STATISTICS
*
PDELTA ANALYSIS PRINT LOAD DATA
CHANGE
LOAD LIST 6 7 8 9 10
*-------------------------------OUTPUTS
PRINT CG
PRINT JOINT DISPLACEMENTS
PRINT SUPPORT RECATIONS
PRINT MEMBER FORCES
PRINT MEMBER STRESSES LIST 129 133 244 241 TO 264 325 TO 336 337 TO 348
PRINT ELEMENT FORCES LIST 409 TO 417 472 TO 485 463 TO 471
*
*-------------------------------DESIGN
START CONCRETE DESIGN
CODE ACI
UNIT PO IN
FY 60000.0 ALL
FC 4000.0 ALL
TRACK 1.0 MEMBER 129 131 133 134 143 207 239 240 192 224 160
TRACK 1.0 MEMBER 241 242 244 245 330 247 248 342 354 366 329 297 325
DESIGN COLUMN 129 131 133 134 143 207 239 240 192 224 160
DESIGN BEAM 241 242 244 245 330 247 248 342 354 366 329 297 325 340
DESIGN ELEMENT 409 410 412 413 472 TO 485
*
END CONCRETE DESIGN
```

```
*
PLOT STRESS FILE
PLOT DISP FILE
*
FINISH
```

Appendix B. RIGHT HAND RULE

The global joint coordinate system consists of a right-hand orthogonal set of axes and is known by that name due to the orientation of the individual axes with respect to the right hand. If the fingers of the right hand are oriented as shown in the figure 29, the X axis will be considered to extend along the index finger, while the Y and Z axes extend along the middle finger and the thumb respectively. As long as the axes maintain the same relative positions, the three planes XY, XZ, and YZ, which are formed by the axes, are perpendicular to each other, no matter how the coordinate system is rotated in space [Fleming, 1986].

For plane structures, the mathematical model will always be located in the XY plane, and any joint position can be designated by two coordinates , X and Y. For space structures, the X, Y, and Z coordinates must be specified to designate the position of any joint.

An applied joint force component or a computed joint translation component will be considered to be positive if it is in the positive direction of the particular coordinate axis.

The right hand rotation rule is shown in Figure 30. By this rule, the thumb of the right hand is extended along the positive direction of the axis and the fingers are curved naturally. A positive moment or rotation is in the direction extending from the knuckles to the finger tips. This definition holds for moment or rotation components about any of the three coordinate axes.

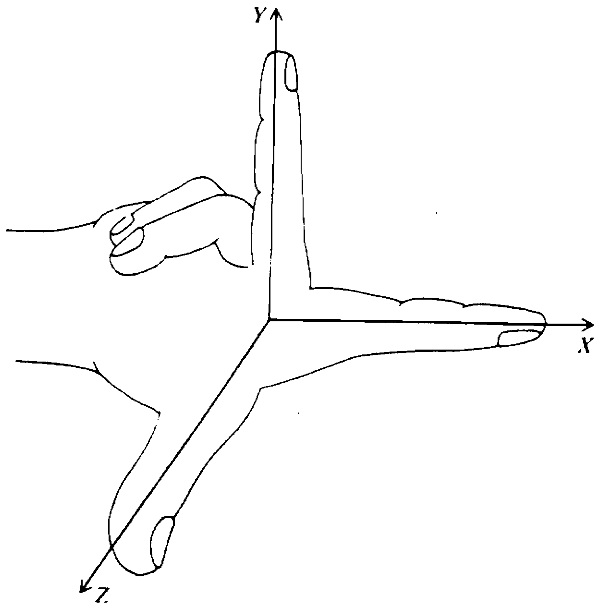


Figure 29. Right-hand coordinate system

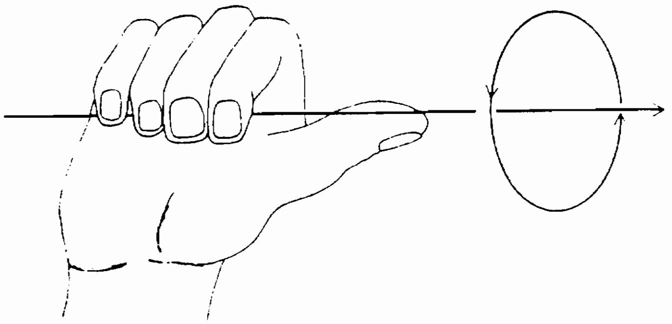


Figure 30. Right-hand rotation rule

Appendix C. GENERAL NOMENCLATURE

a	Depth of equivalent rectangular stress block.
A_b	Area of bar, sq in.
A_c	The combined effective area, in square feet, of the shear walls in the first story of the structure.
A_e	The minimum cross-sectional shear area in any horizontal plane in the first story, in square feet, of a shear wall.
A_g	Gross area of section.
A_s	Area of non-prestressed tension reinforcement, sq in.
A_v	Area of shear reinforcement within a distance s.
b_w,b	Width of compression face of member.
C	Numerical coefficient specified in UBC.
C_e	Combined height, exposure and gust factor coefficient.
C_q	Pressure coefficient for the structure or portion of structure under consideration.
C_t	Numerical coefficient given in UBC 1988.
D	Width of structure in transverse direction in plan.
d_b	Nominal diameter of bar, in.

d_e, d	Distance from extreme compression fiber to centroid of tension reinforcement, in.
D_e	The length, in feet, of a shear wall in the first story in the direction parallel to the applied forces.
f_c	Specified compressive strength of concrete.
F_n	Lateral force applied to level n.
F_t	That portion of the base shear, V, considered concentrated at the top in addition to shear at n'th level.
f_y	Specified yield strength of non-prestressed steel reinforcement, sq in.
h	Overall thickness of the member.
H_f	Thickness of the slab.
h_n	Height in feet above the base to the uppermost main portion of the structure.
I	Importance factor.
K_n	Constant depending upon strength of steel and concrete.
l_a	Additional embedment length at support or at point of inflection, in.
l_d	Development length, in.
M_d	Dead load moment, kip-feet.
M_l	Live load moment, kip-feet.
M_n	Nominal moment strength at section, kip-feet.
M_u	Factored moment at section, kip-feet.
p	Design wind pressure.
P_u	Factored axial load at given eccentricity, kips.
q_s	Wind stagnation pressure at the standard height of 30 feet, psf.
R_w	Numerical coefficient specified in UBC.

s	Spacing of shear or torsion reinforcement in a direction parallel to longitudinal reinforcement, in.
T	Fundamental period of vibration of structure, in Seconds.
V	The total design lateral force or shear at the base.
V_c	Nominal shear strength provided by concrete.
V_n	Nominal shear strength.
V_s	Nominal shear strength provided by shear reinforcement.
V_u	Factored shear force at section.
W	The total seismic dead load defined in UBC.
z	Lever arm factor
Z	Seismic zone factor.
ρ_g	Ratio of area of steel to gross area of section.
Φ	Strength reduction factor.