COMPUTERS &
STRUCTURES
INC.

ETABS®

Three Dimensional Analysis of Building Systems

USER'S MANUAL

by
Ashraf Habibullah

Version 6.2
Revised May, 1997

Developed and written in U.S.A.

COPYRIGHT

Further information and copies of this documentation may be obtained from:

Computers and Structures, Inc.
1995 University Avenue
Berkeley, California 94704 USA

Phone: (510) 845-2177
FAX: (510) 845-4096
e-mail: info@csiberkeley.com

DISCLAIMER

CONSIDERABLE TIME, EFFORT AND EXPENSE HAVE GONE INTO THE DEVELOPMENT AND DOCUMENTATION OF ETABS. THE PROGRAM HAS BEEN THOROUGHLY TESTED AND USED. IN USING THE PROGRAM, HOWEVER, THE USER ACCEPTS AND UNDERSTANDS THAT NO WARRANTY IS EXPRESSED OR IMPLIED BY THE DEVELOPERS OR THE DISTRIBUTORS ON THE ACCURACY OR THE RELIABILITY OF THE PROGRAM.

THE USER MUST EXPLICITLY UNDERSTAND THE ASSUMPTIONS OF THE PROGRAM AND MUST INDEPENDENTLY VERIFY THE RESULTS.

ACKNOWLEDGMENT

Thanks are due to all of the numerous structural engineers, who over the years have given valuable feedback that has contributed toward the enhancement of this product to its current state.

Special recognition is due Dr. Edward L. Wilson, Professor Emeritus, University of California at Berkeley, who was responsible for the conception and development of the original ETABS and whose continued originality has produced many unique concepts that have been implemented in this version.

Recognition is also due Dr. Jeffrey Hollings for his efforts associated with the initial development of ETABS.

The author also wishes to thank his colleagues and co-workers who have participated in the development of this application.

Table of Contents

Chapter I

Introduction

A. Special Purpose Software for Buildings

A wide variety of general purpose computer software is currently available for the static and dynamic structural analysis of complex frame structures [1,2]. Most of these programs can be used for the analysis of multistory frame and shear wall buildings. However, from an analytical point of view, building systems represent a unique class of structures that deserve special treatment.

Special purpose computer programs for addressing such problems have proven to be very practical and efficient, resulting in significant savings in the time associated with data preparation, output interpretation and execution throughput over general purpose computer programs for the following reasons:

- The input and output conventions of the user interfaces correspond to common building terminology. The models are defined logically floor-by-floor, column-by-column, bay-by-bay and wall-by-wall and not as a stream of non-descript nodes and elements as in general purpose computer programs. Thus the structural definition is simple, concise and meaningful.

- The results produced by the programs are in a form directly usable by the engineer. General purpose computer programs produce results in a general form that may need additional processing before they are usable in structural design.

- Experience with numerical techniques has clearly demonstrated that a single solution algorithm to solve all structural problems, as in general purpose programs, does not yield the most efficient solution procedure. Targeting a particular problem, isolating the particular characteristics of the problem, and then designing a special purpose program that takes advantage of the numerical characteristics of the problem will yield an algorithm that is robust and demonstrates much better performance and capacity.

The concept of special purpose programs for building type structures was introduced over 35 years ago [3], however, the need for special purpose programs has never been more evident as Structural Engineers put nonlinear dynamic analysis into practice and use the greater computer power available today to create larger analytical models.

B. The Special Purpose Computer Program ETABS

ETABS is a special purpose computer program for the linear and nonlinear, static and dynamic analysis of buildings [4,5,6,7].

The ETABS building is idealized as an assemblage of column, beam, brace and wall elements interconnected by horizontal floor diaphragm slabs which may be rigid or flexible in their own plane. The basic frame geometry is defined with reference to a simple three-dimensional grid system formed by intersecting floor planes and column lines. With relatively simple modeling techniques very complex framing situations may be considered.

The buildings may be unsymmetrical and non-rectangular in plan. Torsional behavior of the floors and interstory compatibility of the floors are accurately reflected in the results. The solution enforces complete three-dimensional displacement compatibility, making it possible to capture tubular effects associated with the behavior of tall structures having relatively closely spaced columns.

Semi-rigid floor diaphragms may be modeled using a floor element to capture the effects of in-plane floor deformations. This element may span between adjacent levels for the creation of sloped floors.

Modeling of partial diaphragms, such as in mezzanines, setbacks, atriums and floor openings is possible. It is also possible to consider situations with multiple diaphragms at each level thereby allowing the modeling of buildings consisting of several towers rising from a combined structure below or vice-versa.

A diagonal bracing element is available to model braced frames such as A-, X-, or K-braced or eccentrically-braced systems.

A special panel element is implemented for the modeling of general three-dimensional shear wall configurations, such as C-shaped core elevator walls, curved shear walls, discontinuous shear walls and shear walls with arbitrarily located openings. Torsional and warping effects in three-dimensional walls are accurately captured.

The panel element is based upon an isoparametric finite element membrane formulation with the in-plane rotational stiffness being defined. Therefore, any beams or columns that frame into the panel element, in the plane of the wall, will receive full continuity without any special modeling.

One or more panel elements at a particular level may be used to define a "wall," which may be planar or three-dimensional. Integration of the panel finite element internal forces into a set of forces and moments for the "wall" is completely automated in a manner that produces complete equilibrium with the applied forces. The user can apply the versatility of the finite element method to model complex shear wall systems and also expect to get results in a form that is directly usable in the wall design.

Special formulations of a nonlinear support element and two-point nonlinear link element have been implemented for allowing the modeling of biaxial hysteretic and friction pendulum base isolation devices. Uniaxial gap, damper and plasticity options are also available. These elements may be used for the modeling of added stiffness and damping elements, slotted-bolted energy dissipators, supplemental dampers and other passive energy absorbers and for evaluating the effects of three dimensional structural pounding.

The columns and beams may be non prismatic, and beams may have partial fixity at the end connections. Beams may also have uniform, partial uniform or trapezoidal load patterns. The column, beam, brace and panel element formulations include the effects of bending, axial and shear deformations. All elements are intercompatible with each other. The effects of the finite dimensions of the beams and columns on the stiffness of a frame system are automatically included.

Static analyses for user specified vertical and lateral floor or story loads are possible. Vertical uniform loads on the floor elements are automatically converted to span loads on adjoining beams, thereby automating the tedious task of transferring floor tributary loads to the floor beams without explicit modeling of the secondary framing. Generation of lateral wind and seismic load patterns defined by the requirements of various building codes is automated in the solution procedure.

Thermal stress analyses for user specified distributions of temperature are possible.

Three-dimensional mode shapes and frequencies, modal participation factors, direction factors and participating mass percentages are evaluated using subspace iteration or Jacobi diagonalization, depending upon the numerical sensitivity of the system.

The P-delta effects are included in the basic formulation of the structural lateral stiffness matrix as a geometric correction [8]. This causes equilibrium to be satisfied in the deformed position, and the P-delta problem is solved exactly with no iteration and no additional numerical effort. Also, as the correction is on the lateral stiffness matrix, the P-delta effects appear in the static analysis and filter into the Eigen, response spectrum and time history analyses.

Response spectrum analysis is based upon the mode superposition method using the complete quadratic modal combination (CQC) technique [9,10]. The structure may be excited from two different directions in any one run with independent spectra. Composite modal damping effects from supplemental dampers are included in the analysis.

The linear time history analysis uses a variable time step closed form integration technique for the evaluation of the modal coordinates [7]. Time-dependent ground accelerations can excite the structure concurrently in any two orthogonal horizontal directions with independent excitations. The nonlinear time history analysis is based upon a very efficient iterative vector superposition integration scheme [11,12]. The time history results may be displayed as time-functions (such as displacement vs. time) or as function-function (such as force vs. deformation). Response spectrum curves may be created from acceleration time histories generated by ETABS.

Results from the various static load conditions may be combined with each other or with the results from the dynamic response spectrum or time history analyses.

The output includes static and dynamic story displacements and story shears, inter-story drifts, and joint displacements, reactions and member forces.

The ETABS system is integrated with a series of design post processors for steel and concrete design. These processors work off of a centralized data base created by ETABS. The structural geometry and the analytical results are automatically recovered from this database and are never re-entered by the user. All members are checked or designed for a series of user specified loading combinations and the results from the controlling combination are reported. The output is displayed in a form that aids the structural engineer in taking effective remedial action if the reported results are not satisfactory.

C. Advantages of ETABS Over Other Programs

The following are some of the characteristics that are inherent in the basic nature of a building type structure that a general structural analysis program may not recognize, thereby resulting in a significant loss in man-hours, processing time and possibly accuracy:

- Most buildings are of simple geometry with horizontal beams and vertical columns. A simple grid system defined by horizontal floor lines and vertical column lines can establish such a geometry with minimal effort.

- Many of the frames and shear walls in buildings are typical. Most general programs do not recognize this fact; therefore, for typical regions of the structure many of the internal calculations may be unnecessarily duplicated.

- In most buildings the dimensions of the members are large in relation to the bay widths and story heights. These dimensions have a significant effect on the stiffness of the frame. Corrections for these effects must be included in the formulation of the member stiffness. Most general purpose programs work on centerline dimensions and such stiffness corrections are usually very tedious to implement.

- In the analysis of buildings the member forces need to be produced at the outer faces of the supports of the members. Such transformations are not automatic in general purpose programs.

- The in-plane stiffness of the floor systems of most buildings is very high. Such situations justify the introduction of automatic constraints to define rigid floor diaphragms. Such constraints result in a model that produces a set of equilibrium equations that are in general very well conditioned. Also, the computational effort associated with the solution of such a system is smaller, conceivably by several orders of magnitude. In addition, the story diaphragms define the location for the application of lateral floor loads, the locations of the lumping of the story masses and the locations of the linear and nonlinear dynamic degrees of freedom of the structure.

- The automatic generation of the reduced lateral stiffness system, containing the dynamic degrees of freedom, forms the basis for the numerical solution schemes for Eigen analysis, P-Delta analysis, linear and nonlinear time history integration that are unmatched in performance and stability.

- The loading in building systems is of a restricted form. Loads, in general, are either vertically down (dead or live) or lateral (wind or seismic). The vertical loads are usually applied on the floors and beams and the lateral loads are generated at the story levels. Tributary floor loads need to be automatically transferred to the building frames. Also, various code loading requirements need special options that allow convenient generation and combination of the vertical and lateral static and dynamic loadings.

- It is desirable to have a building analysis computer output printed in a special format, such as, in terms of a particular frame, story, column, beam, brace or wall. Also, special output, such as lateral story displacements and inter-story drifts are required.

All of the above mentioned characteristics of building systems are recognized by ETABS, making it ideally suited for the specific application.

D. The ETABS Series of Programs

The **ETABS** System is comprised of the following modules:

- **ETABSIN** - A Windows based model building module
- **ETABS** - The main analysis module
- **ETABSOUT** - A Windows based output display and selective printing module
- **STEELER** - Steel frame design module

- **CONKER** - Concrete frame design module
- **WALLER** - Concrete shear wall design module

E. Organization of Manual

The documentation of the ETABS series of programs is contained in two volumes. The first volume contains this ETABS User's Manual, the ETABSIN User's Manual, the ETABSOUT Manual and the ETABS Examples Manual. The second volume contains the STEELER User's Manual, the CONKER User's Manual and the WALLER User's Manual.

This manual describes the use of the ETABS analysis program. The following are brief descriptions of the remaining chapters of this manual:

- Chapter II describes the installation procedure and the execution procedure for the ETABS program. Brief notes for network users are also contained in this chapter.
- Chapter III describes the terminology used by the ETABS program and the different analysis techniques used.
- Chapter IV describes the various elements available in ETABS to model building structures and their usage.
- Chapter V describes the format of the ETABS input file. It is highly recommended that Chapters III and IV be read first before commencing to model the building.
- Chapter VI describes the contents of the output files produced by ETABS.

F. Program Versions

ETABS Version 6.2 is available in the following versions:

- **ETABS Standard** - with capabilities as documented in this manual except nonlinear time history analysis is not available and problem size is limited to 6000 equations. Design programs are not available.
- **ETABS Plus** - with capabilities as documented in this manual except nonlinear time history analysis is not available and size of problem essentially limited by RAM and/or hard disk availability. The design programs STEELER, CONKER and WALLER are included.
- **ETABS Nonlinear** - with capabilities as documented in this manual including nonlinear time history analysis and size of problem essentially limited by RAM and/or hard disk availability. The design programs STEELER, CONKER and WALLER are included.

Chapter II

Installation and Execution Procedure

This chapter deals with the installation and execution of ETABS on a Windows 95 or a Windows NT 4.0 based computer system.

The programs require at least a 80486 based computer with a minimum of 8MB of RAM (more is recommended).

User familiarity with Windows is assumed.

The complete ETABS package includes:

- This manual
- The design post processors manual (Plus and Nonlinear versions only)
- ETABS floppy disks, containing the following:
  a. Setup program for ETABS
  b. Program executables, support files and sample data for ETABSIN, ETABS and ETABSOUT
  c. Design program executables and sample data for STEELER, CONKER and WALLER (Plus and Nonlinear versions only)

- Hardware copy protection device, except for educational and evaluation versions

- Supplementary disk for Windows 95 and Windows NT.

A. Installing and Testing

The programs provided come with a setup utility to facilitate installation. ETABS must be installed from Windows.

To install the programs:

- Insert the ETABS Disk 1 into drive A
- Select Run from the Start menu of Windows
- Type A:\SETUP in the command line
- Choose OK
- Respond to the prompts of the SETUP to complete the installation

The installation program will:

- Decompress and copy program files to a user specified directory on the hard disk
- Decompress and copy sample data files and results to a subdirectory called EXAMPLES
- Insert a user specified Company Name into the program executable
- Add a Program Group titled ETABS to the Start Menu with icons for the various programs

The program ETABS is protected with a hardware copy protection device. The protection device should be attached to the parallel printer port, LPT1 of the computer. The device goes between the computer and the printer (or any data transfer switches). The device does not require the printer to be connected or, if connected, to be powered.

Network users not using a network device can install the program on a file server directory but can use the ETABS program only from the workstation that has the protection device installed.

Network users using a network device (Nonlinear version only) should install the program on a file server directory. The protection device must be installed on one of the workstations (not on the file server). The workstation with the device is designated as the ETABSserver. The ETABSserver should run the program NSRVGX. This program is provided with the network device. This program must be run from the DOS prompt. Once run, this program is memory resident and allows the protection device to be accessed from any workstation. Concurrent usage of the program is allowed from different workstations up to the limit of the network device. Several network devices can exist on the same network by designating several workstations as ETABSservers. It must be noted that network devices may take a few moments, based on network usage, before they are detected by the ETABS program. Also, the ETABSserver should not be shut down while another workstation is running ETABS.

To access the hardware device correctly some installations may need the Supplementary Disk for Windows 95 or Windows NT to be installed. To install this disk:

- Insert the appropriate disk in drive A
- Run SETUP.EXE from the disk using the Start menu
- Select Install Sentinel Driver from the Functions menu
- Click on OK to accept the path for the driver
- When complete the message "Driver Installed! Restart your system" is displayed. Click on OK and restart your computer

This is a one-time installation process. The Sentinel driver will automatically run every time you start your computer.

Before putting the system into a production mode, the user should test the system by running some of the sample examples provided. The output files produced should then be compared to the corresponding output files that are also provided on the distribution disks.

B. Input Preparation Before Executing ETABS

Before executing ETABS the user needs to prepare the data for the specific structure that is to be analyzed. The user must thoroughly read the manual and understand the basic assumptions of the program before attempting to model the building. The input data for ETABS can be prepared by using the interactive graphical model generator, ETABSIN, which saves the model in a file compatible with the ETABS input data. Alternatively, the file can be directly created by use of a text editor. The data must conform to the program specifications detailed in Chapter V of this manual. Sample examples are discussed and their input data files are provided in the ETABS Examples Manual included in this volume.

C. Executing the ETABS Program

This section explains how to execute the ETABS program.

ETABS is a DOS program which can be run in a DOS Window or can be launched from the Windows Icon.

Running ETABS in a DOS Window

To execute ETABS in a DOS window enter the following command at the DOS prompt:

ETABS ***etabsfile***

Where ***etabsfile*** is the name of the ETABS input file which has been prepared as explained in the previous section. The ***etabsfile*** can include a path.

As an example say that the data associated with the problem the user wishes to analyze has been entered into a data file called EXAMPLE. In order to execute the ETABS program, from the directory where the ETABS input data file is resident, enter the following command at the DOS prompt:

ETABS EXAMPLE

Note: Since no paths have been specified with the ETABS file name the file must reside in the current directory where the command is entered. Also the ETABS executable must reside in the same directory unless a path to the ETABS executable has been activated.

The program will then enter the execution phase and a series of progress messages will be flashed to the screen until the job is complete. A successful execution of the program is so reported on the last screen which also notes the names of the output files created. The messages that come to the screen are also written to a log file with the input file name with a .LOG extension. The log file should always be reviewed for warning messages and should be reviewed for error messages if the program did not complete successfully.

Two command line options are also available when executing ETABS. One or both may be used at a time. The full command line will look as follows:

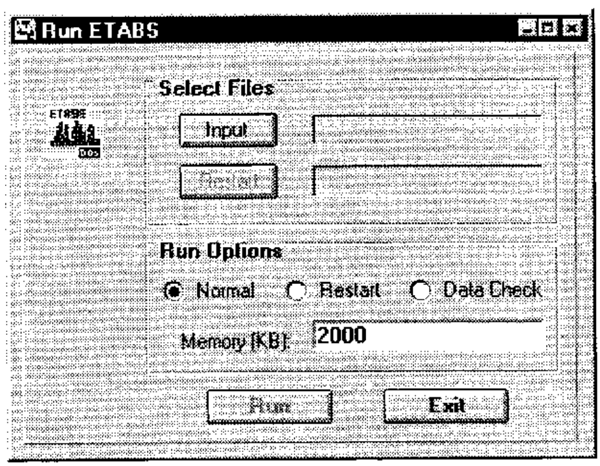
ETABS ***etabsfile*** **/I** **/M:nnnn**

Where the **/I** switch instructs the program to only check the input and not execute the full ETABS program. This option is useful at the data preparation stage and allows the user to quickly check the input data as it is being prepared. This option produces the input echo file with an .EKO extension, which should be thoroughly reviewed for correctness. This option also produces a partial post processing file which can be used with ETABSOUT to graphically check the input.

The **/M:nnnn** switch is to define how much RAM is to be used for data. The amount of RAM is specified in KB as **nnnn**. The default amount of RAM used for data if the /M switch is not used is 2MB. The program runs faster the more memory it is allowed to use. When solving a large problem, if the program runs out of RAM then it should be rerun using this switch to allocate more memory. Also, if only a small problem is being solved, then the switch may be used to reduce the amount of memory allowed to ETABS so other applications running simultaneously have more memory available.

Running ETABS from Windows

To execute ETABS from Windows double click on the ETABS icon in the ETABS program group installed by the setup program. The following dialog box will appear:



Click the **Input** button which brings up the Open File Dialog box. Select the ETABS input file name. Clicking on **Run** launches the ETABS program. The program runs minimized in a DOS window. The amount of memory and the check box for data check run are to specify ETABS command line options as discussed earlier in this section. Restart is discussed in a separate section below.

A successful execution of the program creates several output files in the same directory as the input file. The output can be reviewed through the Windows based graphics display program ETABSOUT. This program uses the binary post processing file with a .PST extension created by the ETABS program. Several ASCII output files are also created by the ETABS program. The contents of these output files is briefly described in Chapter VI. The files can be viewed and printed using any text editor. To print an output file from DOS the **PRINT** command may be used. Appropriate line counts and page ejects are built into the files.

D. Restarting the ETABS Program

In order to execute the ETABS program in the restart mode, the execution procedure is exactly the same as described in the previous section except that the DOS command line now has the name of two input files:

ETABS ***etabsfile*** ***etabsfileold***

Where ***etabsfile*** is the name of the ETABS input file to be analyzed and ***etabsfileold*** is the name of the file which has been analyzed earlier and from which the restart is to occur. Both the ***etabsfile*** and ***etabsoldfile*** can include a path.

Chapter III

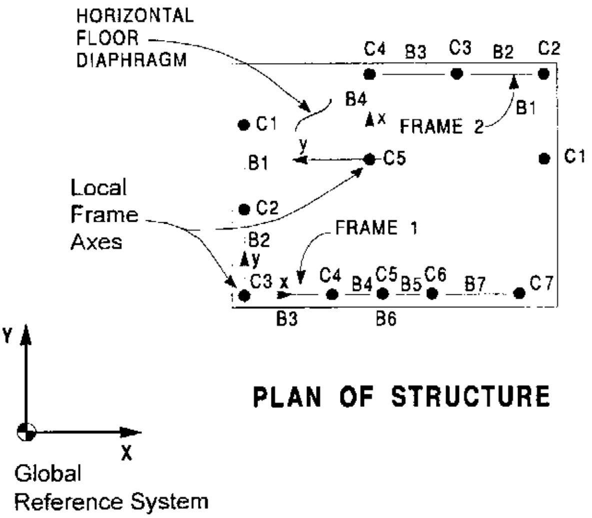
ETABS Terminology and Techniques

This chapter and Chapter IV constitute the basis of the ETABS computer program. It is imperative that the user comprehend the information provided in these chapters in order to use the program correctly. The basic terminology associated with the program and some of the techniques used for solution are detailed in this chapter.

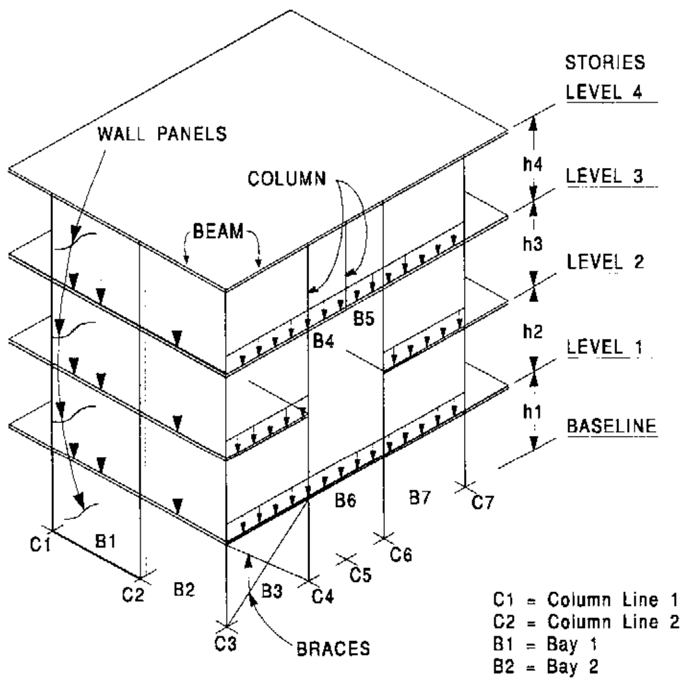
A. ETABS Frames

The program views the building system as an assemblage of vertical frames interconnected at each story level by horizontal floor diaphragms.

A frame basically consists of columns that are vertical and beams that are horizontal. Frames can be three-dimensional with shear walls and diagonal bracing. The gridwork formed by the horizontal floor diaphragms and the vertical column lines forms the basic reference system for the description of the frame. See Figure III-1. The column lines (identified as C1, C2, C3) are located with coordinates in the X-Y plane and the bays (identified as B1, B2, B3) are located in plan as connectivities between the column lines. A bay is defined as a connection between any two column lines.



PLAN OF STRUCTURE



VIEW OF FRAME 1

Typical Building System
Figure III-1

The following is an important ETABS convention: a column line is a vertical line parallel to the Z-axis defined in plan by X,Y coordinates. On each column line there **may** exist columns (or column elements) corresponding to the story levels. For example, in Figure III-1, column line C3 has 4 columns, one corresponding to each level, whereas column line C5 has only one column, corresponding to level 4.

Similarly, for each bay there **may** exist beams (or beam elements) corresponding to the story levels. For example, in Figure III-1, bay B3 has 4 beams, one corresponding to each level, whereas bay B4 has only two beams corresponding to levels 3 and 4, respectively. Note that bay 6, located between C4 and C6, has only one beam at level 1. Summarizing, therefore, **columns** are to **column lines** as **beams** are to **bays.**

The concept of a frame in the ETABS environment is an assemblage of column lines C1, C2, C3 (each of which may or may not have columns), each of which in turn may be connected to each other by beam bays, floor bays, braces or panels. Note that a beam, a brace or a panel always exists between two column lines. The beams, braces and panels are viewed by the program as elements that connect column lines.

Now let us consider a series of column lines in a building and a corresponding set of elements that connect only to this series of column lines.

Then consider a second series of column lines in the same building and a second corresponding set of elements that connect only to the second series of column lines.

If such a situation exists in a building, the first series of column lines and their associated elements constitute the first frame of the building and the second series of column lines and their associated elements constitute a second frame. Thus a "frame" is defined by a series of columns and a corresponding series of elements. The elements of the frame can only connect to the column lines included in the frame. The frames satisfy what is termed **connectivity isolation**. Column lines are linked to each other from a lateral standpoint and the connectivity isolation is only of a vertical nature.

In Figure III-1 the plan of the structure shows two frames that are isolated from a connectivity standpoint. Modeling the structure as two frames (one with 7 column lines and one with 5 column lines) lends significantly increased efficiency and capacity to the solution process. However, there is nothing that prohibits the user from modeling the structure as a single frame with (7 + 5) = 12 column lines, if so desired, even if the portions of the combined frame demonstrate connectivity isolation. **In fact, unless the structure consists of very large frames that are typical, it is recommended that the structure be modeled as a single frame.**

As the first step of the frame data preparation, it is recommended that folded out frame elevations be drawn of all the typical frames of the structure. The story lines and the column lines that form the basic gridwork for the frame description should be shown on the elevations, including the column line numbers and the bay numbers.

The beams and floors associated with a particular story exist at the corresponding story line, whereas the columns, panels and diagonals associated with a particular story exist **below** the corresponding story line. Beams, columns, panels, braces and floor sharing the same properties may be generated over a series of story levels.

Although the basic gridwork for the definition of frame geometry is essentially rectangular, complex situations can be very easily modeled by creative use of the brace and panel elements and by defining geometries using null beam and column property assignments where needed. Modeling of X-braced, K-braced and eccentrically braced systems, A-frames, setback frames and Vierendeel trusses are all possible. Complex systems consisting of discontinuous shear walls and shear walls with arbitrarily located openings are easily modeled using the shear panel and the special modeling techniques described later in Chapter IV.

The columns, beams, panels and diagonals must be assigned property numbers. Properties associated with standard AISC sections are available in a program data base.

Vertical load data input for load conditions I through III is prepared as part of the frame data. Self weight of the frame members modeled can be included automatically. The beam span loadings associated with load conditions I through III must be assigned pattern numbers. In determining these vertical load patterns the user must take care to include those loads coming from secondary framing members which are not part of the model.

B. ETABS Floor Diaphragms

Unless special program options are used, the horizontal floor at any one level is modeled by the program as a rigid horizontal plane (or diaphragm) that has infinite in-plane stiffness (in the X-Y plane) and no out-of-plane stiffness. There may be one or more floor diaphragms associated with each level. This horizontal plane, of each diaphragm, in effect, ties together all the column lines associated with the diaphragm. Therefore, all the column lines connected to a rigid diaphragm cannot displace (in the X, Y or Θ_Z directions) independent of each other. In other words, two column lines on a rigid diaphragm will have exactly the same spatial relationship with respect to each other (in the X-Y plane) before and after the diaphragm displaces.

There are options available whereby the user may arbitrarily disconnect particular column lines from the diaphragm to allow unrestrained column displacements.

A floor diaphragm at a story level can either exist and have infinite in-plane stiffness, not exist at all and have zero stiffness, or have a finite, semi-rigid stiffness via the use of flexible floor elements.

The rigid floor diaphragms must be horizontal. Sloped diaphragms may be modeled using the flexible floor element.

All masses required for the dynamic analysis of the structure are associated with the rigid diaphragms. The masses are lumped at the centers of mass of the corresponding diaphragms. Masses of elements are automatically included in a combined diaphragm mass with a corresponding contribution to the mass moments of inertia.

In the dynamic analysis of systems where the diaphragm flexibility is to be considered, the floors are modeled as a series of multiple rigid diaphragms and flexible floor elements. In general the number of diaphragms at a particular level will be governed by the required distribution of the mass of the floor, as the masses are lumped at the centers of mass of each rigid diaphragm in a tributary fashion.

C. ETABS Reference Coordinate Systems

a. Global Reference Point and Reference Axes

The global reference point is an arbitrary point selected by the user in the plan view of the building. This point is the origin of the **global** X, Y axes and is the same for all levels of the structure. The story centers of mass, the structural lateral loads (static and dynamic) and the positions of the various frames are all located with respect to the reference point and reference axes. The lateral loading and geometry are thereby uniquely located with respect to each other, regardless of the choice of the reference point. The reference point may be chosen to be any dimensionally convenient point in the structural plan. There is only one global reference point for the whole building.

b. Frame Reference Point and Reference Axes

The frame reference point is an arbitrary reference point selected by the user in the plan view of the frame. The point is the origin of the frame **local** x, y axes and is the same for all the levels of the frame. All frame data is prepared with respect to these axes. This reference point may be chosen as any dimensionally convenient point on the structural plan. Every frame has its own frame reference point. If there are no duplicate frames in the structure, it is usually convenient to coincide the global and frame reference point locations. Note that the local frame z-axis and the global Z-axis are always parallel, vertically up.

D. Frame Location Data

It is possible for the structure to have some frames (with identical geometry and vertical loading) that are repeated at different locations in the structural plan. The data for such typical frames need only be prepared once.

In other words, one set of frame data is set up for each different frame in the structure. Every one of the different frames may then be placed at one or more locations in the structural plan, via the frame location data. This data locates the local axes of the frames with respect to the global reference axes. Most buildings usually end up being modeled as a single frame.

Static and dynamic story displacements and interstory drift ratios are reported by the program at the point at each story level that corresponds to the origin of the frame (in plan). By creating a dummy frame (single column line and all zero properties) and locating it at selected points on the structural plan, the user can conveniently obtain critical static and dynamic displacement and drift information which otherwise may be very tedious or impossible to extrapolate.

E. ETABS Load Conditions and Load Cases

It is important to recognize the subtle distinction between a load condition and a load case as defined in the terminology of ETABS.

The load conditions are the independent loadings for which the structure is analyzed internally. These loadings are: three vertical static load conditions (I, II and III), three lateral static load conditions (A, B and C) and two dynamic load conditions (D1 and D2).

The load cases are assembled as combinations of the load conditions. Although the number of independent load conditions is fixed at eight (six for static runs), there is no limit on the number of load cases that may be formulated as combinations of the load conditions.

The vertical static load data input for load conditions I through III is prepared as part of the frame data. The self dead load of the members can be accounted for automatically by assigning unit weights to the frame materials. The unit weight specification causes the frame self weight to be added into the user-specified load condition.

Lateral static load data (for load conditions A, B and C) includes magnitudes and points of application for each rigid diaphragm at each story level. Each force acts on the complete diaphragm and gets distributed to each individual frame in accordance with the stiffness and location of the frame.

Lateral static loads may be due to wind or earthquake. A general static load pattern input option is also available. Individual lateral loads may also be applied to the individual joints of a frame.

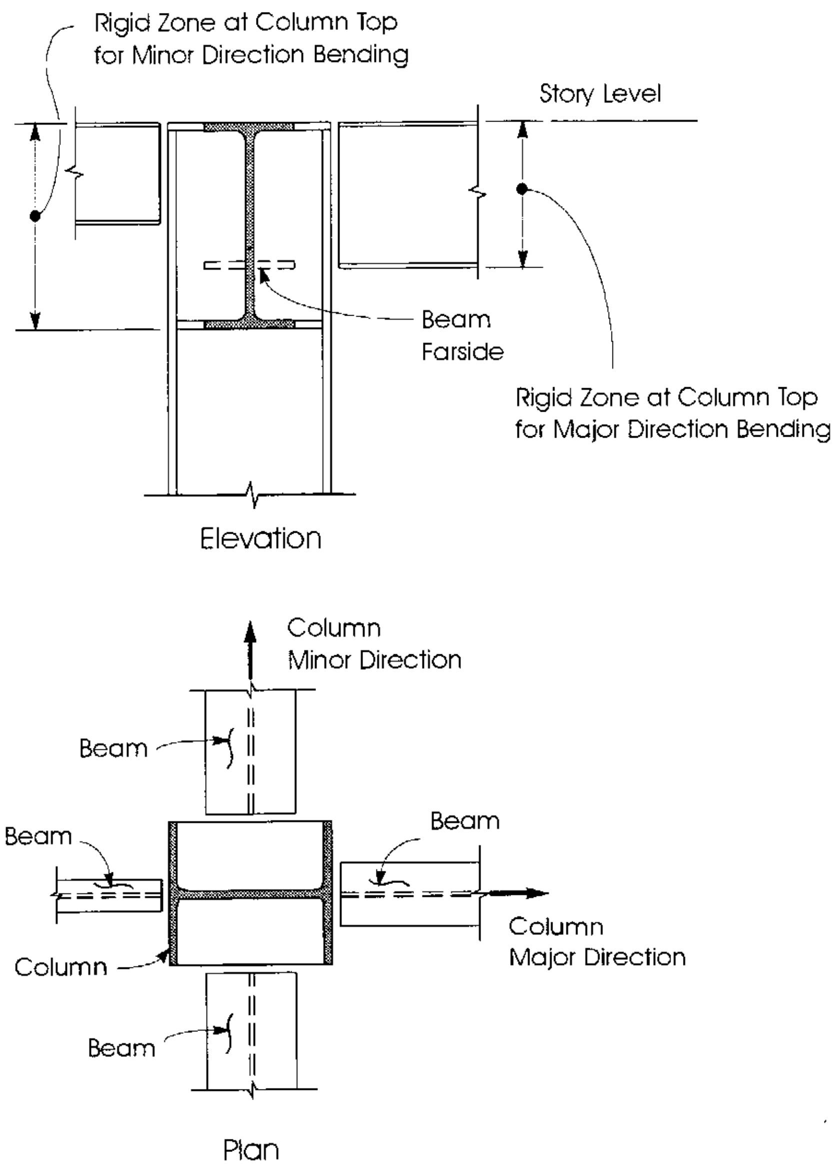
The wind loads can be calculated by the program based upon wind exposure data provided by the user, based upon the UBC [13], BOCA [15], ASCE 7-93 [16] or NBCC [17] Specifications.

The equivalent lateral seismic load data can also be generated by the program based upon the UBC [13], ATC [14], BOCA [15] or NBCC [17] requirements.

Lateral dynamic seismic ground excitation loading may either be in the form of an acceleration response spectrum or in the form of an acceleration time history input. This data is, in general, provided by the engineer responsible for the geotechnical evaluation of the building site.

Time history data in digitized form for well-known earthquakes is available from the National Information Service/Earthquake Engineering, University of California, Berkeley, California. Response spectra can be generated from time history data.

Vertical dynamic analysis options are not available.



Column Rigid Zone
Figure III-2

F. Rigid Zone

All structural members have finite dimensions. In many structures the dimensions of the members are large and have a significant effect on the stiffness of the structure. An analysis based upon a centerline to centerline geometry, in general, overestimates the deflections. Also, engineers prefer to have member forces output at the support faces. The rigid zone is defined as the distances from the joints to the face of the supports. See Figures III-2 and III-3. The beam and column member forces are output at the outer ends of the rigid zones. The clear length of the member is given by

$$L^* = L - (Ri + Rj)$$

where L is the actual element length.

In general it is assumed that no bending or shear deformations occur in beams and columns within the rigid zone. However, it has been found that an analysis based upon rigid zone lengths to the outer face of the supports can underestimate the deflections of the structure. A rigid zone reduction factor is used to reduce the lengths of the rigid zones, thereby compensating for some of the deformations that do exist in the zone bounded by the finite dimensions of the joint. The flexible element length is then given by

$$L^* = L - (1-z)(Ri + Rj)$$

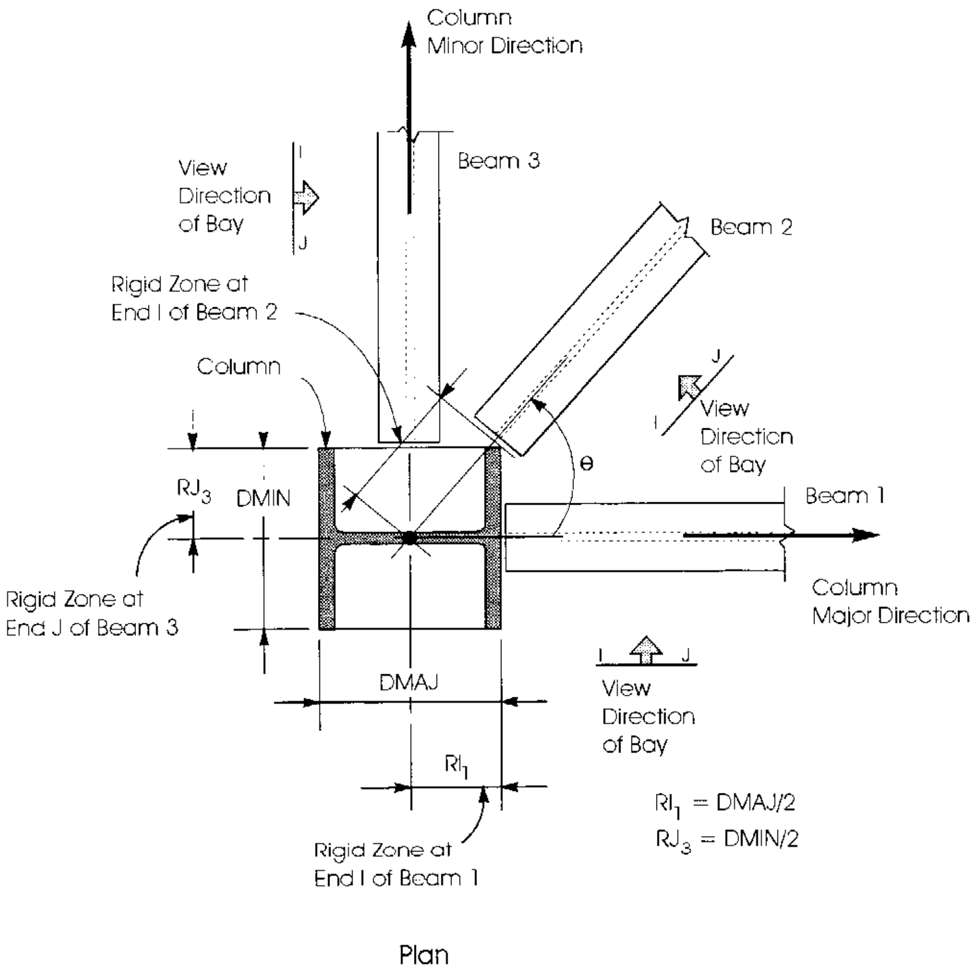
Irrespective of the value of z, **the member forces will always be output at the outer faces of the rigid offsets.**

The column element has four rigid zones, **RTMAJ, RTMIN, RBMAJ** and **RBMIN,** corresponding to the major and minor directions of the top and bottom ends of the column respectively. These zones are set by the depth dimension of the beams that frame into the columns. See Figure III-2.

A column will have a top rigid zone in the major direction, **RTMAJ,** only if beams framing into the top of the column are in a direction that has a component parallel to the column line major direction.

For each such beam, the program calculates the quantity

$$R = DB * \cos^2\theta$$



Beam Rigid Zone
Figure III-3

where DB is the depth of the beam below the diaphragm, and θ is the angle between the direction of the beam and the column major direction.

The value of **RTMAJ** is then the largest of the R values thus obtained.

Similarly, **RTMIN,** the column top rigid zone for the column minor direction, is obtained by calculating the quantity

$$R = DB * \sin^2\theta$$

for each beam that frames into the top of the column. The value of RTMIN is then the largest of the R values thus obtained.

The rigid zones for the column bottom end **RBMAJ** and the **RBMIN** are defined in a similar fashion except that the beams framing into the bottom end of the column and the corresponding beam depths above the diaphragm are examined.

It is obvious that if a column has beams framing only in a direction parallel to the column major direction, it will have no minor direction rigid zones and vice versa.

Also, beams with zero depth values do not make any contributions to the column rigid zones.

The beam element has two rigid zones, **RI** and **RJ**, corresponding to the I and J ends of the beam. These zones are set by the major or minor dimensions, respectively, of the columns that exist at the ends of the beams.

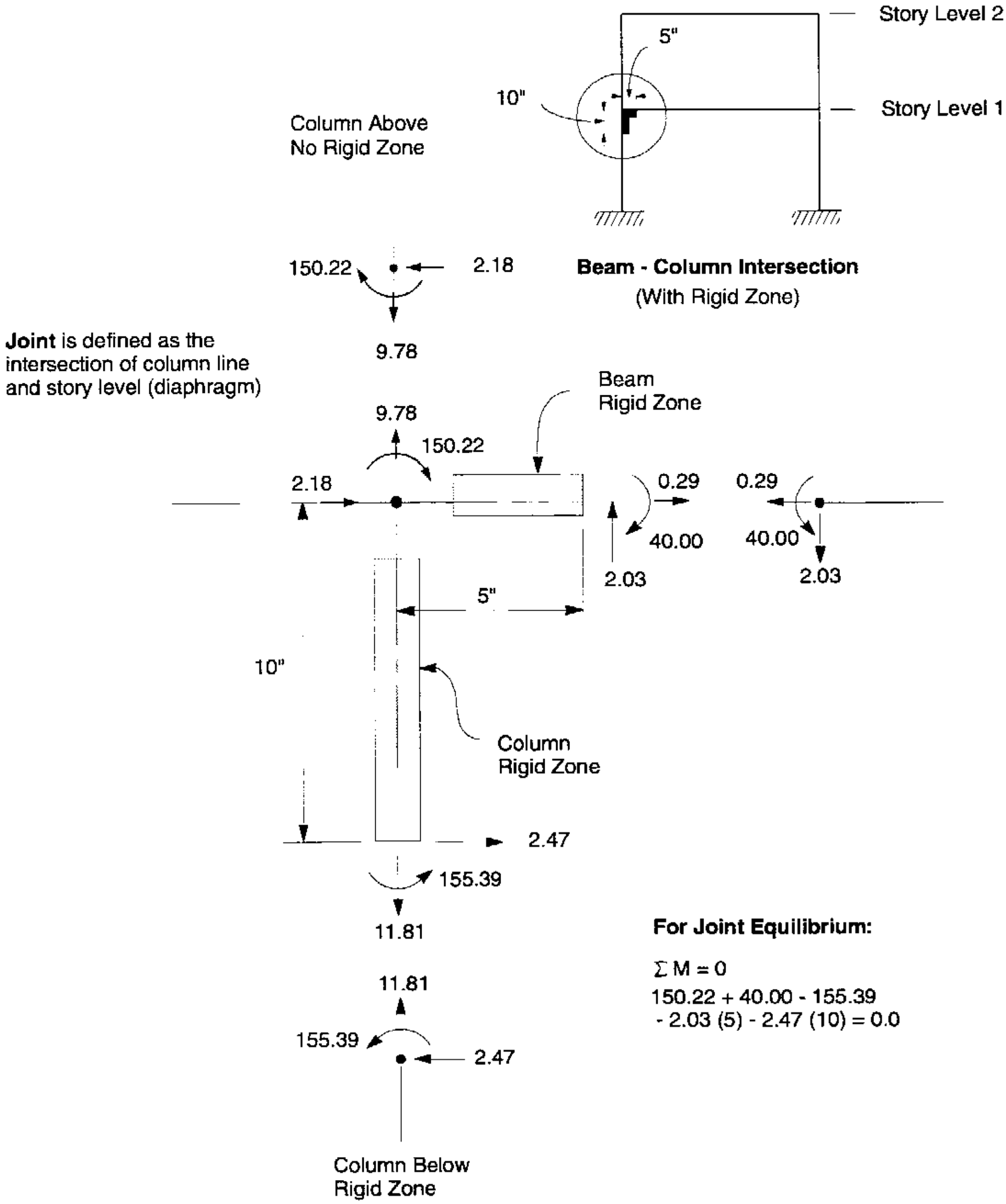
In calculating **RI**, the two column elements above and below the End I of the beam are examined. The quantities

$$R = \frac{DMAJ \cos^2\theta}{2} + \frac{DMIN \sin^2\theta}{2}$$

are then established for each of the column elements, where **DMAJ** and **DMIN** are the major and minor dimensions of the column element, and θ is the angle between the direction of the beam and the column major direction.

The value of **RI** is then established as the largest of the two R values thus obtained.

Similarly, **RJ** is established by examining the column above and below the beam at End J.



Joint Static Equilibrium/Beam-column Intersection (2D Analysis)
Figure III-4

The rigid zones for the beam are active only in the major direction bending of the beams.

It is obvious that if the column dimensions are defined as zero, no rigid zones will be generated for the corresponding beams.

There are no rigid zones generated for the brace or panel elements, and these elements are not included in determining the rigid zones for the column and beam elements.

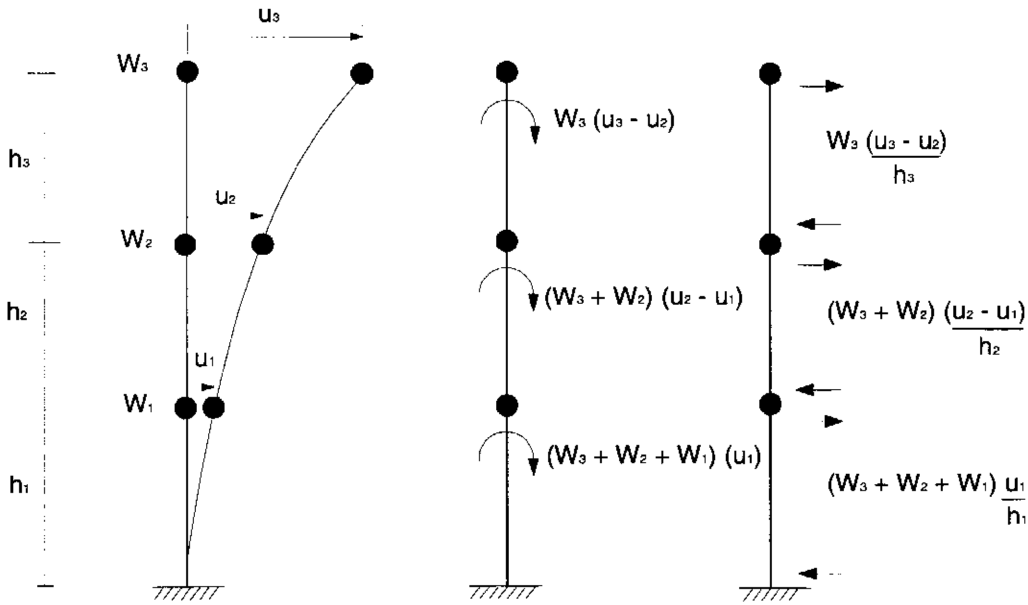
G. Statics Check

Results from all static load conditions must satisfy statics. However, as the column and beam moments are evaluated at the outer faces of the rigid zones of the corresponding members, joint moment equilibrium is not readily obvious. Checking joint statics involves transformations of the column and beam moments to the point of intersection of the corresponding column line and story level. Statics will be satisfied once all moments are transformed to this common point. It is clear that the beam and column shears and the dimensions of the rigid zones will need to be a part of the moment equilibrium equations. An illustration of joint static equilibrium is presented in Figure III-4.

Results from all dynamic load conditions, in general, will not satisfy statics. In the response spectrum analysis, the analysis technique involves the summation of the modal components by methods which cause the signs of the resultant parameters to be lost. In the time history analysis absolute maxima of the displacements and member forces are printed, thereby causing the sign to be lost. Besides, these maxima may not exist at the same instant in the analysis time span.

H. P-Delta Effects

In the analysis of building systems subjected to lateral displacements, the movement of the structural mass to a deformed position generates second-order overturning moments. This second-order behavior has been termed the P-Delta effect since the additional overturning moments on the building are equal to the sum of story weights "P" times the lateral displacements "Delta."



Displaced Position
of Story Weights

Additional Overturning
Moments or Lateral loads

P-Delta Effects
Figure III-5

The P-Delta phenomenon has always been an area of analytical concern for structural engineers. Traditional methods for including P-Delta effects in the analysis of multistory buildings have been tedious and time-consuming. Such methods have been based upon iterative techniques, which offer limited accuracy, and in general have been used only in two-dimensional static analysis environments.

In the solution algorithm of ETABS, the P-Delta effects are incorporated into the basic formulation of the structural stiffness matrix as a geometric stiffness correction. In this manner, the effects are exactly represented in all aspects of the structural analysis without any additional computational effort or iterative algorithms.

The additional displacements and overturning moments due to the softening effects of the P-Delta corrections automatically manifest themselves in analysis, so that three-dimensional overturning equilibrium in the final deformed configuration is immediately satisfied for the complete building.

The algorithm also accounts for the lengthening of the structural time periods due to P-Delta effects and changes in the corresponding mode shapes that finally affect the dynamic displacements and member forces of the structure.

The comparison of a set of analyses with and without P-Delta effects will illustrate the magnitude of the P-Delta effects on the analysis.

A well-designed building usually has well-conditioned level by level stiffness/weight ratios. For such structures, P-Delta effects are usually not very significant. The changes in the displacements and member forces are less than 10%.

However, if the weight of the structure is high in proportion to the lateral stiffness of the structure, the contributions from the P-Delta effects are highly amplified and under certain circumstances, can change the displacements and member forces by 25 or 30%. Excessive P-Delta effects will eventually introduce singularities into the solution, indicating physical structure instability. Such behavior is clearly indicative of a poorly designed structure that is in need of additional stiffness.

Reference [8] provides detailed information associated with the P-Delta formulation of ETABS.

The net effect of the geometric stiffness correction is to replace the overturning moment with an equivalent lateral force couple, as shown in Figure III-5.

Therefore, the overturning moment of the applied loads will equal the base overturning only when Σ P Δ is added to the overturning moment, where P is the weight of a story and Δ is the corresponding story lateral displacement, and, the structural base shear will add up to the applied loads only if the force given by $P_n\ \Delta_n/h_n$ is subtracted from the sum of the lateral column shears, where P_n, Δ_n and h_n are the cumulative weight, displacement and story height of the bottom level of the frame.

ETABS obtains the story weights, P, from the specified story masses and the story displacements, Δ, are the computed elastic calculated. It allows for a multiplier for the P-Delta effect to account for factored story weights and/or inelastic displacements.

I. Center of Rigidity and Accumulated Center of Mass

The center of rigidity has been traditionally defined as the location of the resultant of the rigidities of planar single story lateral resisting elements located arbitrarily in plan. The definition has worked well with single story structures, however, it is approximate at best in its extrapolation into multistory buildings idealized as a stack of decoupled single story segments.

In the general three dimensional analysis of a building, where the behavior of the structure is completely coupled in plan as well as through the height of the structure, the center of rigidity requires a broader definition.

In this broader redefinition, when translational lateral loads are applied at the center of rigidity of a particular rigid floor diaphragm, of any three dimensional structure, the displacements of that diaphragm will contain only translational (X and/or Y) components, with no rotation, provided that there are no loads applied on any of the other diaphragms. However, the resulting displacements of the other diaphragms, in general, will contain translational as well as rotational components.

The evaluation of the coordinates of the center of rigidity involves a static analysis of the whole structure for three independent load conditions, two translations and one rotation, for each diaphragm. This is done automatically in ETABS and the resulting center of rigidity for each diaphragm is reported.

The accumulated center of mass for a particular diaphragm represents the location of the resultant of the masses of all the diaphragms (with the same diaphragm identification number) from all levels above and including the particular diaphragm. The offsets between the center of rigidity and the accumulated center of mass are a measure of the torsional eccentricity of the diaphragm at a particular level.

J. ETABS Nonlinear Analysis

The method of nonlinear dynamic analysis used in ETABS is an extension of the method developed by Wilson [11,12]. The method is extremely efficient and is designed to be used for structural systems with a limited number of predefined nonlinear elements. A short description of the method follows.

The dynamic equilibrium equations of an elastic structure with nonlinear elements subjected to ground motions can be written as:

$$\mathbf{M}\,\ddot{\mathbf{u}}(t) + \mathbf{C}\,\dot{\mathbf{u}}(t) + \mathbf{K}_E\,\mathbf{u}(t) + \mathbf{R}_N(t) = -\mathbf{M}\,\ddot{\mathbf{u}}_g(t)$$

where $\mathbf{M}$ is the mass matrix, $\mathbf{C}$ the viscous damping matrix, $\mathbf{K}_E$ the stiffness matrix contribution from the elastic elements, $\mathbf{R}_N$ the nodal forces from the nonlinear elements, $\mathbf{u}$, $\dot{\mathbf{u}}$ and $\ddot{\mathbf{u}}$ are the relative displacements, velocities and accelerations with respect to ground, and $\ddot{\mathbf{u}}_g$ the ground acceleration.

This equation can be rewritten as:

$$\mathbf{M}\,\ddot{\mathbf{u}}(t) + \mathbf{C}\,\dot{\mathbf{u}}(t) + \mathbf{K}\,\mathbf{u}(t) = -\mathbf{M}\,\ddot{\mathbf{u}}_g(t) - [\,\mathbf{R}_N(t) - \mathbf{K}_{Neff}\,\mathbf{u}(t)\,]$$

where $\mathbf{K} = [\,\mathbf{K}_E + \mathbf{K}_{Neff}\,]$. $\mathbf{K}_{Neff}$ being the effective linear stiffness of the nonlinear elements. It should be noted that the above equation is valid for any assumption of the effective linear stiffness of the nonlinear elements.

Using standard techniques the above equation can be written in modal form as:

$$\mathbf{I}\,\ddot{\mathbf{y}}(t) + \mathbf{\Lambda}\,\dot{\mathbf{y}}(t) + \mathbf{\Omega}^2\,\mathbf{y}(t) = -\,\mathbf{F}_g(t) - \mathbf{F}_N(t)$$

where $\mathbf{I} = \mathbf{\Phi}^T\mathbf{M}\mathbf{\Phi}$ is the identity matrix, $\mathbf{\Phi}$ being the mode shapes, $\mathbf{\Lambda} = \mathbf{\Phi}^T\mathbf{C}\mathbf{\Phi}$ is the modal damping matrix which is assumed to be diagonal, $\mathbf{\Omega}^2 = \mathbf{\Phi}^T\mathbf{K}\mathbf{\Phi}$ is a diagonal matrix of structural frequencies squared, $\mathbf{F}_g$ is an array of modal input loads given by $\mathbf{\Phi}^T\mathbf{M}\,\ddot{\mathbf{u}}_g(t)$, and **FN** is an array of modal forces from the nonlinear elements and is given by $\mathbf{\Phi}^T[\,\mathbf{R}_N(t) - \mathbf{K}_{Neff}\,\mathbf{u}(t)\,]$.

It should be noted that unlike linear dynamic analysis the above modal equations are not decoupled. In general the $\mathbf{F}_N$ terms will couple these equations.

It is important to recognize that the above method of solution is dependent on being able to represent the nonlinear forces as modal forces. Therefore, firstly, the nonlinear degrees of freedom should also be mass degrees of freedom and secondly, sufficient number of modes should be used to capture the deformation in the nonlinear elements completely.

Also it should be noted that the assumption of proportional modal damping is being made with respect to a stiffness matrix which includes the effective stiffness from the nonlinear elements. If non-zero modal damping is to be used then the effective stiffness specified for these elements is important. The effective stiffness should be selected such that the modes for which these damping values are specified are correctly evaluated. In general it is recommended that either the initial stiffness of the element be used as the effective stiffness or the secant stiffness obtained from tests at the expected value of the maximum displacement be used. Initially open gap elements and damper elements in general should be specified with zero effective stiffness.

The above modal equations are solved iteratively by ETABS using closed form integration assuming linear variation of the right hand side of the equation during a time step [7]. The iterations are carried out until the change in the right hand side of the equation, expressed as a ratio, is below a certain user specified tolerance value.

The program has a built-in substepping algorithm. Whenever convergence in a step is not achieved within a user specified maximum number of iterations the program bisects the integration step. This substepping continues until convergence is reached or the step becomes too small for numerical accuracy in which case the execution is terminated. This procedure is very effective in capturing the effects of sudden changes in stiffness that may occur during the solution, such as in the closing or opening of gap elements or in the slipping of friction elements.

The program also allows for underrelaxation of the nonlinear element forces. This allows change in nonlinear force from one step to another to be gradually applied, a small fraction at a time, in successive iterations. This is also a user controlled variable. Generally no underrelaxation is necessary for convergence, however, in some instances, especially when nonlinear dashpot dampers are present and modes are highly coupled by the nonlinear elements, the iteration process may become unstable. Using an underrelaxation factor and allowing more iterations per step usually solves the problem.

Chapter **IV**

ETABS Elements and Their Applications

This chapter describes some of the important characteristics of the various linear and nonlinear elements available in ETABS to model the building's structural system. These characteristics must be recognized by the user in order to correctly use, understand and apply the ETABS program. Modeling of complex systems using the various elements, general points of caution associated with the use of each element and the limitations of the program are also discussed in this chapter.

A. Structural Elements of ETABS

The structural elements available in ETABS are column, beam, floor, brace, panel, link and spring (grounded). The following are some important characteristics of the elements.

a. Column Element

- The column always exists vertically at any level on a column line. The length of the column element is the story height.

- The top or bottom of columns may be either continuous or pinned in the major or minor directions of bending.

- Columns may be prismatic or nonprismatic from floor level to floor level.

- The column element may have rigid zones for stiffness corrections. The dimensions of these zones are automatically evaluated by the program. The evaluation methodology is described in Chapter III. The member forces are output at the outer faces of the rigid zones in the column local coordinate system. See Figure VI-3 for the definition of the coordinate system.

- The column element formulation includes the effects of axial, shear, bending and torsional deformations.

- The column is connected to a joint at each story level. The diaphragm connectivity of the column end is determined by the diaphragm connectivity of the corresponding joint.

- The self weight of the column element is based upon the story-to-story length of the column and is lumped equally at each end of the column.

- The self mass of the column element is based upon the story-to-story length of the column and is lumped equally at each end of the column.

b. Beam Element

- A beam may exist at any level in a predefined bay between two column lines. Beams are in general horizontal, however, any one end of the beam may be dropped a level to model sloping beams.

- Beam connections at either end may be either fixed or pinned in the major or minor directions of bending. Partially restrained connections are also possible.

- Beams may be prismatic or nonprismatic.

- The beam element may have rigid zones for stiffness corrections. The dimensions of these zones are automatically evaluated by the program. The evaluation methodology is described in Chapter III. The member forces are output at the outer faces of the rigid zones in the beam local coordinate system. See Figure VI-4 for the definition of the coordinate system.

- The beam element formulation includes the effects of axial, shear, bending and torsional deformations. In beams the axial deformation effects and minor direction shear and bending deformation effects are only activated if the ends of the beam connect to different diaphragms or at least one end is disconnected from the diaphragm. Diaphragm connectivity of the ends of the beam is determined by the diaphragm connectivity of the corresponding joint.

- The self weight of the beam is applied as a uniform or variable load along the clear length of the beam proportional to the beam cross-section.

- The self mass of the beam is based upon the clear length of the beam and is lumped equally at each end of the beam.

- Any participation of the structural floors in the bending of the beams must be reflected in the properties of the beams (T-beam or L-beam action) provided by the user, if it is to be included in the analysis.

- Vertical loading may be applied as uniform, trapezoidal or point loads on the beam span in the major plane of the beam. Also, vertical uniform loads on the floor elements are automatically converted to span loads on adjoining beams, thereby automating the tedious task of transferring floor tributary loads to the floor beams without explicit modeling of the secondary framing.

c. Floor Element

- A floor element may exist at any level in a predefined floor bay between three or four column lines. Floors are in general horizontal, however, any one edge of the floor may be dropped a level to model sloping roof diaphragms or ramped floors.

- The floor element may have equally spaced secondary beams specified in any direction. This information is used to automatically transfer vertical surface tributary loads to adjoining beams. See Figure V-18.

- The floor element forces are output at the joints as nodal forces in the floor local coordinate system. See Figure V-12 for the definition of the coordinate system.

- The floor element formulation includes only membrane stiffness. However, orthotropic properties are possible. The floor element stiffness is only activated if all floor joints of an element are not connected to the same diaphragm.

- The diaphragm connectivity of the nodes of the floor is determined by the diaphragm connectivity of the corresponding joints.

- The self weight of the floor element is assumed uniformly distributed on the floor and gets applied either to the floor joints or as beam span loads on adjoining beams.

- The self mass of the floor is assumed uniformly distributed on the floor and gets applied as lumped masses at the floor joints.
- Uniform vertical loading may be applied to the floor element. This vertical load is either lumped at the floor joints or applied as span loads on adjoining beams.
- Uniform lateral loading may be applied to the floor element. This lateral load is lumped at the floor joints.

d. Brace Element

- The brace element may exist in any vertical plane between any two column lines (consecutive or nonconsecutive) between any two floors (consecutive or nonconsecutive).
- The ends of the brace may be continuous or pinned.
- The brace element must be prismatic from floor level to floor level.
- The brace has no options for rigid zones for stiffness corrections. The member forces are output at the floor levels in the brace local coordinate system. See Figure VI-5 for the definition of the coordinate system.
- The brace element formulation includes the effects of axial, shear, bending and torsional deformations.
- The diaphragm connectivity of the ends of the brace is determined by the diaphragm connectivity of the corresponding joint.
- The self weight of the brace is based upon the story to story length of the brace, and is lumped equally at each end of the brace.

e. Panel Element

- The panel element can exist between any two column lines (consecutive or nonconsecutive), between any two consecutive levels.
- Panels may be modeled with piers at the two ends. In this case the panel is inserted in the model at the center of the piers. See Figure V-14. Piers may be modeled as equal to thickness of the panel, allowing an easy method of varying the panel insertion point in the model.

- Panels in the lowermost story are assumed fixed at the bottom.

- The panel element must be prismatic from floor level to floor level.

- The panel stiffness is based upon a length equal to the story height with no rigid zones due to the depths of the beams that may be framing into the panel.

- Panels may share column lines and assemblages of panel elements (at a particular level) may be defined to form T-, L- or C-shaped "walls," and other complex "wall" configurations. The program will produce integrated moments, shears and axial forces for the "wall" defined by the panel assemblage at the center of gravity of the "wall", or forces for planar segments of such assemblages may be produced.

- The section forces are output at the floor levels in the local coordinate system of the panel assemblage. See Figure VI-6 for the definition of the coordinate system.

- The panel element is based upon an isoparametric finite element membrane formulation with incompatible modes. The end piers provide additional axial, torsional and out of plane bending stiffness at their respective ends.

- The formulation includes in-plane rotational stiffness components. This allows the panel to connect to the column, beam and brace elements and achieve moment continuity, without any artificial elements such as rigid beams.

- The diaphragm connectivity of the panel is determined by the diaphragm connectivity of the corresponding joints of the panel. If a joint of a panel at a particular level is disconnected from the diaphragm, the panel will be disconnected from the diaphragm at the corresponding end.

- The self weight of the panel is based upon the story to story height of the panel and the weight is lumped equally at the four joints of the panel.

- The panel element has been designed to be used in the modeling of shear wall systems where the primary mode of bending is vertical, i.e. associated with horizontal shears.

- The panel element may be used to model situations where the bending deformations are associated with vertical shears. The rigid diaphragm constraint will make the element stiff in bending but the shear deformations are captured accurately. However, the section shear forces and moments output will still correspond to horizontal sections.

- Caution is advised when very slender column-like elements are modeled using the panel element. Any column or beam elements framing into these slender panels will not receive correct fixity at their ends. Also, the slender panel element is too stiff for flexural deformation. These slender elements are better modeled as columns.

f. Link Element

- The link element may exist between any two joints.
- The ends of the link may be continuous or pinned.
- The link element is defined with total stiffness properties rather than section properties.
- Link elements can take linear or nonlinear properties.
- The link member forces are output at the floor levels in the link local coordinate system. The link element local coordinate system is defined as a column element if the link is vertical or if it has zero length, it is defined as a beam element if the link is horizontal and it is defined as a brace element if the link is defined between two different floors and not connected to the same column line.
- The diaphragm connectivity of the ends of the link is determined by the diaphragm connectivity of the corresponding joint.
- The link element has no self weight or self mass.

g. Spring Element

- The spring element may exist at any joint, except at the baseline.
- The spring element is assumed connected to ground at the other end.
- The spring element properties are defined with three dimensional translational and rotational spring constants.
- Spring elements can take linear or nonlinear properties.
- The spring member forces are output at the floor levels in the column local coordinate system.

- The diaphragm connectivity of the spring is determined by the diaphragm connectivity of the joint.
- The spring element has no self weight or self mass.

B. Nonlinear Elements of ETABS

The nonlinear elements in ETABS can be used either as springs (grounded) or links. The ETABS nonlinear elements include a uniaxial damper element, a uniaxial gap element, a uniaxial plasticity element, a biaxial hysteretic isolator element, and a biaxial friction pendulum element.

The nonlinear properties of these elements are only used during nonlinear dynamic time history analysis. Some of these elements allow an effective stiffness to be specified for use in static analysis, dynamic response spectrum analysis and linear time history analysis. Also some elements allow specification of an effective damping ratio which is used in the evaluation of composite modal damping ratios that are then added to the specified modal damping for use in the dynamic response spectrum analysis and the linear time history analysis.

Nonlinear analysis using these elements can be started using initial force conditions from a user specified static load combination. It is reiterated that the static analysis is done using the effective stiffness of the elements.

Some important characteristics of the elements are described below.

a. Uniaxial Damper

- The uniaxial damper can be specified in either the axial or the major shear or the minor shear direction. See Figure V-15.
- The damper force varies as $\mathbf{F} = \mathbf{C}\,\mathbf{V}^{\alpha}$, where **V** is the velocity of deformation in the element, **C** is a coefficient and α is an exponent which is normally between 0.2 and 2.0.
- The damper element does not allow for an effective stiffness to be specified for linear analysis.

- For linear response spectrum and time history analysis the program uses the coefficient **C** to obtain a modal damping value, assuming α is one and the damping is proportional.

b. Uniaxial Gap

- The uniaxial gap element can be specified in either the axial or the major shear or the minor shear direction. See Figure V-15.
- The gap element has linear elastic stiffness when closed and zero stiffness when open.
- The gap element allows for a negative initial opening to be specified to model precompression.
- The gap element allows for an effective stiffness to be specified for linear analysis.
- The gap element does not allow for an effective damping ratio to be specified for linear dynamic analysis.

c. Uniaxial Plasticity Element

- The uniaxial plasticity element can be specified in either the axial or the major shear or the minor shear direction. See Figure V-15.
- The plasticity element is based on the hysteretic behavior proposed by Wen [18] and allows for parameters to vary post yield stiffness and the shape and width of the hysteresis loops.
- The plasticity element allows for an effective stiffness to be specified for linear analysis.
- The plasticity element allows for an effective damping ratio to be specified for linear dynamic analysis to account for its hysteretic behavior.

d. Biaxial Hysteretic Isolator Element

- The biaxial hysteretic isolator element has stiffness in axial and the two shear directions. The axial direction has linear stiffness and the shear directions have hysteretic behavior. See Figure V-15.

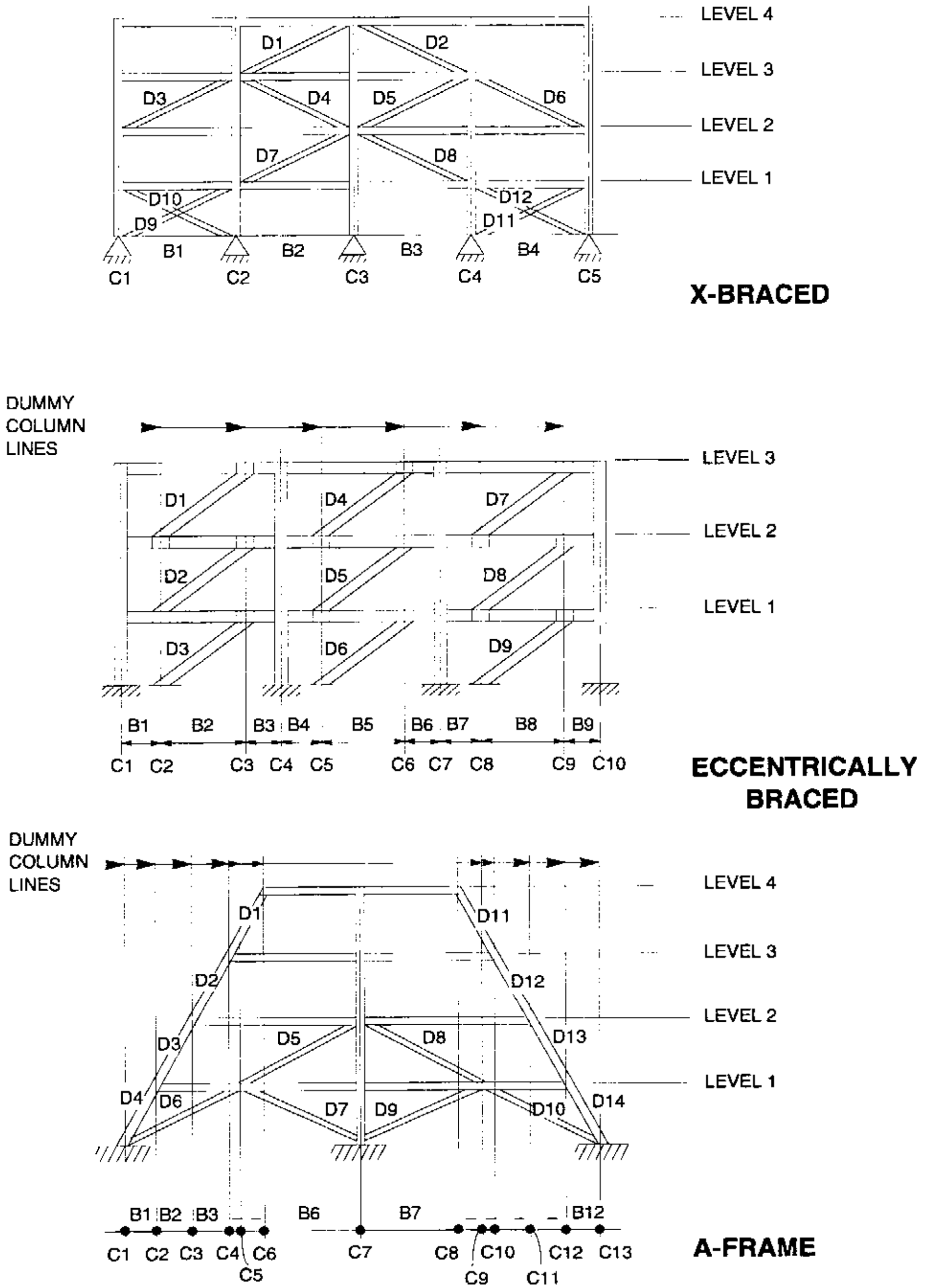
- The isolator element is based on the hysteretic behavior proposed by Wen [18], and Park, et al. [19] and recommended for base isolation analysis by Nagarajaiah, et al. [20].
- The isolator element allows for an effective stiffness to be specified for linear analysis for the two shear directions.
- The isolator element allows for an effective damping ratio to be specified for linear dynamic analysis to account for hysteretic behavior in the shear directions.

e. Biaxial Friction Pendulum Isolator Element

- The biaxial friction pendulum isolator element has stiffness in axial and the two shear directions. The axial direction has linear stiffness and the shear directions have hysteretic behavior due to friction and a post slip stiffness due to the finite pendulum radius of the slipping surfaces. See Figure V-15.
- The friction portion is based on the hysteretic behavior proposed by Wen [18], and Park, et al. [19] and recommended for friction base isolation analysis by Nagarajaiah, et al. [20]. The pendulum part is as recommended by Zayas, et al.[21].
- The shear forces due to friction and due to the pendulum action are dependent on the axial load in the element. The element allows for the axial load to vary due to overturning effects. Tension in the element causes the shear forces due to both the friction part and the pendulum part to go down to zero. It is important for this element to start the nonlinear analysis from an initial state that includes gravity loads to correctly account for the axial load in the isolators.
- The isolator element allows for an effective stiffness to be specified for linear analysis for the two shear directions.
- The isolator element allows for an effective damping ratio to be specified for linear dynamic analysis to account for the friction loss in the shear directions.

C. Braced Frame Systems

By effectively using column lines for the definition of the frames as shown in Figure IV-1, complex bracing systems can be conveniently modeled. The



Modeling Braced Frame
Figure IV-1

"dummy" column lines are for geometric definition only and do not need any property assignments since the program default values are zero.

Null (zero) property assignments can be effectively used for dummy beams and columns in modeling structures with setbacks and other situations involving missing elements on the grid system used for the frame definition.

The user is reminded that using a rigid diaphragm model will result in zero axial forces in the beams. These forces may be important in braced frames. The user must disconnect some of the joints from the diaphragm to allow beams to carry axial loads. It is recommended that joints along only one of the column lines of the braced frame remain connected to the rigid diaphragm.

D. Modeling Shear Wall Systems

One of the most powerful aspects of the ETABS computer program is associated with the modeling of shear walls. The program has a special panel element that combines the versatility of the finite element method with the design requirement that the wall output be in terms of total moments and forces, instead of the usual finite element output of direct stress, shear stress and principal stress values.

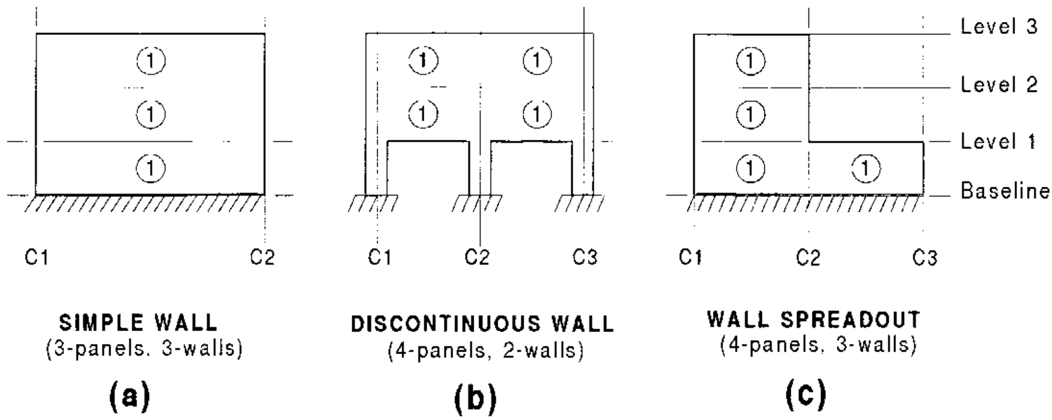
In the input, each panel element is assigned a wall ID number for output purposes.

All the panel elements **at a particular level with the same wall ID number** form an assemblage that defines the "wall" with the associated ID at the particular level.

The moments and forces produced by the program for the "wall" will be at the center of gravity of the "wall," in the "wall" local coordinate system.

The "wall" forces are obtained by consistently integrating the panel element internal forces of all the elements that form the "wall" at a particular level. The integration is implemented at the stress transformation level in order to be consistently carried through the static and dynamic analysis options.

All **connected** panel element systems are treated identically by the program internally, but output for a system may be requested in different forms. For the three-dimensional shear wall systems shown in Figure IV-4, the output may be as shown or in the form of three planar "walls" for (a) and four for (b). The latter output form is more useful for wall design in general and is required if the WALLER post processor is to be used.



Note: The circled numbers are wall identification numbers.

Planar Shear Wall Systems
Figure IV-2

a. Simple Wall

A simple cantilever shear wall is shown in Figure IV-2(a). This wall is modeled as a two column line system with three panel elements. No bays need to be defined (unless there is vertical loading). The column lines are used as basic reference lines to define the extent of the panels.

Each panel has been assigned a wall ID of 1, producing three "walls," namely, "wall" with ID 1 at Level 3, "wall" with ID 1 at Level 2, and "wall" with ID 1 at Level 1.

It is important to remember that a "wall" with a particular ID at a particular level is created by an assemblage of only those elements that exist at the particular level and have a particular wall ID.

Simple walls of this form may also be modeled with a single column line placed at the center of the wall. Column properties representing the wall axial area, shear area and moment of inertia are then assigned to the column line.

b. Discontinuous Wall

In this example, shown in Figure IV-2(b), shear walls from above terminate on a series of three columns below.

The model for this situation will require three column lines and four panel elements. No bays need be defined. Each panel has been assigned a wall ID of 1, producing two "walls," namely the two panel "wall" with ID 1 at Level 3, and the two panel "wall" with ID 1 at Level 2.

The column elements defined at Level 1 will receive continuity with the panel elements at the top ends without any further special modeling.

c. Wall Spread out

In this example, shown in Figure IV-2(c), shear walls from above spread out to a wider dimension below.

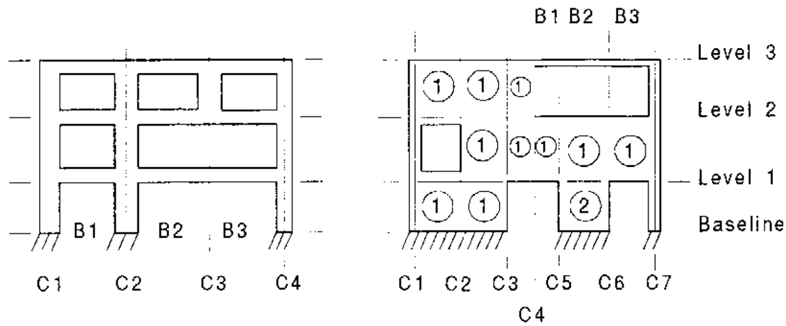
The model for this situation will require three column lines and four panel elements. No bays need be defined.

Each panel has been assigned a wall ID of 1, producing three "walls," one corresponding to each level, with a two panel "wall" at the bottom level.

d. Simple Pier-Spandrel System

A pier-spandrel system is simply a beam-column system in which the dimensions of the elements are large compared to the overall dimensions of the frame. Such systems are conveniently modeled with ETABS because the effects of the finite dimensions of the members on the stiffness of the frame are automatically considered. The pier-spandrel system shown in Figure IV-3(a) is modeled as a four column line three bay system, using default zero column property identifications to define missing elements.

The user should recognize, however, that attempts to describe small wall openings (such as those required for duct and pipe penetrations) as bays encompassed by wide columns and deep beams can introduce modeling difficulties into the analysis, resulting in unrealistic results. In general, it is better to ignore small openings. The loss of accuracy in neglecting them is much less than it is in trying to incorporate them into the structural model.



PIER-SPANDREL WALL (no-panels)

(a)

WALL W/RANDOM OPENINGS (11-panels, 4-walls)

(b)

Note: The circled numbers are wall identification numbers.

Planar Shear Wall Systems with Openings
Figure IV-3

e. Wall with Random Openings

For this example, shown in Figure IV-3(b), the model will require seven column lines, three bays and eleven panel elements.

The bays are required for defining the beams at Level 3.

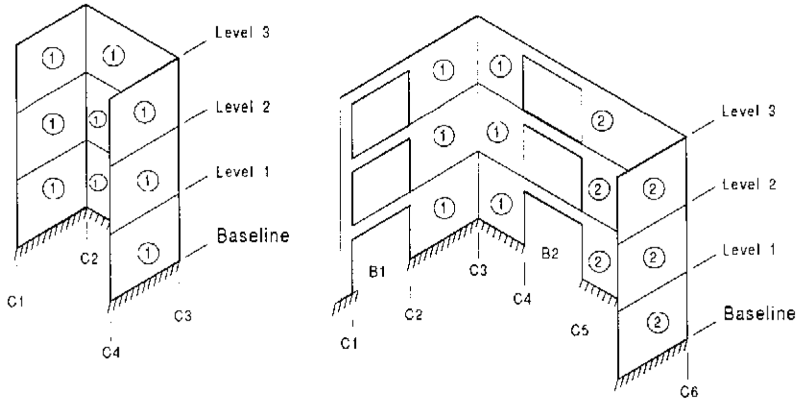
There are two "walls" at Level 1. The eleven panel elements produce a total of four "walls." Story levels identify locations of wall 1 at the three different levels.

Column elements are used for modeling the columns on column lines 1 and 7.

f. Three-Dimensional Shear Wall Systems

The panel element can be very effectively used to model three-dimensional shear wall systems such as C-shaped (or E-shaped) or L-shaped walls, as shown in Figure IV-4.

The three-story C-shaped wall shown in Figure IV-4(a) is modeled with nine panel elements that define three C-shaped "walls," one corresponding to each level.



C-SHAPED WALL
(9-panels, 3-walls)

(a)

L-SHAPED WALLS
W/BEAMS AND COLUMNS
(12-panels, 6-walls)

(b)

Note: The circled numbers are wall identification numbers.

Three-dimensional Shear Wall Systems
Figure IV-4

The output for each "wall" will consist of biaxial moments, torsion, axial force and biaxial shear forces about the center of gravity of the C-section, at each level.

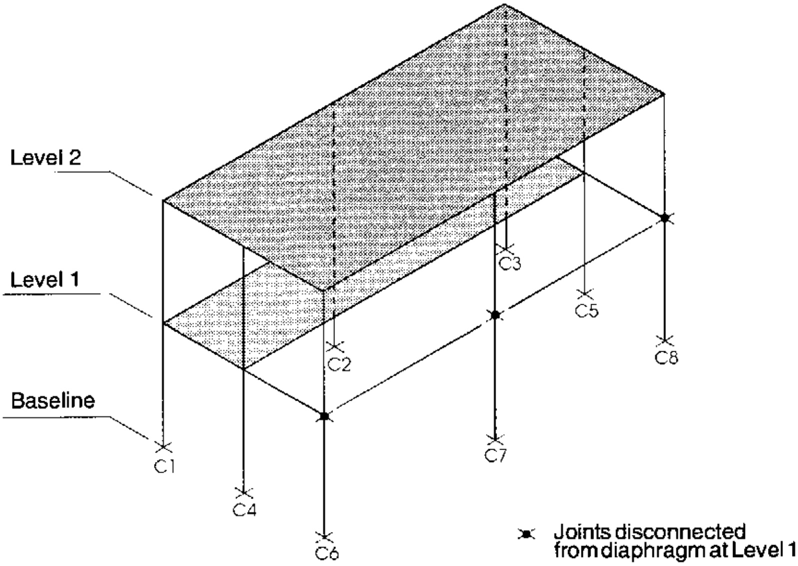
Figure IV-4(b) illustrates the use of the program options to model L-shaped walls. As shown in this example, beam elements may frame into the panel elements to form complete shear wall/frame systems.

There are two L-shaped walls, each modeled with six panel elements. There are two "walls" defined at each level, to obtain the moments, shears, axial force and torque for each L-section at each level separately, about the center of gravity of each L-section.

As shown, the model requires a total of seven column lines, two bays and twelve panel elements.

As mentioned earlier, for wall design convenience the wall identification numbers could be different for each planar segment. This would be a requirement for instance if design is to be made through WALLER.

Bay definition is required for beams and for those portions of walls which receive vertical loads.



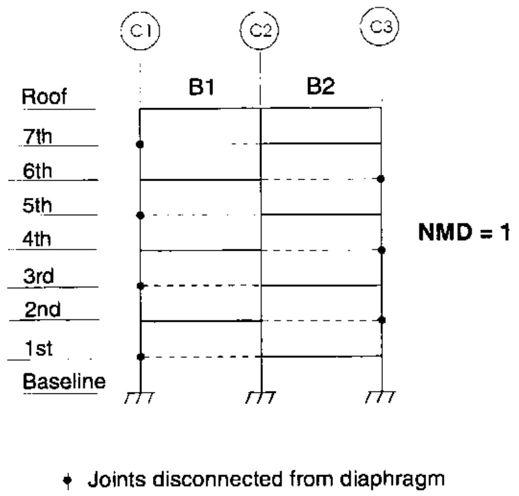
Mezzanine Modeling Using Diaphragm Disconnection
Figure IV-5

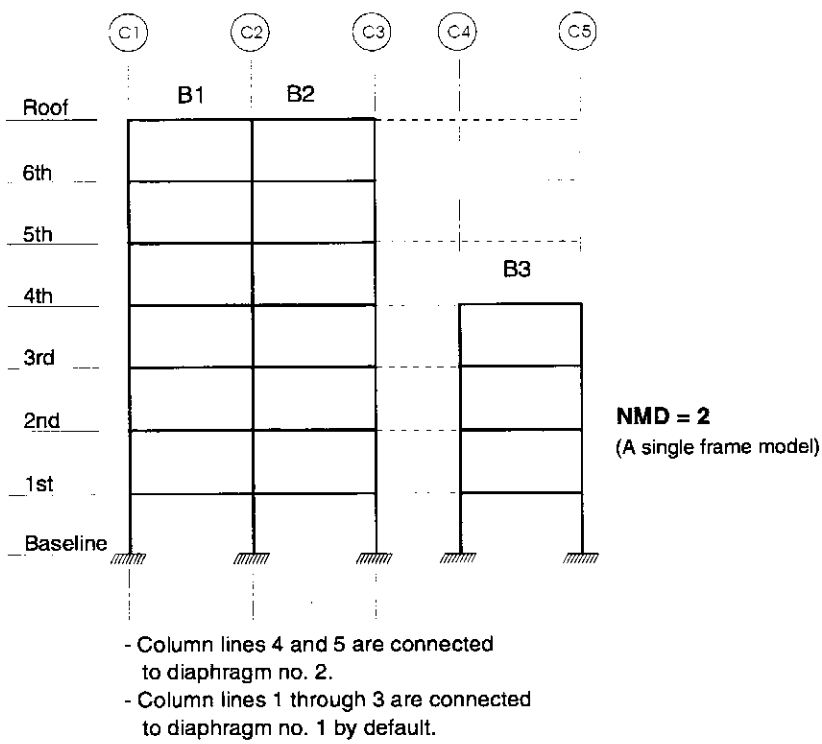
E. Multi-diaphragms and Diaphragm Disconnects

Every floor (or level) of the structure has at least one associated horizontal rigid floor diaphragm. All the joints of every frame by default connect to the rigid diaphragm number 1 of the corresponding floor level. There can be more than one floor diaphragm at any level and any joint at a particular level may be assigned to connect up with any one of the diaphragms. The joint may also be completely disconnected from all of the diaphragms.

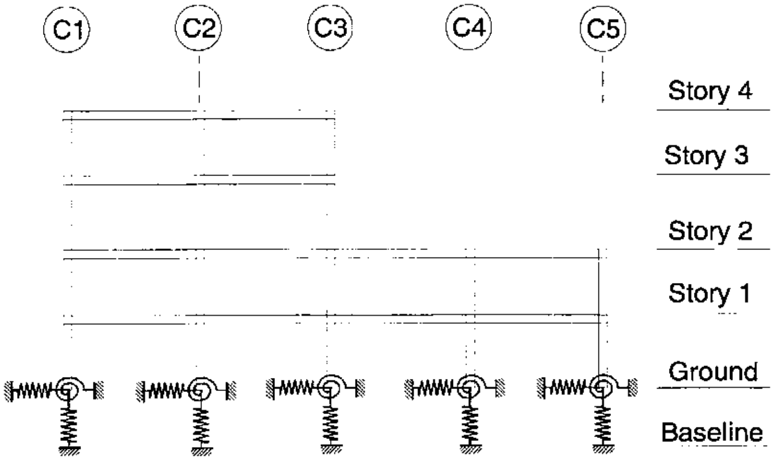
Creative use of the multiple rigid diaphragms along with diaphragm disconnects in conjunction with the flexible floor element gives the user the versatility to model multi-tower structures, mezzanines, sloping floors and roofs and stepped floor diaphragms.

Figure IV-5 illustrates the application of diaphragm disconnects in the modeling of a mezzanine level. In this example all joints on a column line of the frame do not attach to all the diaphragms associated with all the story levels of the frame. For example, column lines C6, C7 and C8 do not connect to the partial (mezzanine) floor diaphragm at Level 1. The joint disconnection option is used for modeling

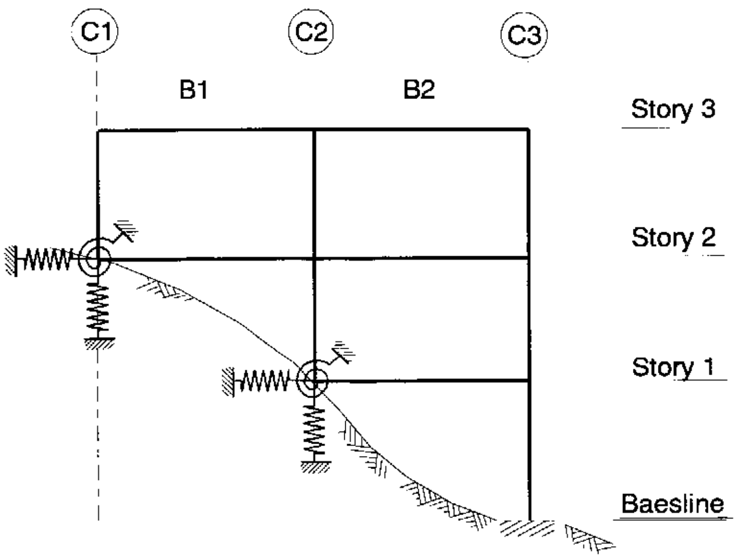




Partial and Multi-Diaphragm Modeling
Figure IV-6



Soil-Structure Interaction



Structure on a Slope

Foundation Flexibility Modeling
Figure IV-7

such situations. Activation of this option causes the joint to be free from the diaphragm. All elements attached to disconnected joints are disconnected from the corresponding diaphragm at the corresponding joint.

Another common use of the disconnection option is the modeling of structures with wood floor diaphragms. In such situations all the joints are disconnected at the level of the wood floor diaphragm and the lateral loads are assigned on a tributary basis to the various disconnected joints.

Disconnection of all the joints from the diaphragm essentially removes the rigid floor diaphragm. In such cases the automatic lateral static load generation options which apply loads to rigid diaphragms are meaningless.

Other examples of partial and multi-diaphragm modeling are shown in Figure IV-6.

E. Foundation Flexibility

Translational and rotational spring supports may be assigned at any joint of the frame. This option is very useful in modeling foundation flexibility and/or modeling support provided by ground to structures on slopes. This procedure can also be used very effectively to model base isolation systems.

By introducing a "dummy" story to the structure at the foundation level, as shown in Figure IV-7, it is possible to introduce soil springs to model foundation flexibility. Similarly, the introduction of soil springs at the different story levels can be used to model structures on slope. In each of these situations the analytical results tend to be very sensitive to the input restraint conditions and it is important that a practical solution to such problems be based on several analyses, such that the sensitivities of the restraint parameters are evaluated and their relative importance is established.

Chapter V

ETABS Input Data File

This chapter details the format of the input data file that numerically defines the structural geometry and loading associated with the building that is to be analyzed using the computer program ETABS. Optionally, the input data file can be prepared through ETABSIN, the Windows based graphic model generator, which will automatically generate the input data file from the model in the required format.

The user must read Chapters III and IV and understand the assumptions of the ETABS program before proceeding with the data preparation described in this chapter.

There are basically eleven data sections associated with the ETABS input. A summary of the data setup is shown in Figure V-1.

All input data for ETABS is prepared in free format. In other words, the data on a particular line does not have to correspond with prespecified column locations. The data is input as a string of numbers which are separated by one or more blanks. It is important to enter all items even if they are zero; however, trailing zeros on any data line need not be input. No data line may be more than eighty characters in length. Also, the data file should not contain any special formatting characters that are generated by some editors while preparing a data file.

Decimal points for whole floating point numbers are not necessary. For example, the number (6.0) may be entered as (6). Scientific exponential notation is also allowed. For example, the number 1.5E10 is read as 1.5×10^{10}.

Alphabetic characters may be uppercase or lowercase.

Any line having a dollar sign ($) in Column 1 is treated as a comment line and ignored by the program. The $ sign may also be used in any other column on any data line. In such cases entries to the right of the $ sign will be treated as comment data and ignored by the program. This option allows the user to effectively comment the data as it is being prepared.

Simple arithmetic statements are possible when entering floating point real numbers in the free format fields. The following types of operators can be used:

+	for addition
-	for subtraction
/	for division
*	for multiplication
P	for raising to the power of

The operators are applied as they are encountered in the scan from left to right.

The following are examples of data entries that are possible and how they are interpreted by the program:

11.92*12	is evaluated as 11.92×12
7.63/386.4	is evaluated as $\frac{7.63}{386.4}$
3P.5	is evaluated as $3^{.5}$
150P1.5*33	is evaluated as $150^{1.5} \times 33$
6.66-1.11*7.66/12.2	is evaluated as $\frac{(6.66 - 1.11) \times 7.66}{12.2}$

There are no built-in units in the ETABS computer program. The user must prepare the input in a consistent set of units. The output produced by the program will then conform to the same set of units.

Therefore, if the user chooses to use kips (1000 pounds) and inches as the input units, all the dimensions of the structure must be entered in inches and all the loads in kips. The material properties should also conform to these units. The output units will then be in kips and inches, so that member axial forces and shear forces will be in kips, and all bending moments will be in kip-inches. Displacements will be in inches and joint rotations will be in radians.

However, if the user wishes to use a database of section properties or the automatic lateral wind load generation feature of the program the user needs to specify the units used in the input data file. Similarly, if the user anticipates using any of the design post processors of the ETABS system, such as STEELER, CONKER or WALLER, the ETABS input data will need to be prepared in a specific set of units as defined in the corresponding post processor manuals.

The following is the convention used to define each data line:

First, the format giving the sequence of the entries of each data line is presented as a series of abbreviations of the options (or variables).

Each data section is then followed by a tabular description in the form:

Variable   Field   Note   Entry

The **Variable** is the abbreviation of the entry made on the data line.

The **Field** is a number that corresponds to the sequence in which the variable exists on the data line. Thus if a variable is the fourth entry on a data line, it will have a field number of 4.

The **Note** number refers to the series of notes that exist at the end of the corresponding data section. The notes describe the data options in more detail and give important information to aid the user in better understanding the options of the program.

The **Entry** is a brief description of the option.

It is recommended that the user be thoroughly familiar with the main control variables in Section 1.c and the frame control variables in Section 6.i.b. Repeated references are made to these variables throughout this chapter.

Data Block	When Needed
1. Main Control Data	Always
2. Automatic Mass Properties Calculation Data	Only if **NMASS** > 0
3. Story Data	Always
4. Material Property Data	Always
5. Section Property Data	Always
i. Column Properties	only if **NCP** > 0
ii. Beam Properties	only if **NBP** > 0
iii. Floor Properties	only if **NFP** > 0
iv. Brace Properties	only if **NBRP** > 0
v. Panel Properties	only if **NPP** > 0
vi. Spring / Link Properties	only if **NSP** > 0
6. Frame Data Sets	Always (**NDF** sets)
i. Frame Control Data	always
ii. Column Line Coordinates and Orientation	always
iii. Beam Bay Connectivity	only if **NB** > 0
iv. Floor Bay Connectivity	only if **NF** > 0
v. Joint Load Patterns	only if **NJLP** > 0
vi. Beam Load Patterns	only if **NBLP** > 0
vii. Floor Load Patterns	only if **NFLP** > 0
viii. Joint Assignments (Diaphragm and Springs)	only if **IJ** > 0
ix. Column Assignments	always
x. Beam Assignments	only if **NB** > 0
xi. Floor Assignments	only if **NF** > 0
xii. Brace Location and Assignments	only if **MBR** > 0
xiii. Panel Location and Assignments	only if **MP** > 0
xiv. Link Location and Assignments	only if **ML** > 0
xv. Joint Load Assignments	only if **NJLP** > 0
xvi. Beam Load Assignments	only if **NBLP** > 0 & **NB** > 0
xvii. Floor Load Assignments	only if **NFLP** > 0 & **NF** > 0
7. Frame Location Data	Always
8. Lateral Static Load Data	Only if **ILAT** > 0
9. Lateral Dynamic Spectrum Data	Only if **IDYN** = 2
10. Lateral Dynamic Time History Data	Only if **IDYN** = 3 or 4
11. Load Case Data	Only if **NLD** > 0

Typical Data Setup for ETABS
Figure V-1

1. Main Control Data

Prepare the following data as defined in Sections a, b, c and d below. This data is always needed. A total of 5 data lines is required.

Format

a. Program Name and Version

Prepare one line of data to give the program name and version as follows:

ETABS 6.0

b. Heading Data

Prepare two lines of data for output labeling, up to 70 characters per line. This information will appear on every page.

c. Control Data

Prepare one line of control data for the whole building in the following form:

NST NMD NDF NTF NMASS NLD NPER NMAT NCP
NBP NFP NBRP NPP NSP ILAT IDYN IST IPD IRGD
ISLF IUNIT

Note: The 21 entries in the line above should be entered on one data line in the input. They are shown on three lines above because the abbreviations need that much space.

d. Miscellaneous Parameters

Provide one data line in the following form:

GRAV EVT CUT PDFAC PROPFILE

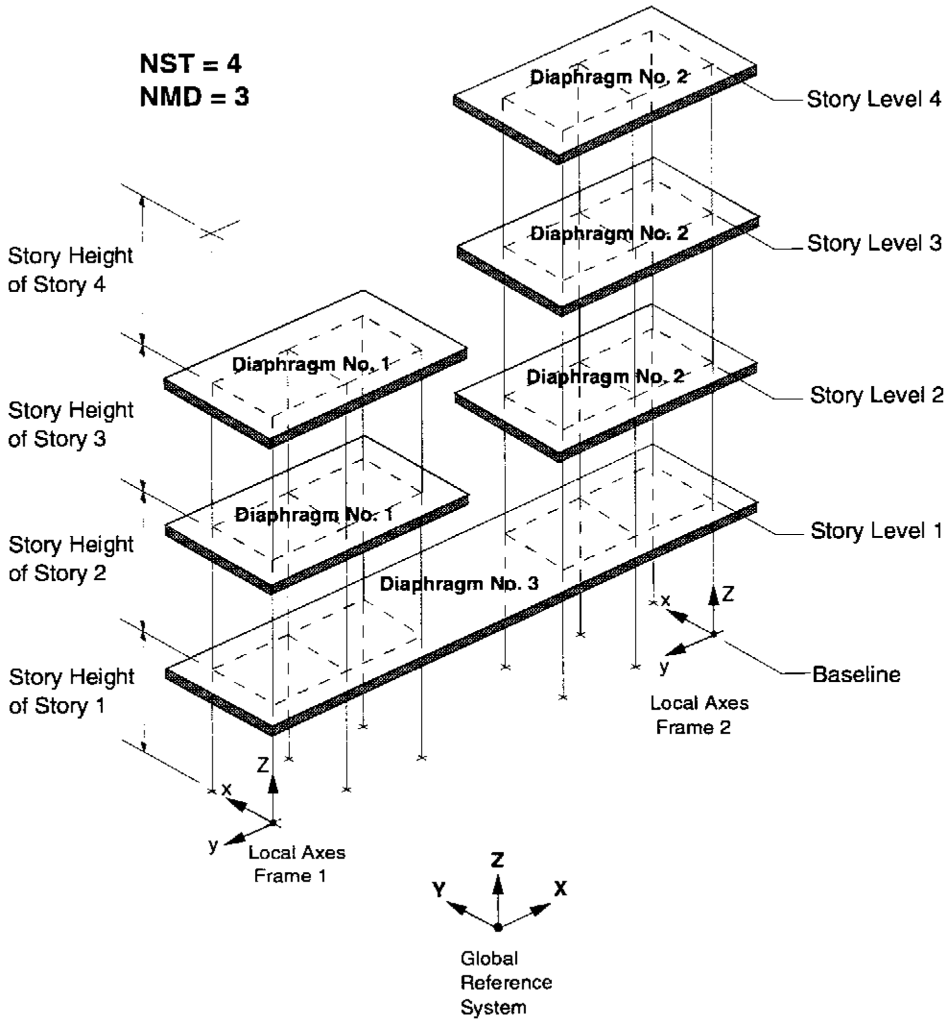
Description

Variable	Field	Note	Entry
c. Control Data			
NST	1	(1)	Number of stories in the building, not including the baseline.
NMD	2	(2)	Largest diaphragm number at any story level. Not less than 1.
NDF	3	(3)	Number of frames in the building with different properties or different loading.
NTF	4	(4)	Total number of frames in the building.
NMASS	5	(5)	Number of diaphragm mass types: = 0 No automatic mass properties calculated. > 0 Automatic calculation of **NMASS** mass type properties.
NLD	6	(6)	Total number of structural load cases.
NPER	7	(7)	Number of structural periods and mode shapes to be generated.
NMAT	8	(8)	Number of different material property types in all the frames. Must be at least 1.
NCP	9	(9)	Number of different column section property types in all the frames.
NBP	10	(10)	Number of different beam section property types in all the frames.
NFP	11	(11)	Number of different floor section property types in all the frames.
NBRP	12	(12)	Number of different brace section property types in all the frames.

Variable	Field	Note	Entry
NPP	13	(13)	Number of different panel section property types in all the frames.
NSP	14	(14)	Number of different spring and link property types in all the frames.
ILAT	15	(15)	Static lateral analysis code: = 0 No lateral static loads. = 1 User-defined lateral loads. = 11 UBC '94 lateral seismic loads. = 12 BOCA '93 lateral seismic loads. = 13 NBCC '90 lateral seismic loads. = 14 ATC 3-06 lateral seismic loads. = 21 UBC '94 lateral wind loads. = 22 BOCA '93 lateral wind loads. = 23 NBCC '90 lateral wind loads. = 24 ASCE 7-93 lateral wind loads. = 31 UBC '94 lateral seismic loads with base isolation.
IDYN	16	(16)	Dynamic lateral analysis code: = 0 No dynamic analysis. = 1 Time periods and mode shapes only. = 2 Response spectrum analysis. = 3 Linear time history analysis. = 4 Nonlinear time history analysis.
IST	17	(17)	Structure type code: = 0 Three-dimensional, with diaphragm rotations (i.e. unlocked). = 1 Two-dimensional. Displacements in global X-Z plane only. = 2 Two-dimensional. Displacements in global Y-Z plane only. = 3 Three-dimensional, without diaphragm rotations (i.e. locked).

Variable	Field	Note	Entry
IPD	18	(18,25)	P-Delta analysis code: = 0 Do not include P-Delta effects. = 1 Include P-Delta effects. Diaphragm masses must be provided for P-Delta effects to be included.
IRGD	19	(19)	Frame joint stiffness modification code: = 0 Do not reduce rigid zones. = 1 Reduce rigid zones by 25%. = 2 Reduce rigid zones by 50%. = 3 Reduce rigid zones by 75%. = 4 Reduce rigid zones by 100%.
ISLF	20	(20)	Frame self weight calculation code: = 0 Frame self weight not automatically included in analysis. = 1 Self weight calculated and included in load condition I. = 2 Self weight calculated and included in load condition II. = 3 Self weight calculated and included in load condition III.
IUNIT	21	(21)	Code for consistent set of units used in input: = 0 Default. = 1 Kip - inch. = 2 Kilogram force - meters. = 3 KiloNewton - meters. = 4 Pound - inch. = 5 Kip - feet. = 6 Kilogram force - millimeters. = 7 KiloNewton - millimeters. = 8 Pound - feet.

Variable	Field	Note	Entry
d. Miscellaneous Parameters			
GRAV	1	(22)	Gravitational acceleration.
EVT	2	(23)	Eigenvalue tolerance.
CUT	3	(24)	Cutoff time period.
PDFAC	4	(25)	Factor to be applied to P-Delta effects.
PROPFILE	5	(26)	User section property filename.



An Example of Story Level and Diaphragm Numbering
Figure V-2

Notes

1. The number of stories in the building is the number of levels above the modeled baseline of the building. See Figure V-2.

2. This is the largest identification number assigned to any rigid diaphragm in the story data specified in Section 3 below. The program assumes that all **NMD** diaphragms exist at all story levels. If a particular diaphragm is not connected to any member at a particular story level the program automatically restrains it. Although non-sequential diaphragm numbering is possible it is not recommended. **NMD** may not be less than 1.

3. This variable controls the number of frame data sets to be read in Section 6 below.

   Input data for frames with identical frame properties and frame loading need only be prepared once.

4. The **NTF** total frames consisting of **NDF** different frames, are located through the frame location data specified in Section 7 below.

5. This variable controls the number of data sets to be read in Section 2 below for the automatic calculation of diaphragm mass properties.

6. This variable controls the number of load cases to be read in Section 11 below. Load cases are defined as combinations of the eight basic load conditions.

7. For single diaphragm per floor the maximum possible number of modes is 3***NST** for 3-D structures and **NST** for 2-D structures. For multiple diaphragms per floor the maximum number of modes possible is three times the number of diaphragms.

8. This variable controls the number of material property data lines to be read in Section 4 below.

9. This variable controls the number of column property data sets to be read in Section 5.i. below.

10. This variable controls the number of beam property data sets to be read in Section 5.ii. below.

11. This variable controls the number of floor property data lines to be read in Section 5.iii. below.

12. This variable controls the number of brace property data sets to be read in Section 5.iv. below.

13. This variable controls the number of panel property data lines to be read in Section 5.v. below.

14. This variable controls the number of spring and link property data sets to be read in Section 5.vi. below.

15. This variable controls the definition of the static lateral loading of the building to be defined in Section 8 below.

16. This variable triggers the dynamic analysis options of the program, thereby controlling the type of dynamic data required by the program in the input sequence. See Sections 9 and 10 below.

17. This parameter only affects the lateral movements of the rigid diaphragms at the different story levels. All joints not connected to any rigid diaphragm are unaffected by this parameter and maintain full three dimensional translational and rotational freedom.

    If **IST** is 0, the diaphragms are allowed to translate in the global X- and Y-directions and rotate about the global Z-direction.

    If **IST** is 1, the diaphragms are allowed to translate only in the global X-direction. The global Y-translation and the global Z-rotation are restrained.

    If **IST** is 2, the diaphragms are allowed to translate only in the global Y-direction. The global X-translation and the global Z-rotation are restrained.

    If **IST** is 3, the diaphragms are allowed to translate in the global X- and Y-directions but the global Z-rotation is restrained (locked). This option can be used to lock the diaphragms in rotation to study the effects the elimination of rotations have on the distribution of the structural lateral shear forces.

18. Overall structural P-Delta effects are included as defined in Reference [8]. Diaphragm weight computed by the program from the input diaphragm mass, and the story heights below, are used to modify the lateral stiffness between stories.

    If multiple diaphragms per story are present, the P-Delta effect is included between diaphragms with the same number. Caution is advised where vertical support elements are connected to different diaphragm numbers from one story to the next or are absent below because of modeling stepped structures. Manual modifications to the stiffness may be modeled using link elements with negative stiffness to account for the P-Delta effect in such situations.

19. The dimensions of beam and column rigid zones for member stiffness formulations can be modified by the use of this parameter. See Chapter III for details.

20. The self weights of the frames can be added to any of the three vertical load conditions.

21. The program assumes the same consistent set of units is used throughout the input. The units information is used to correctly convert the section properties recovered from the database which may be in a different set of units. This information is also used to convert code based lateral static wind loads which are unit dependent.

22. The gravitational acceleration (e.g. 9.81 m/sec^2 or 386.4 in/sec^2) is needed to convert mass units to weight units at various steps in the program, such as in the evaluation of P-Delta effects and in the automatic calculation of the lateral static seismic forces.

23. The eigenvalues are evaluated by an accelerated subspace iteration algorithm. The iteration will continue until the change in the time period, T, of a particular mode in successive iterations is less than **EVT**, that is:

    $$| T_n - T_{n+1} | < \textbf{EVT}$$

    where n and n+1 denote successive iteration numbers.

    If **EVT** is not specified, it is assumed to be 0.0001.

24. The program will terminate the eigensolution for the **NPER** time periods when all the modes having a time period greater than **CUT** have been found.

    Only the number of modes that are evaluated will be included in any subsequent response spectrum or time history analysis processing.

25. The P-Delta effects are automatically accounted for in the program (if **IPD** = 1) by calculating the effects of the diaphragm weights, obtained from the specified diaphragm masses and the gravitational constant, deflecting over the calculated elastic diaphragm displacements. The **PDFAC** is used as a multiplier for the P-Delta effects. This allows the user to account for the factored story weights and/or inelastic displacements.

    If **IPD** = 1 and **PDFAC** is left as zero it defaults to 1.

26. The program has the ability to extract section property information from a database. The program always searches for the section labels first in a file named SECTIONS.PRO. The user also can specify one additional file to be searched if a section label is not found in file SECTIONS.PRO. **PROPFILE** is the name of this additional file. **PROPFILE** is a filename (maximum 8 characters) with a .PRO extension. Both the file SECTIONS.PRO and the one specified through **PROPFILE** must be prepared using the ETABS utility program PROPER. Both these files must be present either in the directory of the input file or in the directory where the program ETABS resides.

    The program ETABS comes with a file AISC.PRO containing the database of properties for the sections given in the "Manual of Steel Construction," published by American Institute of Steel Construction (AISC). The available labels are given in the file AISC.LBL. The file AISC.PRO also comes copied into the file SECTIONS.PRO for immediate use. Users not using AISC sections should copy their database files into SECTIONS.PRO.

2. Automatic Mass Properties Calculation Data

If **NMASS** is 0, no automatic calculation of diaphragm mass properties is to be made. Therefore, skip this data section.

Otherwise, provide one set of data (a and b below) for each of the **NMASS** mass types. Each mass type is discretized as a series of **NSEG** points, lines, circular arcs, rectangular segments and circular segments, each having its own mass, defined by mass intensity **AM**. See Figure V-3.

Mass types defined here are associated with a rigid diaphragm through the Story Data as described in Section 3 of this chapter.

Format

a. Control Data

Prepare one line of data in the following form:

MID NSEG SF

b. Segment Data Lines

Provide **NSEG** (defined above) data lines in the following form dependent on the type of mass segment been defined:

for point mass (**MTYPE** = POINT)

MTYPE AM XC YC

for line mass (**MTYPE** = LINE)

MTYPE AM XC YC BB ANG

for rectangular area mass (**MTYPE** = RECT)

MTYPE AM XC YC BB DD ANG

for circular arc mass (**MTYPE** = ARC)

MTYPE AM XC YC BB ANG RAD

for circular area mass (**MTYPE** = CIRCLE)

MTYPE AM XC YC BB DD ANG RAD

for triangular area mass (**MTYPE** = TRIANGLE)

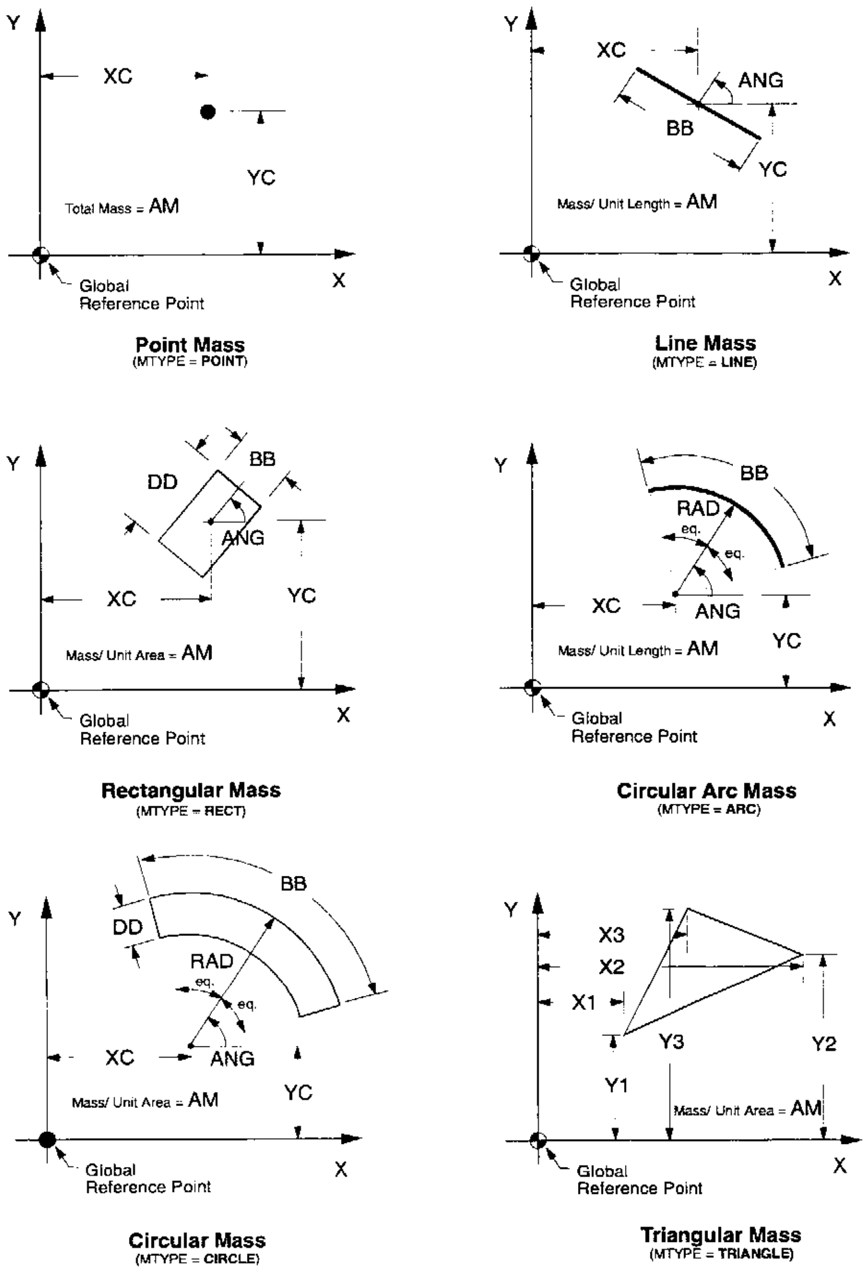
MTYPE AM X1 Y1 X2 Y2 X3 Y3

Description

Variable	Field	Note	Entry
a. Control Data			
MID	1	(1)	Mass type identification number.
NSEG	2	(2)	Number of segments defining mass distribution.
SF	3	(3)	Scale factor for scaling mass **AM**.
b. Segment Data Lines			
for point mass (**MTYPE** = POINT)			
MTYPE	1	(4)	Mass shape type: = POINT
AM	2	(5)	Total mass of this point segment.
XC	3	(6)	x distance to mass point.
YC	4		y distance to mass point.
for line mass (**MTYPE** = LINE)			
MTYPE	1	(4)	Mass shape type: = LINE
AM	2	(5)	Mass per unit length of this segment.
XC	3	(6)	x distance to center of linc.
YC	4		y distance to center of line.
BB	5		Length of line.
ANG	6		Angle in degrees as defined in Figure V-3.

Variable	Field	Note	Entry
for rectangular area mass (**MTYPE** = RECT)			
MTYPE	1	(4)	Mass shape type: = RECT
AM	2	(5)	Mass per unit area of this segment.
XC	3	(6)	x distance to center of rectangle.
YC	4		y distance to center of rectangle.
BB	5		Width of rectangle.
DD	6		Depth of rectangle.
ANG	7		Angle in degrees as defined in Figure V-3.
for circular arc mass (**MTYPE** = ARC)			
MTYPE	1	(4)	Mass shape type: = ARC
AM	2	(5)	Mass per unit length of this segment.
XC	3	(6)	x distance to center of circle.
YC	4		y distance to center of circle.
BB	5		Length of arc.
ANG	6		Angle in degrees as defined in Figure V-3.
RAD	7		Radius of circle.

Variable	Field	Note	Entry
for circular area mass (**MTYPE** = CIRCLE)			
MTYPE	1	(4)	Mass shape type: = CIRCLE
AM	2	(5)	Mass per unit area of this segment.
XC	3	(6)	x distance to center of circle.
YC	4		y distance to center of circle.
BB	5		Outer circumferential dimension of circular area.
DD	6		Radial dimension of circular area.
ANG	7		Angle in degrees as defined in Figure V-3.
RAD	8		Outer radius of circular area.
for triangular area mass (**MTYPE** = TRIANGLE)			
MTYPE	1	(4)	Mass shape type: = TRIANGLE
AM	2	(5)	Mass per unit area of this segment.
X1	3	(6)	x coordinate of vertex 1 of triangle.
Y1	4		y coordinate of vertex 1 of triangle.
X2	5		x coordinate of vertex 2 of triangle.
Y2	6		y coordinate of vertex 2 of triangle.
X3	7		x coordinate of vertex 3 of triangle.
Y3	8		y coordinate of vertex 3 of triangle.



Definition of Mass Segment Dimensions
Figure V-3

Notes

1. Mass type identification numbers must be between 1 and **NMASS** and specified in ascending, consecutive order.

2. The diaphragms are discretized as a series of points, lines, rectangular areas, circular arcs, circular areas and triangular areas each having a mass intensity **AM** and dimensions as defined in Figure V-3. Using this data the program calculates a total mass, the global coordinates of the center of mass, and a mass moment of inertia about a vertical axis passing through the center of mass. There is no limit on the number of segments used to define a mass type.

3. This number is a multiplier for all **AM** entries of this mass type. For instance, the user may input all the **AM** entries in weight units and then use a scale factor of (1/**GRAV**) to convert the weight quantities to mass quantities. If **SF** is not entered, a value of 1.0 is used.

4. This entry defines the shape of mass segment being defined.

5. Mass has units of force divided by gravitational acceleration (W/g).

6. All dimensions are as defined in Figure V-3 and are given in reference to the global coordinate system.

3. Story Data

This data is always needed. Prepare one set of data (a and b below) corresponding to each story of the building. The data is to be prepared starting from the top story of the building and progressing in sequence towards the bottom of the building. The data amounts to a total of **NST** sets as follows.

Format

a. Control Data

Prepare one line of data in the following form:

SID SH NDIA

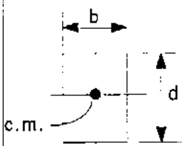
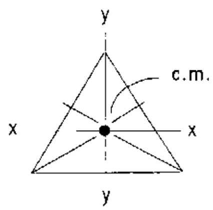
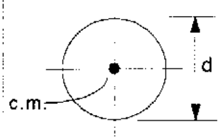
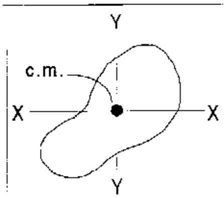
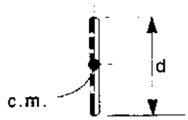
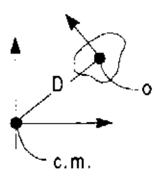
b. Diaphragm Data Lines

Provide **NDIA** (defined above) data lines in the following form. Skip this data line if **NDIA** is zero.

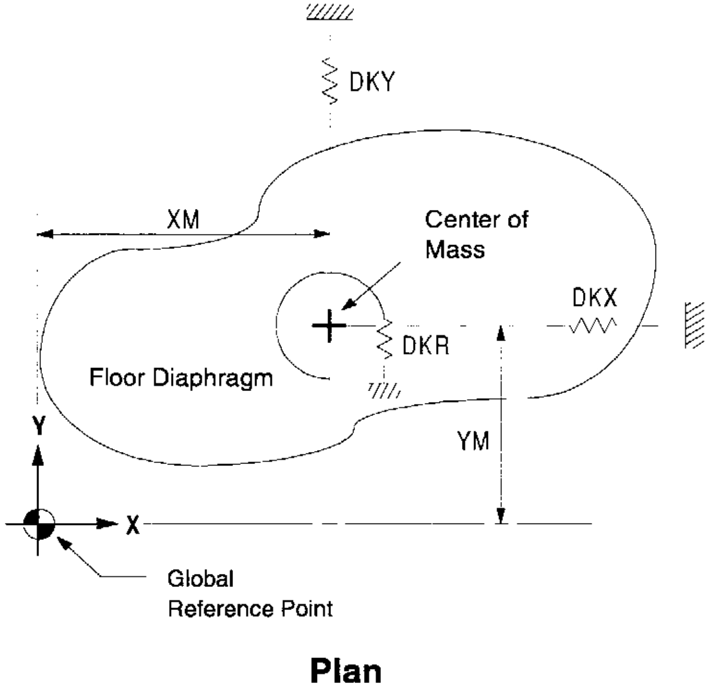
ND IMST DMASS DMMI XM YM DKX DKY DKR

Description

Variable	Field	Note	Entry
a. Control Data			
SID	1	(1)	Story identification label for this level.
SH	2	(2)	Story height associated with this level.
NDIA	3	(3)	Number of separate rigid diaphragms at this story level at which mass or external diaphragm stiffness are to be specified.
b. Diaphragm Data Lines			
ND	1	(4)	Diaphragm identification number.
IMST	2	(5)	Mass type code: = 0 Mass properties of this diaphragm are as input in this data. > 0 Mass properties of this diaphragm are equal to those computed from mass type **MID**, as defined in Mass Data, Section 2.
DMASS	3	(6)	Translational mass.
DMMI	4	(6)	Rotational mass moment of inertia of mass **DMASS** about a vertical axis through its center.
XM	5	(7)	X-distance to the center of mass **DMASS** measured from the global reference point.
YM	6		Y-distance to the center of mass **DMASS** measured from the global reference point.

Shape in plan	Mass Moment of Inertia about vertical axis (normal to paper) through center of mass	Formula
	Rectangular diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)	$MMI_{cm} = M/12\,(b^2 + d^2)$
	Triangular diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)	Use general diaphragm formula
	Circular diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g)	$MMI_{cm} = \frac{Md^2}{8}$
	General diaphragm Uniformly distributed mass per unit area Total mass of diaphragm = M (or w/g) Area of diaphragm = A Moment of inertia of area about XX = I_x Moment of inertia of area about YY = I_y	$MMI_{cm} = M/A\,(I_x + I_y)$
	Line mass Uniformly distributed mass per unit length Total mass of line = M (or w/g)	$MMI_{cm} = \frac{Md^2}{12}$
	Axis transformation for a mass A If mass is a point mass, $MMI_o = O$	$MMI_{cm} = MMI_o + MD^2$

Formulas for Mass Moment of Inertia
Figure V-4



Typical Diaphragm Plan at a Story Level
Figure V-5

Variable	Field	Note	Entry
DKX	7	(8)	External diaphragm stiffness in the X-direction.
DKY	8		External diaphragm stiffness in the Y-direction.
DKR	9		External diaphragm stiffness in the diaphragm rotational direction.

Notes

1. This entry may have up to eight characters with no embedded blanks. Identification labels for each level must be unique. The word **BASELINE** may not be used as a story identification label.

2. This entry is an inter story height. It defines the level of the horizontal diaphragms of this story. Story lateral loads are assumed to be generated at this level. Frame levels must correspond to structural levels defined in the story data. A zero story height is not allowed. See Figure V-2.

3. This entry identifies the number of lines to be read in the data subsection immediately following (Section b below). This number must be less than or equal to **NMD** defined in data Section 1.c.

4. This entry must be less than or equal to **NMD** defined in data Section 1.c. Also, this identification number may not be repeated in this data subsection.

5. The diaphragm mass properties **DMASS, DMMI, XM** and **YM** may be assigned either by defining the **IMST** entry or by explicitly specifying the mass properties on this data line. This entry must not be greater than **NMASS**. If this entry is non zero, any **DMASS, DMMI, XM** and **YM** entries on this line are superseded by the corresponding values established by the **IMST** entry.

   Formulas for the calculation of the mass moment of inertia of typical floor configurations are shown in Figure V-4.

   Diaphragms can also be assigned tributary element mass through assignment of joints to diaphragms (Section 6.viii below). Any mass so specified is added to any diaphragm mass specified directly in this data section and new center of mass and mass moment of inertia about it are calculated by the program automatically. Element mass can be specified through providing a mass density in the material property data (Section 4 below).

6. The diaphragm mass properties **IMST** or **DMASS, DMMI, XM** and **YM**; and/or element mass must be defined if:

   – The entry for **ILAT** is 11, 12, 13, 14 or 31 (i.e. static seismic loads are specified).

   – The entry for **IDYN** is 1, 2, 3 or 4 (i.e. dynamic analysis is to be made).

- The entry for **IPD** is 1 (i.e. P-Delta effects are to be considered).

7. The center of mass of the diaphragm is calculated by the program from **DMASS** located at **XM** and **YM** and any element mass tributary to the diaphragm. If no mass is present on the diaphragm, **XM** and **YM** represent user defined center of mass of the diaphragm. The center of mass of the diaphragm is related to the following aspects of the solution process:

   - The dynamic lateral forces are generated at this point.

   - The external diaphragm stiffnesses, if any, are assumed to act at this point.

   - The diaphragm displacements (lateral translations and rotation) are printed at this point.

   If the center of mass is not defined, it is assumed to be (0,0), in which case the diaphragm displacements will be printed at the global reference point.

8. These are external lateral spring constants that are located on the floor diaphragm at the calculated center of mass. See Figure V-5. These stiffnesses can be used to model lateral soil springs for levels that exist underground.

4. Material Property Data

Provide one data line for each of the **NMAT** material property types. This data section is always needed.

Format

The data line is to be prepared in the following form:

MID MTYPE E U W M ALPHA DP1 DP2 DP3 DP4

Description

Variable	Field	Note	Entry
MID	1	(1)	Material identification number.
MTYPE	2	(2)	Material type: = S Steel = C Concrete (frames) = W Concrete (walls) = M Masonry = O Other
E	3	(3)	Modulus of elasticity.
U	4	(3)	Poisson's ratio.
W	5	(4)	Weight density (weight/volume).
M	6	(5)	Mass density (mass/volume).
ALPHA	7	(6)	Coefficient of thermal expansion.
DP1	8	(7)	Design parameter 1 (see Figure V-6).
DP2	9		Design parameter 2 (see Figure V-6).
DP3	10		Design parameter 3 (see Figure V-6).
DP4	11		Design parameter 4 (see Figure V-6).

Material Type Code	Design Parameters			
	DP1	DP2	DP3	DP4
S (Steel)	F_y	$*F_{bmaj}$	$*F_{bmin}$	-
C (Concrete Frame)	f_y	f'_c	f_{ys}	f_{cs}
W (Concrete Walls)	f_y	f'_c	f_{ys}	f_{cs}
M (Masonry Walls)	f_y	f'_m	f_{ys}	f_{ms}
O (Other)	-	-	-	-

where,

Fy = Yield stress of structural steel
F_{bmaj} = Allowable bending stresses for structural steel
F_{bmin} sections, in the major and minor directions
f_y = Yield stress of reinforcing steel
f'_c = Ultimate strength of concrete
f'_m = Specified strength of masonry
f_{ys} = Yield stress of shear reinforcing steel
f_{cs} = Equivalent strength of concrete for shear
f_{ms} = Equivalent strength of masonry for shear

* Provide these parameters only if values automatically calculated by program STEELER have to be overwritten.

Material Design/Check Parameters
Figure V-6

Notes

1. Material identification numbers must be input in ascending, consecutive sequence starting with the number 1.

2. A series of design/stress check post processors operating off the ETABS post processing data base are available. The material type designation is basically an indicator for the post processors. The steel stress checking post processor, for example, will only check those members that have a material type S and the concrete frame design post processor will only design the members having a material designation of C. Materials having a designation type O will not be processed by any of the post processors.

   Provide the corresponding entries even if no post processing is anticipated.

3. Poisson's ratio is used for calculating the shear modulus, $G = E/2(1+U)$

4. The weight density of the element material is in force/unit volume units (e.g. 150 pounds/cubic foot for concrete). This is used to calculate the self weight of the element. The self weight is added into vertical load condition **ISLF**. The self weight is based upon the axial area and story to story height for columns, the full length of braces, and the clear length (column face to column face) of beams. It is based on the thickness and area of panel and floor elements. Springs and links do not have any self weight associated with them.

4. The mass density of the element material is in mass/unit volume units. This is used to calculate the self mass of the element. The self mass is calculated for each element similar to the self weight calculation and is lumped at the joints to which the element is connected. This in turn gets lumped at the diaphragm to which this joint is assigned. This mass is in addition to any mass directly assigned to the diaphragms.

6. This entry is used for thermal stress analysis.

7. The design parameters depend upon the material type and are required by the post processors. These entries need not be provided if no post processing is anticipated.

5. Section Property Data

Prepare one (or up to six) of the following data sections, (i) through (vi) below, to define the section property tables of the various element types that make up the frames in the structure.

5.i. Column Properties

If **NCP** is 0, skip this data section. Otherwise, provide **NCP** data sets to define the **NCP** column property types.

Format

Each data set consists of a first data line immediately followed by a possible second data line.

a. First Data Line

Prepare the first data line in the following form:

ID ITYPE IMAT DMAJ DMIN TF TW RJ RIMAJ RIMIN

b. Second Data Line

This data line is only needed if the entry for **ITYPE** on the first data line is equal to USER or VARIABLE. If this line is needed, it should be prepared in the following form:

for **ITYPE** = USER

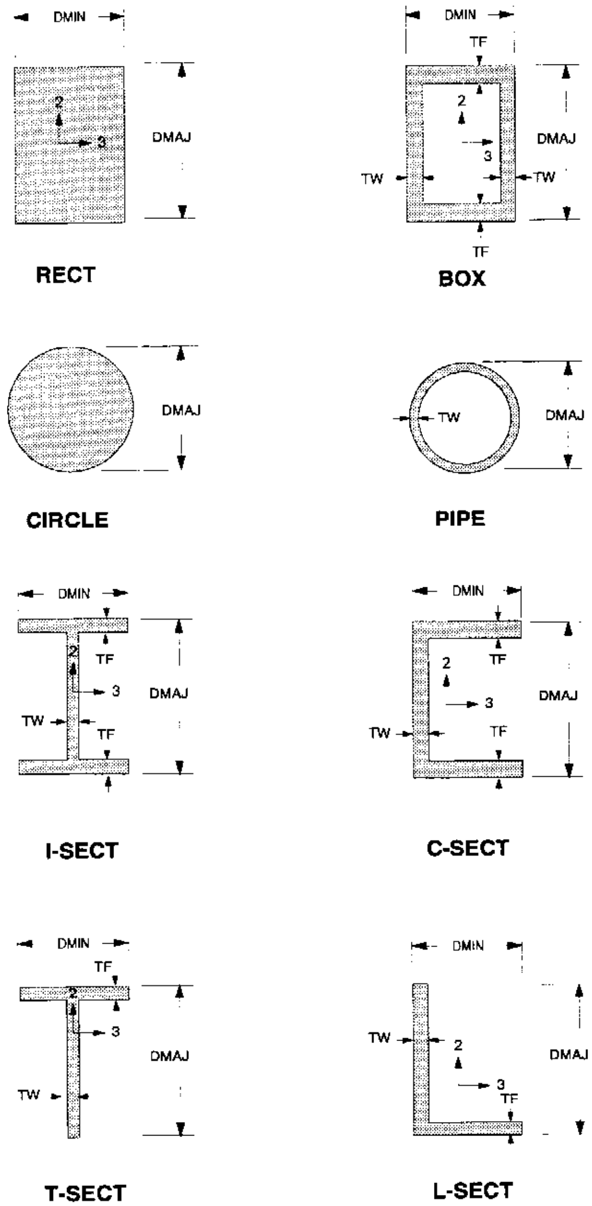
A AMAJ AMIN J IMAJ IMIN SMAJ SMIN ZMAJ ZMIN RMAJ RMIN

or for **ITYPE** = VARIABLE

IV ID1 L1 ID2 L2 ID3 L3 ID4

Description

Variable	Field	Note	Entry
a. First Data Line			
ID	1	(1)	Identification number of column section property set.
ITYPE	2		Section type:
		(2)	= USER
		(3)	= RECT
			= CIRCLE
			= PIPE
			= BOX
			= I-SECT
			= C-SECT
			= T-SECT
			= L-SECT
		(4)	= W14x230 or other label from database
		(5)	= VARIABLE
IMAT	3	(6)	Material identification number for this section property.
DMAJ	4	(3,7)	Section dimension in major direction.
DMIN	5	(3,7)	Section dimension in minor direction.
TF	6	(3)	Flange thickness.
TW	7	(3)	Web thickness.
RJ	8	(8)	Reduction factor for torsional constant.
RIMAJ	9		Reduction factor for moment of inertia about major axis.
RIMIN	10		Reduction factor for moment of inertia about minor axis.



Note: 2 is Major Direction, 3 is Minor Direction

Required Dimensions for Automatic Section Property Generation
Figure V-7

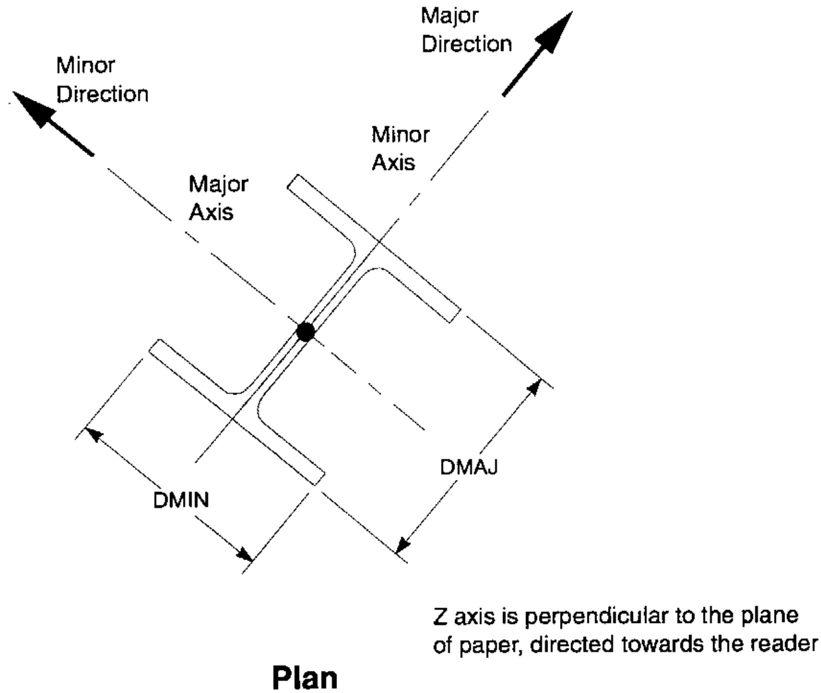
Variable	Field	Note	Entry

b. Second Data Line For ITYPE = USER

Variable	Field	Note	Entry
A	1	(2)	Cross-sectional axial area.
AMAJ	2	(9)	Shear area corresponding to major direction shear forces.
AMIN	3	(9)	Shear area corresponding to minor direction shear forces.
J	4		Torsional constant.
IMAJ	5		Moment of inertia, about major axis.
IMIN	6		Moment of inertia, about minor axis.
SMAJ	7		Section modulus, about major axis.
SMIN	8		Section modulus, about minor axis.
ZMAJ	9		Plastic modulus, about major axis.
ZMIN	10		Plastic modulus, about minor axis.
RMAJ	11		Radius of gyration, about major axis.
RMIN	12		Radius of gyration, about minor axis.

Second Data Line For ITYPE = VARIABLE

Variable	Field	Note	Entry
IV	1	(5)	Code for variation of properties in segments: = 0 Constant = 1 Linear = 2 Parabolic = 3 Cubic
ID1	2		Section property ID at start of member.
L1	3		Relative length of segment 1.
ID2	4		Section property ID at start of segment 2.



Column Section Orientation
Figure V-8

Variable	Field	Note	Entry
L2	5		Relative length of segment 2.
ID3	6		Section property ID at start of segment 3.
L3	7		Relative length of segment 3.
ID4	8		Section property ID at end of segment 3.

Notes

1. The column section property numbers must start with the number 1 and proceed in an ascending, consecutive sequence.

2. If **ITYPE** is **USER**, the user is to calculate all the section properties and provide them on the second data line (the second data line must immediately follow the first data line). Only the first six entries of the second data line are read if the material type associated with this section is not steel (S). The last six entries are needed only if the material type is steel and if steel stress check post processing is anticipated.

3. If **ITYPE** is **RECT, CIRCLE, PIPE, BOX, I-SECT, C-SECT, T-SECT** or **L-SECT**, the program recognizes the shapes given in Figure V-7 and calculates the section properties from the dimensions **DMAJ, DMIN, TF** and **TW**.

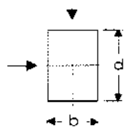
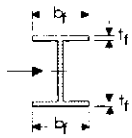
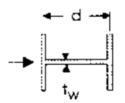
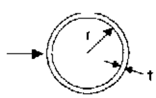
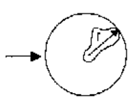
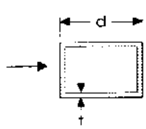
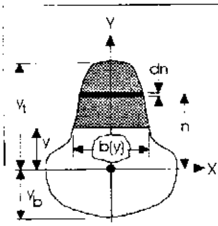
4. If **ITYPE** is a label from the database, the complete property set associated with the section (including the dimensions of the section) is recovered by the program from the data base and assigned to the section property identification number.

   The program searches for the label first in the file SECTIONS.PRO and then in any file specified through **PROPFILE** in the Main Control Data, Section 1.d.

5. If **ITYPE** is **VARIABLE**, the user defines variable section properties along the member length, by referring to section properties already defined, for different segments along the member. Up to three different segments along the member length can be defined. This information is provided on the second data line (the second data line must immediately follow the first data line).

   The material properties for this variable section type are as defined for this type. The geometric properties are obtained from section properties **ID1**, **ID2**, **ID3** and **ID4** which must have been defined earlier and may not in themselves be of variable type. The lengths **L1**, **L2** and **L3** are relative to the clear length of the member and must add up to 1.0.

   The **IV** code determines the variation of properties along the segment lengths. **IV** = 0 means all geometric properties are constant along the segment length and are discontinuous between segments. **ID4** if specified for this case is ignored. **IV** = 1 means all geometric properties vary linearly along the segment length and are continuous between segments. IV = 2 and IV = 3 varies the moment of inertia both for major and minor bending parabolically and cubically, respectively, along the segment lengths. All other properties vary linearly and the properties are continuous between segments. See Figure V-11 for examples of uses of this option.

Section	Description	Effective Shear Area
	Rectangular Section Shear Forces parallel to the b or d directions	$^{5}/_{6}\ bd$
	Wide Flange Section Shear Forces parallel to flange	$^{5}/_{3}\ t_f\ b_f$
	Wide Flange Section Shear Forces parallel to web	$t_w\ d$
	Thin Walled Circular Tube Section Shear Forces from any direction	$\pi\ r\ t$
	Solid Circular Section Shear Forces from any direction	$0.9\ \pi\ r^2$
	Thin Walled Rectangular Tube Section Shear Forces parallel to d-direction	$2\ t\ d$
	General Section Shear Forces parallel to Y-direction I_x = moment of inertia of section about X-X $Q(Y) = \int_{y}^{y_t} n\,b(n)\,dn$	$\dfrac{I_x^2}{\int_{y_b}^{y_t} \frac{Q^2(y)}{b(y)}\,dy}$

Formulas For Calculating Shear Areas
Figure V-9

6. This entry references the material property types previously defined in Section 4 and must not be less than 1 and not greater than **NMAT**.

7. In addition to being used by the automatic section property calculation options, the column dimensions, **DMAJ** and **DMIN**, are also used for determining the lengths of the rigid zones of the beams that frame into the columns defined with this property set. For column section orientation, see Figure V-8.

8. The reduction factors are factors applied to the moment of inertias and torsional constants specified, calculated or recovered from a database. They provide an easy method of modifying the bending or torsional stiffness to account for different modeling conditions, for example, cracking in concrete. If left blank or specified as zero they default to 1.0.

9. A shear area of pure zero will cause the program to exclude the effect of shear deformations. In other words, the shear deformations will be assumed to be zero. Effectively, a pure zero shear area is defaulted to an infinite shear area by the program. Formulas for calculating the shear areas of typical sections are given in Figure V-9.

5.ii. Beam Properties

If **NBP** is 0, skip this data section. Otherwise, provide **NBP** data sets to define the **NBP** beam property types.

Format

Each data set consists of a first data line immediately followed by a possible second data line.

a. First Data Line

Prepare the first data line in the following form:

ID ITYPE IMAT DBMAJ DAMAJ DMIN TF TW RJ RIMAJ RIMIN

b. Second Data Line

This data line is only needed if the entry for **ITYPE** on the first data line is equal to USER or VARIABLE. If this line is needed, it should be prepared in the following form:

for **ITYPE** = USER

A AMAJ AMIN J IMAJ IMIN SMAJ SMIN ZMAJ ZMIN RMAJ RMIN

or for **ITYPE** = VARIABLE

IV ID1 L1 ID2 L2 ID3 L3 ID4

Description

Variable	Field	Note	Entry
a. First Data Line			
ID	1	(1)	Identification number of beam section property set.
ITYPE	2		Section type:
		(2)	= USER
		(3)	= RECT
			= CIRCLE
			= PIPE
			= BOX
			= I-SECT
			= C-SECT
			= T-SECT
			= L-SECT
		(4)	= W14x230 or other label from database
		(5)	= VARIABLE
IMAT	3	(6)	Material identification number for this section property.
DBMAJ	4	(3,7)	Section dimension in major direction, depth below diaphragm.
DAMAJ	5	(3,7)	Section dimension in major direction, depth above diaphragm.
DMIN	6	(3,7)	Section dimension in minor direction.
TF	7	(3)	Flange thickness.
TW	8	(3)	Web thickness.
RJ	9	(8)	Reduction factor for torsional constant.
RIMAJ	10		Reduction factor for moment of inertia about major axis.

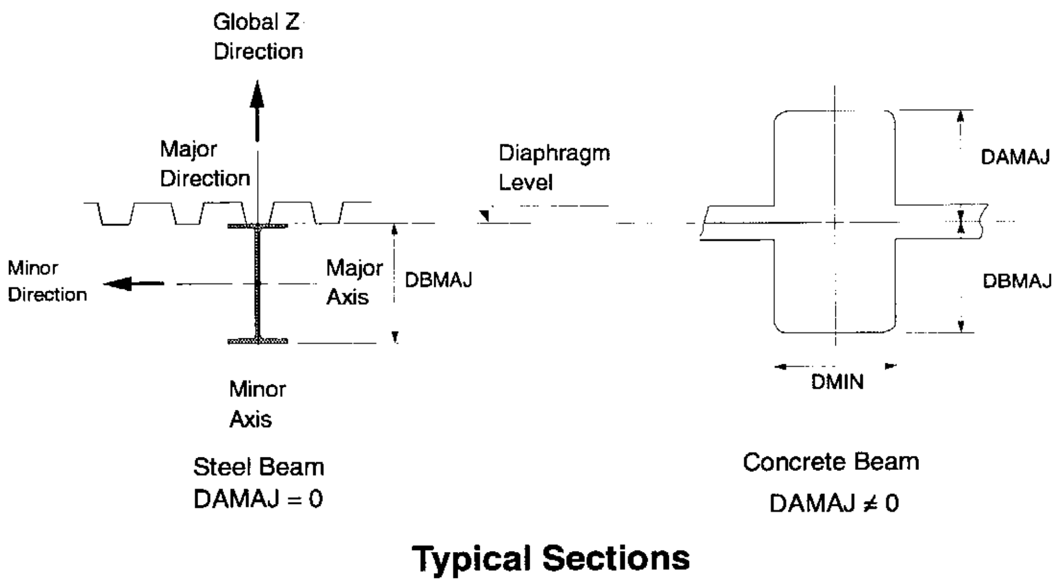
Variable	Field	Note	Entry
RIMIN	11		Reduction factor for moment of inertia about minor axis.

b. Second Data Line For ITYPE = USER

Variable	Field	Note	Entry
A	1	(2)	Cross-sectional axial area..
AMAJ	2	(8)	Shear area corresponding to major direction shear forces.
AMIN	3	(8)	Shear area corresponding to minor direction shear forces.
J	4		Torsional constant.
IMAJ	5		Moment of inertia, about major axis.
IMIN	6		Moment of inertia, about minor axis.
SMAJ	7		Section modulus, about major axis.
SMIN	8		Section modulus, about minor axis.
ZMAJ	9		Plastic modulus, about major axis.
ZMIN	10		Plastic modulus, about minor axis.
RMAJ	11		Radius of gyration, about major axis.
RMIN	12		Radius of gyration, about minor axis.

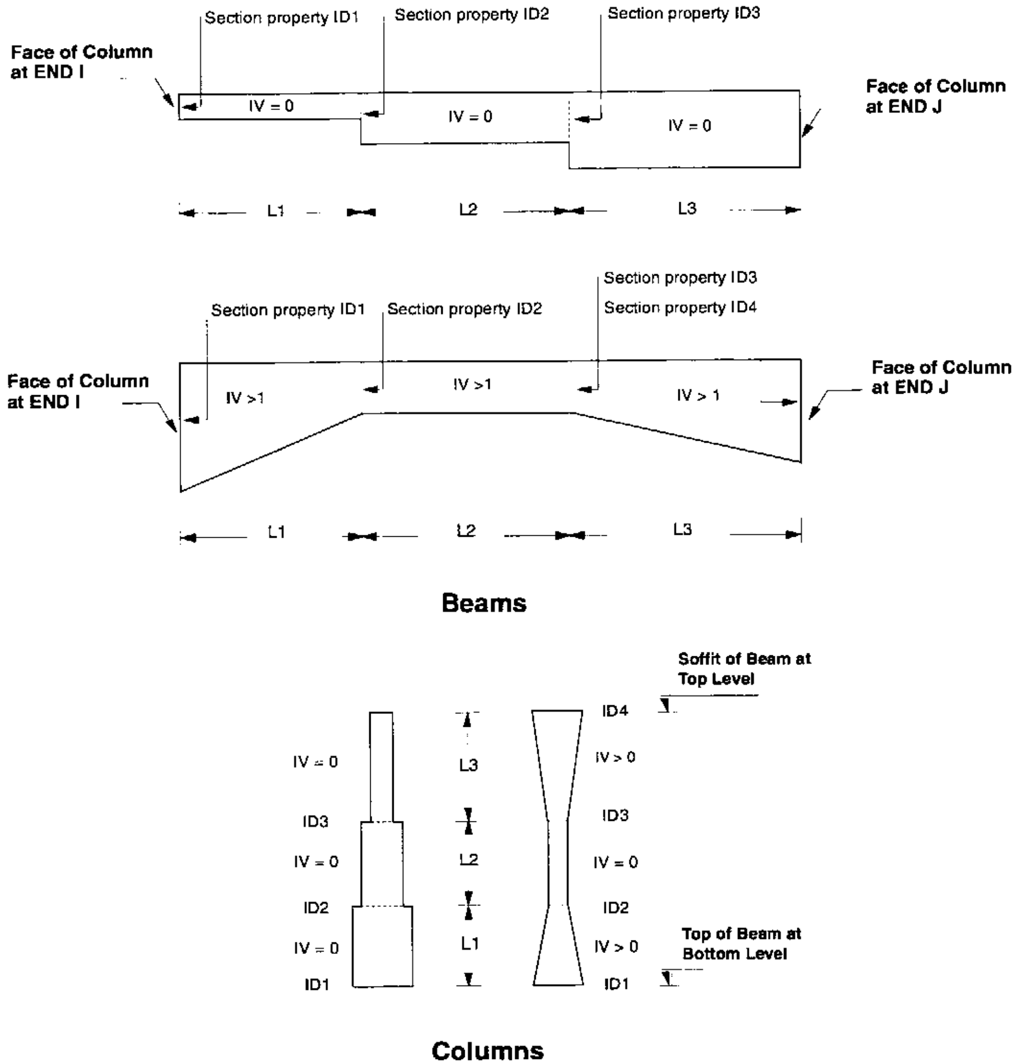
Second Data Line For ITYPE = VARIABLE

Variable	Field	Note	Entry
IV	1	(5)	Code for variation of properties in segments: = 0 Constant = 1 Linear = 2 Parabolic = 3 Cubic
ID1	2		Section property ID at start of member.
L1	3		Relative length of segment 1.



Beam Section Orientation
Figure V-10

Variable	Field	Note	Entry
ID2	4		Section property ID at start of segment 2.
L2	5		Relative length of segment 2.
ID3	6		Section property ID at start of segment 3.
L3	7		Relative length of segment 3.
ID4	8		Section property ID at end of segment 3.



Note:
L1, L2 and L3 are input as fractions of the clear span (length), i.e.
L1 + L2 + L3 = 1

Examples of Elements with Variable Section Properties
Figure V-11

Notes

1. The beam section property numbers must start with the number 1 and proceed in an ascending, consecutive sequence.

2. If **ITYPE** is **USER**, the user is to calculate all the section properties and provide them on the second data line (the second data line must immediately follow the first data line). Only the first six entries of the second data line are read if the material type associated with this section is not steel (S). The last six entries are needed only if the material type is steel and if steel stress check post processing is anticipated.

3. If **ITYPE** is **RECT, CIRCLE, PIPE, BOX, I-SECT, C-SECT, T-SECT** or **L-SECT**, the program recognizes the shapes given in Figure V-7 and calculates the section properties from the dimensions **DMAJ, DMIN, TF** and **TW**.

   In calculating the section properties of shapes in Figure V-7 for beams, **DMAJ = DBMAJ + DAMAJ**.

4. If **ITYPE** is a label from the database, the complete property set associated with the section (including the dimensions of the section) is recovered by the program from the data base and assigned to the section property identification number.

   The program searches for the label first in the file SECTIONS.PRO and then in any file specified through **PROPFILE** in the Main Control Data, Section 1.d.

5. If **ITYPE** is **VARIABLE**, the user defines variable section properties along the member length, by referring to section properties already defined, for different segments along the member. Up to three different segments along the member length can be defined. This information is provided on the second data line (the second data line must immediately follow the first data line).

   The material properties for this variable section type are as defined for this type. The geometric properties are obtained from section properties **ID1, ID2, ID3** and **ID4** which must have been defined earlier and may not in themselves be of variable type. The lengths **L1, L2** and **L3** are relative to the clear length of the member and must add up to 1.0.

   The **IV** code determines the variation of properties along the segment lengths. **IV** = 0 means all geometric properties are constant along the segment length and are discontinuous between segments. **ID4** if specified for this case is ignored. **IV** = 1 means all geometric properties vary linearly along the segment length and are continuous between segments. IV = 2 and IV = 3 varies the moment of inertia both for major and minor bending parabolically and cubically, respectively, along the segment lengths. All other properties vary linearly and the properties are continuous between segments. See Figure V-11 for examples of uses of this option.

6. This entry references the material property types previously defined in Section 4 and must not be less than 1 and not greater than **NMAT**.

7. In addition to being used by the automatic section property calculation options, the beam dimensions, **DBMAJ, DAMAJ** and **DMIN** are also used for determining the lengths of the rigid zones on the ends of the columns that support the beams defined with this property set. Beam depths recovered from the data base are assigned to **DBMAJ** (**DAMAJ** is set to 0) unless these values are specified. See Figure V-10.

8. The reduction factors are factors applied to the moment of inertias and torsional constants specified, calculated or recovered from a database. They provide an easy method of modifying the bending or torsional stiffness to account for different modeling conditions, for example, cracking in concrete. If left blank or specified as zero they default to 1.0.

9. A shear area of pure zero will cause the program to exclude the effect of shear deformations. In other words, the shear deformations will be assumed to be zero. Effectively, a pure zero shear area is defaulted to an infinite shear area by the program. Formulas for calculating the shear areas of typical sections are given in Figure V-9.

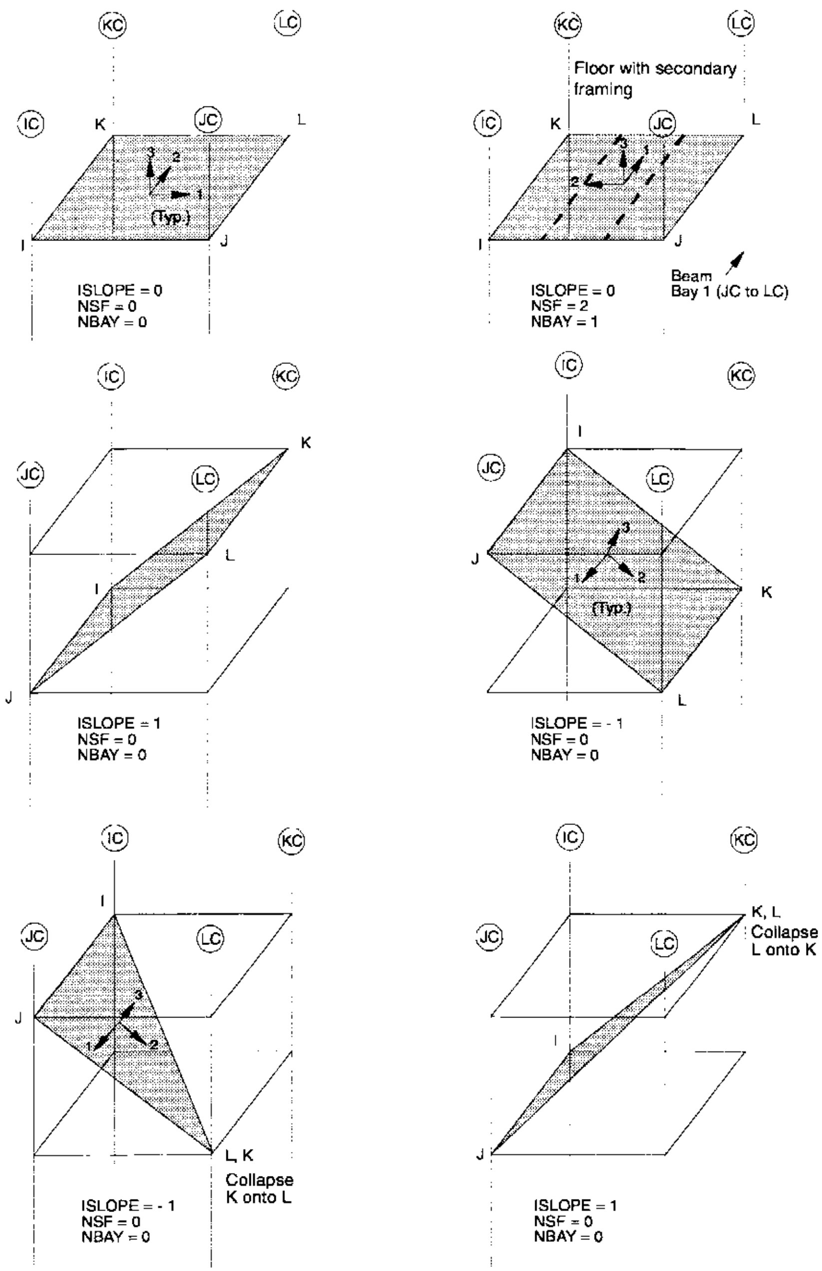
5.iii. Floor Properties

If **NFP** is 0, skip this data section. Otherwise, provide **NFP** data lines to define the **NFP** floor section property types.

Format

Prepare one line of data for each floor property type in the following form:

ID ITYPE IMAT T11 T22 T12



Floor Element Orientation
Figure V-12

Description

Variable	Field	Note	Entry
ID	1	(1)	Identification number of floor section property set.
ITYPE	2	(2)	Section type: = MEMB
IMAT	3	(3)	Material identification number for this section property.
T11	4	(2)	Equivalent floor thickness in element local direction 1.
T22	5		Equivalent floor thickness in element local direction 2.
T12	6		Equivalent floor thickness for shear.

Notes

1. The floor section property numbers must start with the number 1 and proceed in an ascending, consecutive sequence.

2. The only floor element type available is the membrane. That is, it does not have any out of plane stiffness.

   The size of the floor element is recovered from its connectivity. The isoparametric formulation can account for orthotropic properties of the floor element. Three different values of thickness are allowed, one for the element 1-direction, one for the element 2-direction and one for shear. If both **T22** and **T12** are not specified they are taken equal to **T11**. The self weight of the element is based on thickness **T11**. For floor element orientation see Figure V-12.

3. This entry references the material property types previously defined in Section 4 and must not be less than 1 and not greater than **NMAT**.

5.iv. Brace Properties

If **NBRP** is 0, skip this data section. Otherwise, provide **NBRP** data sets to define the **NBRP** brace property data sets.

Format

Each data set consists of a first data line immediately followed by a possible second data line.

a. First Data Line

Prepare the first data line in the following form:

ID ITYPE IMAT DMAJ DMIN TF TW RJ RIMAJ RIMIN

b. Second Data Line

This data line is only needed if the entry for **ITYPE** on the first data line is equal to USER. If this line is needed, it should be prepared in the following form:

A AMAJ AMIN J IMAJ IMIN SMAJ SMIN ZMAJ ZMIN RMAJ RMIN

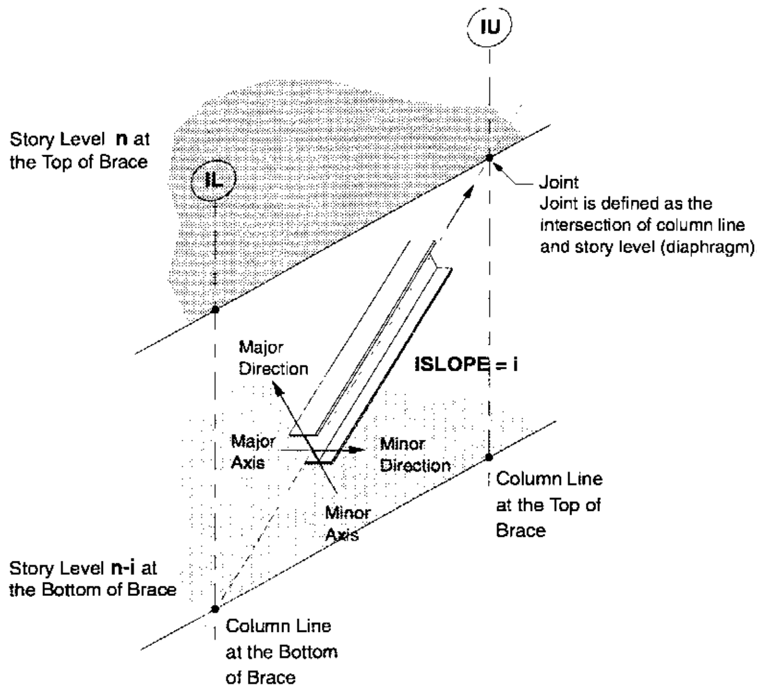
Description

Variable	Field	Note	Entry
a. First Data Line			
ID	1	(1)	Identification number of brace section property set.
ITYPE	2		Section type:
		(2)	= USER
		(3)	= RECT
			= CIRCLE
			= PIPE
			= BOX
			= I-SECT
			= C-SECT
			= T-SECT
			= L-SECT
		(4)	= W14x230 or other label from database
IMAT	3	(5)	Material identification number for this section property.
DMAJ	4	(3,6)	Section dimension in major direction.
DMIN	5	(3,6)	Section dimension in minor direction.
TF	6	(3)	Flange thickness.
TW	7	(3)	Web thickness.
RJ	8	(7)	Reduction factor for torsional constant.
RIMAJ	9		Reduction factor for moment of inertia about major axis.
RIMIN	10		Reduction factor for moment of inertia about minor axis.

Variable	Field	Note	Entry

b. Second Data Line For ITYPE = USER

Variable	Field	Note	Entry
A	1	(2)	Cross-sectional axial area..
AMAJ	2	(8)	Shear area corresponding to major direction shear forces.
AMIN	3	(8)	Shear area corresponding to minor direction shear forces.
J	4		Torsional constant.
IMAJ	5		Moment of inertia, about major axis.
IMIN	6		Moment of inertia, about minor axis.
SMAJ	7		Section modulus, about major axis.
SMIN	8		Section modulus, about minor axis.
ZMAJ	9		Plastic modulus, about major axis.
ZMIN	10		Plastic modulus, about minor axis.
RMAJ	11		Radius of gyration, about major axis.
RMIN	12		Radius of gyration, about minor axis.



Brace Section Orientation
Figure V-13

Notes

1. The brace section property numbers must start with the number 1 and proceed in an ascending, consecutive sequence.

2. If **ITYPE** is **USER**, the user is to calculate all the section properties and provide them on the second data line (the second data line must immediately follow the first data line). Only the first six entries of the second data line are read if the material type associated with this section is not steel (S). The last six entries are needed only if the material type is steel and if steel stress check post processing is anticipated.

3. If **ITYPE** is **RECT, CIRCLE, PIPE, BOX, I-SECT, C-SECT, T-SECT** or **L-SECT**, the program recognizes the shapes given in Figure V-7 and calculates the section properties from the dimensions **DMAJ, DMIN, TF** and **TW**.

4. If **ITYPE** is a label from the database, the complete property set associated with the section (including the dimensions of the section) is recovered by the program from the data base and assigned to the section property identification number.

   The program searches for the label first in the file SECTIONS.PRO and then in any file specified through **PROPFILE** in the Main Control Data, Section 1.d.

5. This entry references the material property types previously defined in Section 4 and must not be less than 1 and not greater than **NMAT**.

6. The brace section dimensions **DMAJ** and **DMIN** are not used in the determination of any rigid end offsets. For brace element orientation see Figure V-13.

7. The reduction factors are factors applied to the moment of inertias and torsional constants specified, calculated or recovered from a database. They provide an easy method of modifying the bending or torsional stiffness to account for different modeling conditions, for example, cracking in concrete. If left blank or specified as zero they default to 1.0.

8. A shear area of pure zero will cause the program to exclude the effect of shear deformations. In other words, the shear deformations will be assumed to be zero. Effectively, a pure zero shear area is defaulted to an infinite shear area by the program. Formulas for calculating the shear areas of typical sections are given in Figure V-9.

5.v. Panel Properties

If **NPP** is 0, skip this data section. Otherwise, provide **NPP** data lines to define the **NPP** panel property types.

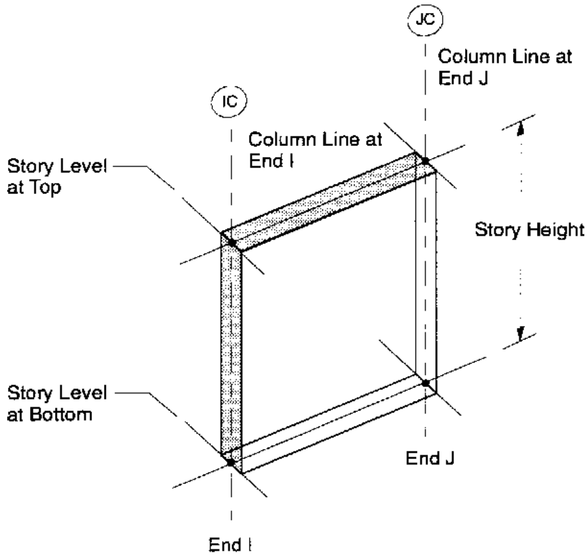
Format

Prepare one line of data for each panel property type in the following form:

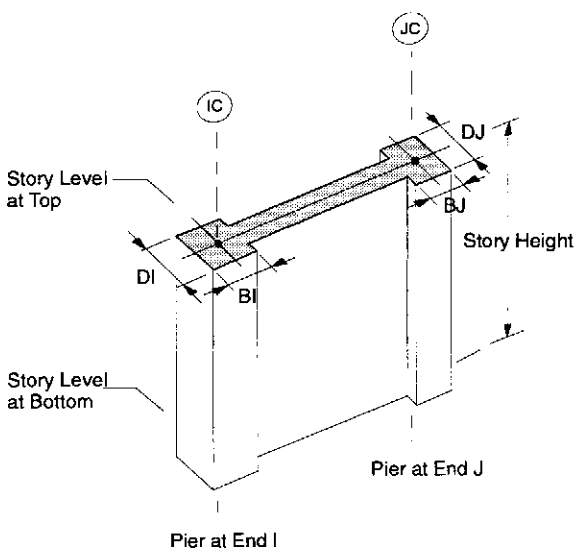
ID ITYPE IMAT TV TH TSHR BI DI BJ DJ

Description

Variable	Field	Note	Entry
ID	1	(1)	Identification number of panel section property set.
ITYPE	2	(2)	Section type: = MEMB
IMAT	3	(3)	Material identification number for this section property.
TV	4	(2)	Equivalent panel thickness for vertical direction loads.
TH	5		Equivalent panel thickness for horizontal direction loads.
TSHR	6		Equivalent panel thickness for shear.
BI	7	(4)	Width of pier at End I.
DI	8		Depth of pier at End I.
BJ	9		Width of pier at End J.
DJ	10		Depth of pier at End J.



Panel Element



Panel Element With End Piers

Panel Section Orientation
Figure V-14

Notes

1. The panel section property numbers must start with the number 1 and proceed in an ascending, consecutive sequence.

2. The only panel element type available is the membrane. That is, it does not have any out of plane stiffness unless piers are specified at its ends. The end piers are treated as column elements providing out of plane bending, shear and torsional stiffness at the corresponding ends and additional in plane bending, shear and axial stiffness.

   The length of the panel, **L**, will be recovered from the locations of the column lines at each end of the panel. The isoparametric formulation can account for orthotropic properties of the panel element. Three different values of thickness are allowed, one for vertical direction stiffness (**TV**), one for horizontal direction stiffness (**TH**) and one for shear (**TSHR**). If both **TH** and **TSHR** are not specified (or left as zeros) they are assumed equal to **TV**. The self weight and mass of the element is based on thickness **TV**. For panel element orientation see Figure V-14.

3. This entry references the material property types previously defined in Section 4 and must not be less than 1 and not greater than **NMAT**.

4. The pair of entries **BI** and **DI** represent the pier at End I. See Figure V-14. If both values are non-zero out of plane bending, shear and torsional stiffness at this end are calculated based on a rectangular column centered at I and in plane bending, shear and axial stiffness are modified. If both values are left as zero the program defaults these to be equal to the thickness **TV**, but only out of plane bending, shear and torsional stiffness are now calculated and this section is assumed to be inside of the column line. If only one value is left as zero the program does not assume any end pier and no out of plane properties exist.

   The same is true for the entries **BJ** and **DJ** as they apply to End J.

5.vi. Spring / Link Properties

If **NSP** is 0, skip this data section. Otherwise, provide **NSP** data sets to define the **NSP** spring / link property types.

Format

Each data set consists of a first data line immediately followed by a second data line. The format of the two lines is dependent on the type of spring being defined.

a. First Data Line

Prepare the first data line in the following form:

for **ITYPE** = LINEAR (i.e., multiaxial elastic spring)

ID ITYPE

or for **ITYPE** = DAMPER (i.e., uniaxial damping)

ID ITYPE IDIR

or for **ITYPE** = GAP (i.e., uniaxial gap)

ID ITYPE IDIR KE

or for **ITYPE** = PLASTIC1 (i.e., uniaxial hysteretic)

ID ITYPE IDIR KE DE

or for **ITYPE** = ISOLATOR1 (i.e., biaxial hysteretic)

ID ITYPE KE2 KE3 DE2 DE3

or for **ITYPE** = ISOLATOR2 (i.e., biaxial friction or friction pendulum)

ID ITYPE KE2 KE3 DE2 DE3

b. Second Data Line

Prepare the second data line in the following form:

for **ITYPE** = LINEAR (i.e., multiaxial elastic spring)

K1 K2 K3 K11 K22 K33

or for **ITYPE** = DAMPER (i.e., uniaxial damping)

C EXPN

or for **ITYPE** = GAP (i.e., uniaxial gap)

K DI

or for **ITYPE** = PLASTIC1 (i.e., uniaxial hysteretic)

K FY RK N BETA

or for **ITYPE** = ISOLATOR1 (i.e., biaxial hysteretic)

K1 K2 K3 FY2 FY3 RK2 RK3

or for **ITYPE** = ISOLATOR2 (i.e., biaxial friction or friction pendulum)

K1 K2 K3 CFF2 CFF3 CFS2 CFS3 A2 A3 R2 R3

Description

Variable	Field	Note	Entry

a. First Data Line For ITYPE = LINEAR

Variable	Field	Note	Entry
ID	1	(1)	Identification number of spring property set.
ITYPE	2	(2)	Spring type: = LINEAR

First Data Line For ITYPE = DAMPER

Variable	Field	Note	Entry
ID	1	(1)	Identification number of spring property set.
ITYPE	2	(2)	Spring type: = DAMPER
IDIR	3	(3)	Direction for damping: = 1 for local axis 1 (axial) = 2 for local axis 2 (major shear) = 3 for local axis 3 (minor shear)

First Data Line For ITYPE = GAP

Variable	Field	Note	Entry
ID	1	(1)	Identification number of spring property set.
ITYPE	2	(2)	Spring type: = GAP
IDIR	3	(3)	Direction for gap and stiffness: = 1 for local axis 1 (axial) = 2 for local axis 2 (major shear) = 3 for local axis 3 (minor shear)
KE	4	(4)	Spring effective stiffness.

First Data Line For ITYPE = PLASTIC1

Variable	Field	Note	Entry
ID	1	(1)	Identification number of spring property set.
ITYPE	2	(2)	Spring type: = PLASTIC1

Variable	Field	Note	Entry
IDIR	3	(3)	Direction for stiffness: = 1 for local axis 1 (axial) = 2 for local axis 2 (major shear) = 3 for local axis 3 (minor shear)
KE	4	(4)	Spring effective stiffness.
DE	5	(5)	Spring effective damping ratio.

First Data Line For ITYPE = ISOLATOR1

Variable	Field	Note	Entry
ID	1	(1)	Identification number of spring property set.
ITYPE	2	(2)	Spring type: = ISOLATOR1
KE2	2	(3,4)	Spring effective stiffness along axis 2.
KE3	3		Spring effective stiffness along axis 3.
DE2	8	(3,5)	Spring effective damping ratio along axis 2.
DE3	9		Spring effective damping ratio along axis 3.

First Data Line For ITYPE = ISOLATOR2

Variable	Field	Note	Entry
ID	1	(1)	Identification number of spring property set.
ITYPE	2	(2)	Spring type: = ISOLATOR2
KE2	2	(3,4)	Spring effective stiffness along axis 2.
KE3	3		Spring effective stiffness along axis 3.
DE2	8	(3,5)	Spring effective damping ratio along axis 2.
DE3	9		Spring effective damping ratio along axis 3.

Variable	Field	Note	Entry

b. Second Data Line For ITYPE = LINEAR

Variable	Field	Note	Entry
K1	1	(3,6)	Spring stiffness along axis 1.
K2	2		Spring stiffness along axis 2.
K3	3		Spring stiffness along axis 3.
K11	4		Rotational spring stiffness around axis 1.
K22	5		Rotational spring stiffness around axis 2.
K33	6		Rotational spring stiffness around axis 3.

Second Data Line For ITYPE = DAMPER

Variable	Field	Note	Entry
C	1	(7)	Viscous damping value.
EXPN	2		Power of deformation velocity to which damping is proportional.

Second Data Line For ITYPE = GAP

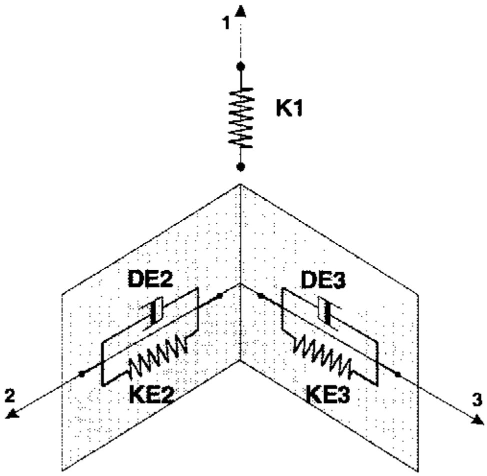
Variable	Field	Note	Entry
K	1	(8)	Spring stiffness after gap closure.
DI	2		Initial gap opening.

Second Data Line For ITYPE = PLASTIC1

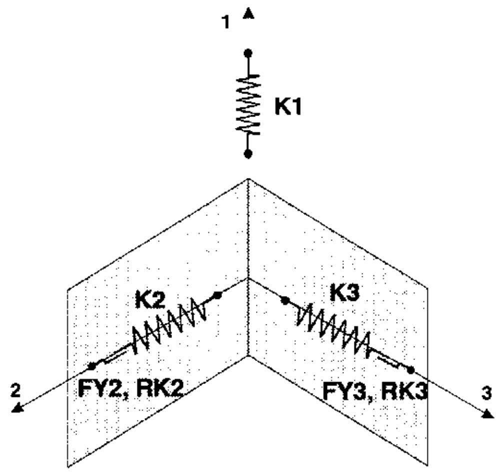
Variable	Field	Note	Entry
K	1	(9)	Initial spring stiffness.
FY	2		Yield force.
RK	3		Post yield stiffness ratio.
N	4		Parameter controlling hysteresis loop shape.
BETA	6		Parameter controlling hysteresis loop shape.

Second Data Line For ITYPE = ISOLATOR1

Variable	Field	Note	Entry
K1	1	(3,10)	Spring stiffness along axis 1.

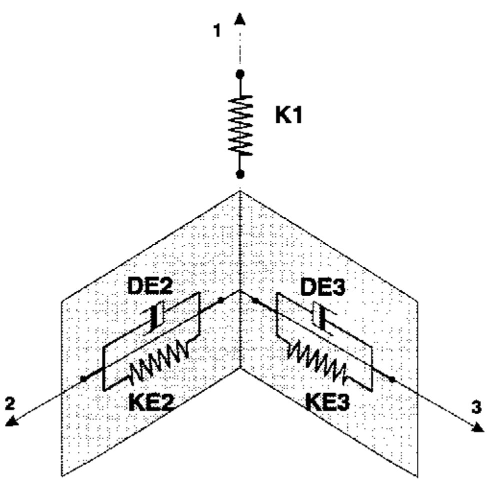


Linear Analysis

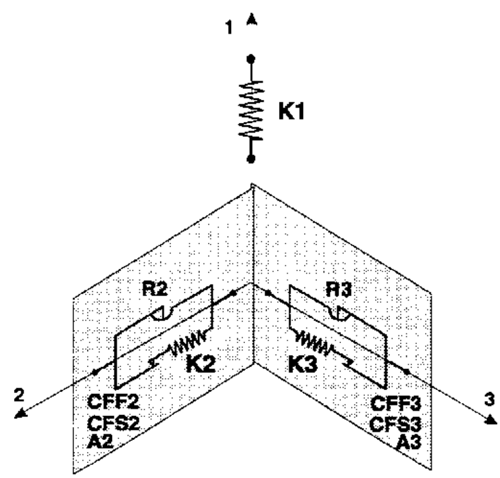


Nonlinear Analysis

ISOLATOR1 Element
(Biaxial Hysteretic)



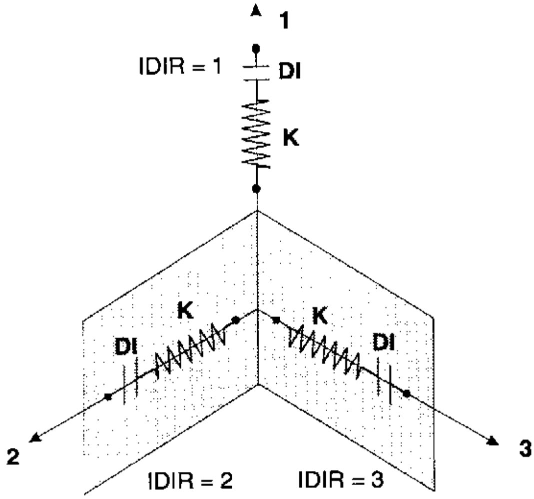
Linear Analysis



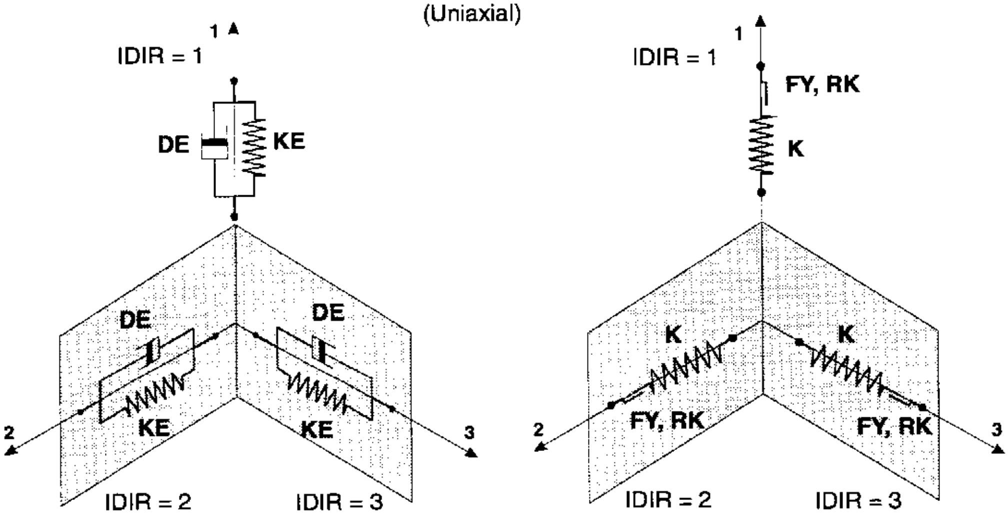
Nonlinear Analysis

ISOLATOR2 Element
(Biaxial Friction or Friction Pendulum))

Spring Support Definitions
Figure V-15 (continued)



GAP Element
(Uniaxial)

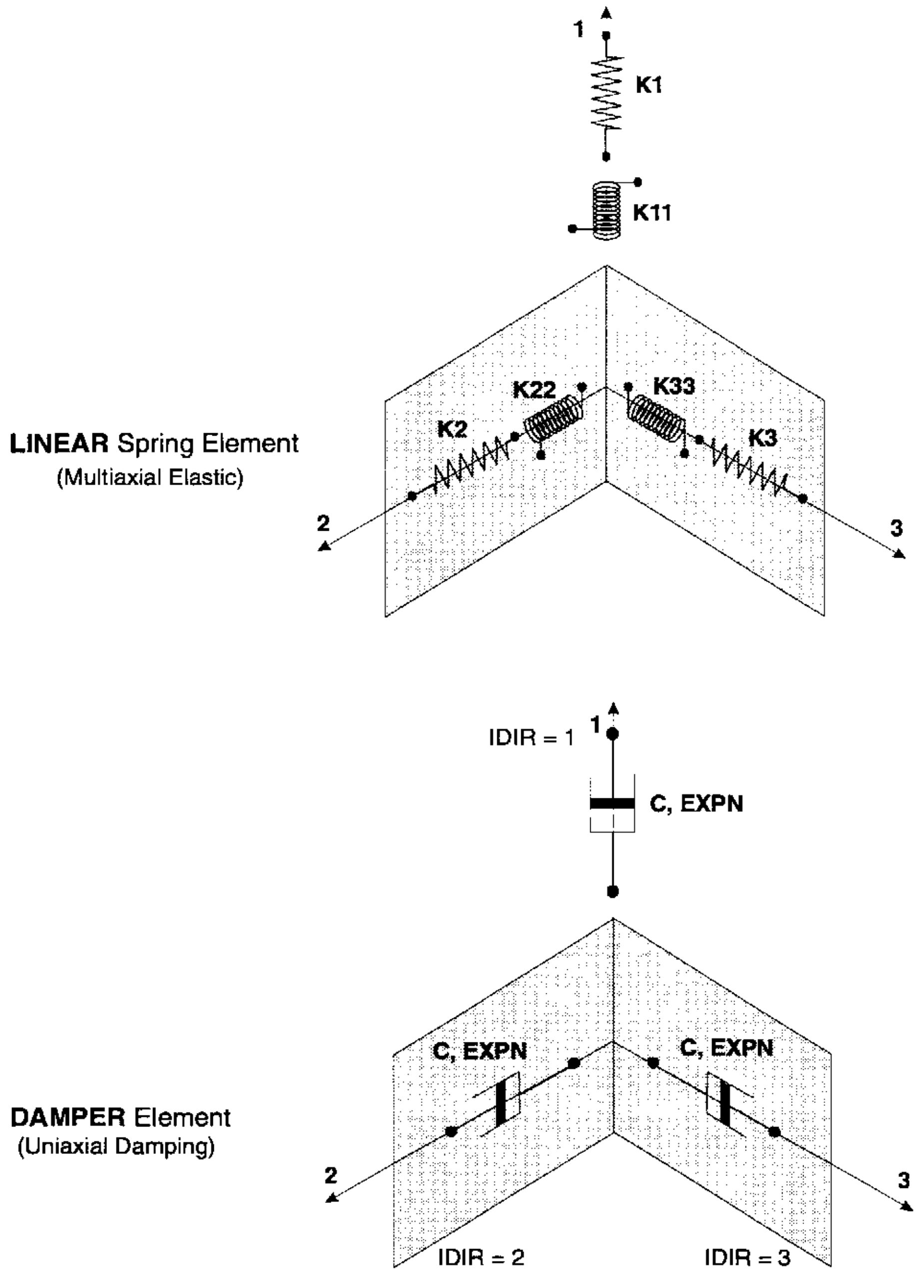


Linear Analysis

Nonlinear Analysis

PLASTIC1 Element
(Uniaxial Hysteretic)

Spring Support Definitions
Figure V-15 (continued)



Spring Support Definitions
Figure V-15

Variable	Field	Note	Entry
K2	2		Initial spring stiffness along axis 2.
K3	3		Initial spring stiffness along axis 3.
FY2	4		Yield force along axis 2.
FY3	5		Yield force along axis 3.
RK2	6		Post yield stiffness ratio along axis 2.
RK3	7		Post yield stiffness ratio along axis 3.

Second Data Line For ITYPE = ISOLATOR2

K1	1	(3,11)	Spring stiffness along axis 1.
K2	2		Initial spring stiffness along axis 2.
K3	3		Initial spring stiffness along axis 3.
CFF2	4		Coefficient of friction at fast velocities along axis 2.
CFF3	5		Coefficient of friction at fast velocities along axis 3.
CFS2	6		Coefficient of friction at slow velocities along axis 2.
CFS3	7		Coefficient of friction at slow velocities along axis 3.
A2	8		Coefficient controlling variation of coefficient of friction with velocity along axis 2.
A3	9		Coefficient controlling variation of coefficient of friction with velocity along axis 3.
R2	10		Radius of contact surfaces in 2-direction.
R3	11		Radius of contact surfaces in 3-direction.

Notes

1. The spring property numbers must start with the number 1 and proceed in an ascending, consecutive sequence.

2. The **ITYPE** parameter describes the type of spring or link property being specified. It also determines the other parameters to be specified. The algorithms used for calculation of force displacement relationships for the different types of springs are detailed in Chapter IV.

3. All properties are specified in the local coordinate system of the spring element. See Figure V-15. The spring properties are used for spring supports and links. For spring supports (from joint to ground) and for zero length links (between two joints spaced less than 10^{-6} apart) the local 1, 2 and 3 axes correspond to the column local 1, 2 and 3 directions, respectively. For links with non zero length the local 1 axis is along the link. The local 2 axis is in the vertical plane unless the link is vertical in which case the axes correspond to the column axes.

4. For the case of nonlinear springs, i.e., **ITYPE** = GAP, PLASTIC1, ISOLATOR1 or ISOLATOR2 the program allows for effective stiffness properties to be input for linear analysis. When carrying out UBC '94 based code analysis the effective stiffness should be the code defined maximum effective stiffness.

   Linear static analysis for all gravity and lateral loads, response spectrum analysis and linear time history analysis all use the effective stiffness of the springs or links and the nonlinear properties are ignored. For **ITYPE** = ISOLATOR1 or ISOLATOR2 elements the effective stiffness along the 1-axis is taken as **K1** specified in the next line for the stiffness along the 1-axis.

   Nonlinear time history analysis calculates the modes based on the effective stiffness but during time integration uses the actual values, specified by the parameters in the next line.

5. For the case of nonlinear energy dissipating springs, i.e., **ITYPE** = PLASTIC1, ISOLATOR1 or ISOLATOR2 the program allows for effective damping ratios to be input. When carrying out UBC '91 based code analysis the effective damping ratio should be determined as per the code definition.

   For response spectrum and linear time history analysis the effective damping values are converted into modal damping values assuming proportional damping, i.e., the cross coupling damping terms are ignored. These modal damping values are added to any other modal damping specified directly and any calculated from damping elements.

Nonlinear time history analysis does not use effective damping values as it accounts for energy dissipation in the elements directly.

6. The parameters describe a multiaxial spring. Each direction of spring stiffness is uncoupled from the other, except in the case of non-zero length links. In which case the **K2** and **K33** stiffness are coupled and the **K3** and **K22** stiffness are coupled.

7. The parameters describe a uniaxial damping element. **C** and **EXPN** must both be positive and represent nonlinear viscous damping of the form, $F = CV^{EXPN}$.

   For response spectrum and linear time history analysis the **C** value is converted into modal damping value assuming linear and proportional damping, i.e., **EXPN** is assumed to be 1.0 and the cross coupling damping terms are ignored. These modal damping values are added to any other modal damping specified directly and any calculated from effective damping values specified for the other energy dissipating elements.

   Nonlinear time history analysis does not convert the **C** value to modal damping but includes it explicitly in the dynamic equilibrium equations. Nonlinear and non-proportional damping is, therefore, accounted for.

8. The parameters describe a uniaxial gap element. **K** must be positive and represents a linear spring stiffness once the gap is closed. **DI** is the initial opening in the gap element. A negative value of **DI** represents a closed gap with initial precompression of **DI** times **K**.

9. The parameters describe a uniaxial nonlinear spring. Details of the hysteresis model are given in Chapter IV.

   The parameters **K** and **FY** must be positive, **RK** can be positive or zero, **N** must be greater than or equal to one (default is 2), **BETA** must be a number between 0 and 1 (default is 0.5).

10. The stiffness in the 1-direction is linear elastic for all levels of force. In the 2 and 3-directions a biaxial nonlinear spring is modeled. Details of the hysteresis model and the coupling between the directions are given in Chapter IV.

    All parameters on this line must be positive, except the post yield stiffness ratios **RK2** and **RK3** which can be zero.

11. The stiffness in the 1-direction is linear elastic for all levels of force. In the 2 and 3-directions a friction pendulum is modeled. The friction force is dependent on the force computed in the 1-direction. Details of the hysteresis model and the coupling between the directions are given in Chapter IV.

    All parameters on this line must be positive, except radii **R2** and **R3** which can be zero. A zero specification for radius **R2** or **R3** is interpreted as an infinite radius, i.e., the surface is assumed to be flat in that direction.

6. Frame Data Sets

Prepare one set of frame data for each of the different frame types in the structure. Frames with different locations but identical properties and loading need only be entered once. A total of **NDF** data sets need to be prepared. At least one set of frame data (Sections 6.i. through 6.xvi.) is always needed.

6.i. Frame Control Data

Format

a. Frame Heading

Prepare one line of data having up to 80 characters of information for identification of this frame type.

b. Frame Control Data

Prepare one line of data in the following form:

NFID NC NB NF NJLP NBLP NFLP IJ MBR MPAN MLNK MCONL

Description

Variable	Field	Note	Entry
b. Frame Control Data			
NFID	1	(1)	Frame type identification number.
NC	2	(2)	Number of vertical column lines in frame. Must be at least 1.
NB	3	(3)	Number of beam bays in frame.
NF	4	(4)	Number of floor bays in frame.
NJLP	5	(5)	Number of joint load patterns in frame.
NBLP	6	(6)	Number of beam load patterns in frame.
NFLP	7	(7)	Number of floor load patterns in frame.
IJ	8	(8)	Code for joint assignments: = 0 No assignments = 1 Assignments present.
MBR	9	(9)	Maximum number of brace elements in frame.
MPAN	10	(10)	Maximum number of panel elements in frame.
MLNK	11	(11)	Maximum number of link elements in frame.
MCONL	12	(12)	Maximum number of point loads in any one beam span loading pattern.

Notes

1. Frame identification numbers must be input in ascending, consecutive sequence starting with the number 1. This frame may be located (repeated) at different positions in the structure through the frame location data. See Section 7 below.

2. This entry controls the number of lines read in the column line coordinate and orientation data in Section 6.ii. below. Beams, floor elements, panels and bracing elements need at least two column lines for definition. Therefore, a single column line frame can have no beams, floor elements, panels or braces.

3. This entry controls the number of lines read in the beam bay connectivity data in Section 6.iii. below. If **NC** is 1, **NB** must be zero, however, **NB** can be zero even if **NC** is greater than 1.

4. This entry controls the number of lines read in the floor bay connectivity data in Section 6.iv. below. If **NC** is less than 3, **NF** must be zero, however, **NF** can be zero even if **NC** is greater than 1.

5. This entry controls the number of joint load patterns that are to be defined in Section 6.v. below. If there are no loaded joints in the frame, enter zero for this number and skip Sections 6.v. and 6.xv.

6. This entry controls the number of different beam load patterns that are to be defined in Section 6.vi. below. If no vertical loads are applied to the beams directly or if this frame has no bays, enter zero for this number and skip Sections 6.vi. and 6.xvi.

7. This entry controls the number of different floor load patterns that are to be defined in Section 6.vii. below. If no loads are applied to the floor bays or if this frame has no floor bays, enter zero for this number and skip Sections 6.vii. and 6.xvii.

8. This code controls the reading of joint-diaphragm connectivity and spring support data in Section 6.viii. If **IJ** is 0 it is assumed that all joints are connected to diaphragm number 1 at all levels and that no spring supports are provided. If **IJ** is 1 the joint assignment data in Section 6.viii. is read and must be provided.

9. This entry defines the maximum number of brace elements that are input (and generated) in Section 6.xii below. An exact count of the number of brace elements is not needed. However, this number must be larger than the number of braces actually input or generated. If **NC** is 1, or if no brace elements are present this entry must be 0. If **MBR** is 1 or higher the brace location and assignment data in Section 6.xii. is read and must be provided.

10. This entry defines the maximum number of panel elements that are input (and generated) in Section 6.xiii. below. An exact count of the number of panel elements is not needed. However, this number must be larger than the number of panel elements actually input or generated. If **NC** is 1 or if no panel elements are present, this entry must be 0. If **MPAN** is 1 or higher the panel location and assignment data in Section 6.xiii. is read and must be provided.

11. This entry defines the maximum number of link elements that are input (and generated) in Section 6.xiv. below. An exact count of the number of link elements is not needed. However, the number must be larger than the number of links actually input or generated. If no link elements are present this entry must be 0. If **MLNK** is 1 or higher the link location and assignment data in Section 6.xiv. is read and must be provided.

12. This number is required to preallocate memory storage for the concentrated loads that may exist in the beam span loading patterns.

6.ii. Column Line Coordinates and Orientation

This data section is always required. Prepare a total of **NC** data lines to define the column line coordinates (in the frame local coordinate system) and orientation for the corresponding **NC** column lines that exist in the frame. See Figure V-16.

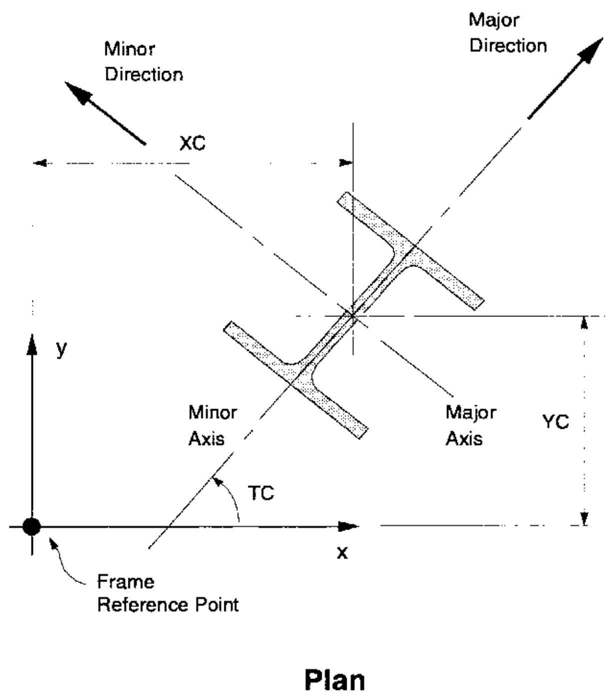
Format

The data should be prepared in the following form:

N XC YC TC

Description

Variable	Field	Note	Entry
N	1	(1)	Column line identification number.
XC	2		X-distance to column line from frame reference point.
YC	3		Y-distance to column line from frame reference point.
TC	4		Orientation angle (in degrees) with the column major direction (not major axis).



Column Orientation
Figure V-16

Notes

1. The column line identification numbers must be input in an ascending, consecutive sequence starting with the number 1.

6.iii. Beam Bay Connectivity

If **NB** is 0, no beam bays exist in the frame. Therefore skip this data section. Otherwise, prepare a total of **NB** data lines to define the column lines at the two ends of each of the **NB** beam bays that exist in the frame. See Figure V-17.

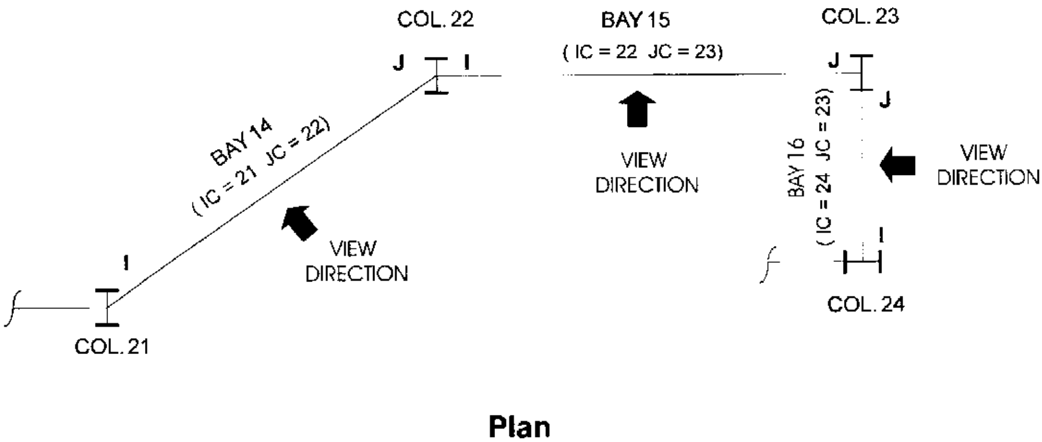
Format

The data should be provided in the following form:

N IC JC ISLOPE

Description

Variable	Field	Note	Entry
N	1	(1)	Beam Bay identification number.
IC	2	(2)	Column line number at End I.
JC	3		Column line number at End J.
ISLOPE	4	(3)	Code for defining sloping element: = 0 No slope = 1 Sloping element. End I lower. = -1 Sloping element. End J lower.



Beam Bay Connectivity
Figure V-17

Notes

1. The beam bay identification number must be input in an ascending, consecutive sequence starting with the number 1.

2. The user must establish a direction to view the bay. The end of the bay to the left of the user is then End I and the end of the bay to the right of the user is then the End J. Once the view direction is established it should be consistently followed for the preparation of all the data associated with all the beams in a particular bay.

3. If this entry is 1 the **IC** node of the element is assumed to be one level below. The element can then be used instead of braces. This option is useful in modeling ramps when gravity loads need to be specified on sloping members.

6.iv. Floor Bay Connectivity

If **NF** is 0, no floor bays exist in the frame. Therefore, skip this data section. Otherwise, prepare a total of **NF** data lines to define the four column lines defining the floor bay, and the number and direction of any secondary framing for each of the **NF** floor bays that exist in the frame. See Figure V-12.

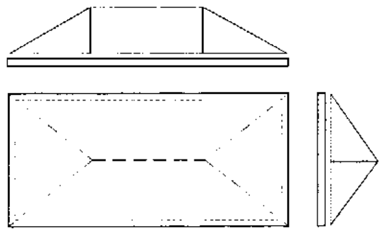
Format

The data should be provided in the following form:

N IC JC KC LC ISLOPE NSF NBAY

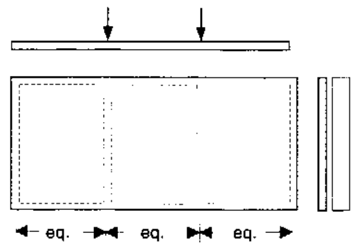
Description

Variable	Field	Note	Entry
N	1	(1)	Floor bay identification number.
IC	2	(2)	Column line number at corner I.
JC	3		Column line number at corner J.
KC	4		Column line number at corner K.
LC	5		Column line number at corner L.
ISLOPE	6	(3)	Code for defining sloping element: = 0 No slope = 1 Sloping element. Corners I and J lower. = -1 Sloping element. Corners K and L lower.
NSF	7	(4)	Number of secondary framing members in this floor bay.
NBAY	8	(4,5)	Beam bay number defining local 1-axis and direction of secondary framing.



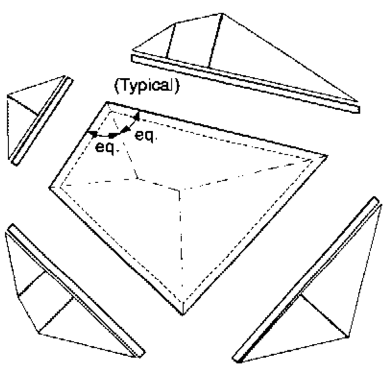
Beams on All Four Edges

NSF = 0



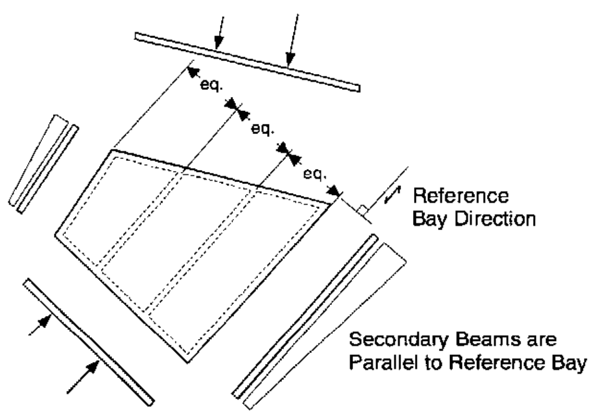
Beams on All Four Edges

NSF = 2



Beams on All Four Edges

NSF = 0



Beams on All Four Edges

NSF = 2

Examples of Tributary Beam Loading Algorithms
Figure V-18

Notes

1. The floor bay identification number must be input in an ascending, consecutive sequence starting with the number 1.

2. The floor bay connectivity must be defined as shown in Figure V-12, i.e., **IC**, **JC**, **LC** and **KC** must be in a counter clockwise order when viewed from above.

   The four nodal bay can be degenerated into a triangular bay by specifying **LC** the same as **KC**.

3. If this entry is 1 the **IC** and **JC** nodes of the element are assumed to be one level below. This option is useful in modeling ramps and other sloping floors.

4. **NSF** and **NBAY** information is used to convert floor bay vertical loading to beam span vertical loads. See Figure V-18 for examples of algorithms used. All secondary framing is assumed to be simply supported at the beams and they load the beams with point loads. If **NSF** is not 0 then beams must frame the floor element on all four sides. For irregular or more complex secondary member framing, or in cases where the floor bay is not completely framed by beams, it is recommended that the user provide beam span loading directly.

   NBAY can be any beam bay in the structure. The direction of the secondary framing is taken as parallel to the beam bay number **NBAY**.

   All lateral load specified on the floor element is converted to joint loads on a tributary basis.

5. **NBAY** is also used to define the local axis direction for the floor bay. See Figure V-12. The direction of beam bay number **NBAY** defines the local axis 1. If **NBAY** is not specified the direction **IC**-**JC** defines the local axis 1 for the floor bay. Local axis 2 is defined by taking cross product of local axis 3, (which is always the upward normal), with local axis 1. The local axis definitions are important if orthotropic properties for the floor are to be specified.

6.v. Joint Load Patterns

If **NJLP** is 0, no joint load patterns need to be defined. Therefore skip this data section. Otherwise, provide **NJLP** data lines to define the load patterns. Loads act at the joints to which they are applied. These patterns are applied to the joints in Section 6.xv. below.

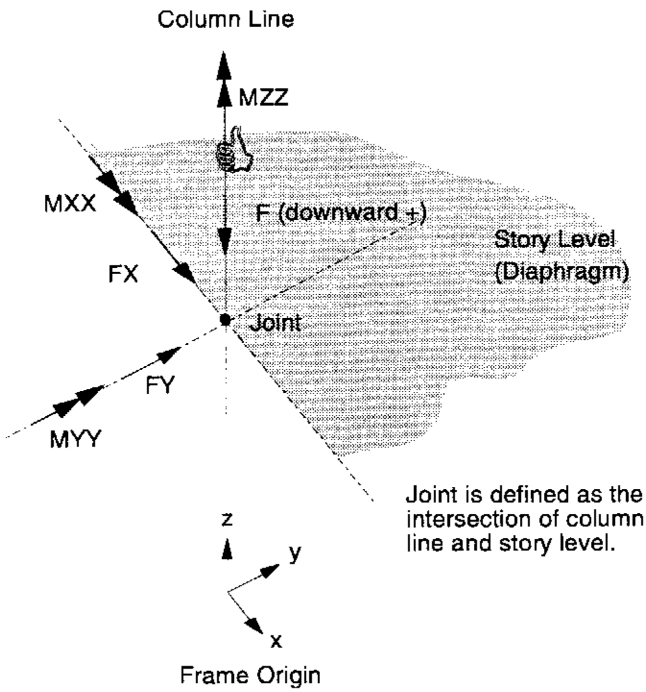
Format

The data should be provided in the following form:

N F FX FY MXX MYY MZZ TEMP

Description

Variable	Field	Note	Entry
N	1	(1)	Joint load pattern identification number.
F	2	(2)	Force acting vertically downwards.
FX	3		Force in the frame local x-direction.
FY	4		Force in the frame local y-direction.
MXX	5		Moment about the frame local x-axis
MYY	6		Moment about the frame local y-axis.
MZZ	7		Moment about the z-axis (vertical).
TEMP	8	(3)	Temperature increase above zero stress state.



Joint Load Sign Convention
Figure V-19

Notes

1. The identification numbers must be input in an ascending, consecutive sequence starting with the number 1.

2. These forces are input in the frame local coordinate system. See Figure V-19 for positive directions.

3. The temperature increase is used to calculate temperature stresses for all elements (column, beam, brace, panel and floor) framing into the joint where this loading pattern is applied.

6.vi. Beam Load Patterns

If **NBLP** is 0, no vertical load is to be defined for any of the beams of this frame. Therefore skip this data section. Otherwise, provide one set of data for each of the **NBLP** beam load patterns of this frame. See Figure V-20. These loads are all vertical and act on the beams to which they are applied. These patterns are applied to the beams in Section 6.xvi. below.

Format

Each data set consists of a first line followed by data lines for the point load data (if needed).

a. First Data Line

Prepare one data line in the following form:

N NCON W1 L1 W2 L2 W3 L3 W4 L4

b. Point Load Data

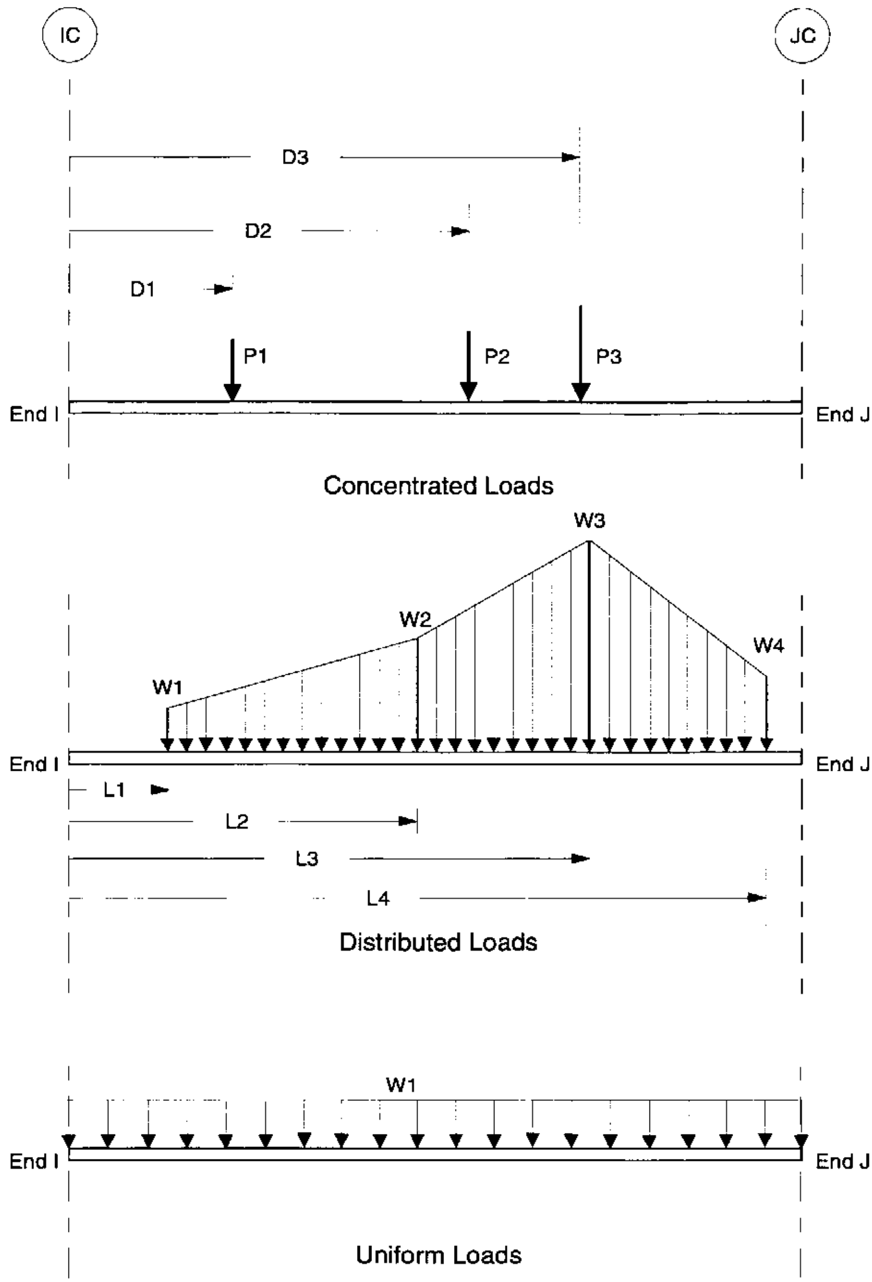
If **NCON** is 0, skip this data section. Otherwise, provide up to four pairs of numbers (**D,P**) on each line to define the concentrated loads in sequence starting with the load closest to End I in the following form:

D1 P1 D2 P2 D3 P3 D4 P4

Skip to the next line if the number of point loads is greater than 4, and so forth.

Description

Variable	Field	Note	Entry
a. First Data Line			
N	1	(1)	Identification number for this beam load set.
NCON	2	(2)	Number of concentrated point loads.
W1	3	(3)	Load intensity at station 1.
LI	4		Distance of station 1 from End I.
W2	5		Load intensity at station 2.
L2	6		Distance of station 2 from End I.
W3	7		Load intensity at station 3.
L3	8		Distance of station 3 from End I.
W4	9		Load intensity at station 4.
L4	10		Distance of station 4 from End I.
b. Point Load Data			
D1	1	(4)	Distance of load 1 from End I.
P1	2		Magnitude of load 1.
D2	3		Distance of load 2 from End I.
P2	4		Magnitude of load 2.
D3	5		Distance of load 3 from End I.
P3	6		Magnitude of load 3.
D4	7		Distance of load 4 from End I.
P4	8		Magnitude of load 4.



Beam Span Loading
Figure V-20

Notes

1. The identification numbers must be input in an ascending, consecutive sequence starting with the number 1.

2. **NCON** should be less than or equal to **MCONL**. **NCON** controls the number of concentrated load entries to be made in the point load data below this first data line.

3. Uniform, partial uniform, trapezoidal or triangular loading can be applied to beams using this option. Loads are assumed to vary linearly between stations. If only **W1** is specified it is assumed to be uniform. See Figure V-20. Positive loads act vertically downwards and are specified as per unit length of the beam. The distances **L1**, **L2**... which define distances of stations will be interpreted as a ratio of the total bay length if **L1**, **L2**... are less than 0 (negative), and actual distances measured from column center line at End I if **L1**, **L2**... are greater than 0.

4. Positive loads act vertically downward. The distances **D1, D2**, . . . which define the points of load application will be interpreted as a ratio of the total bay length if **D1, D2**, . . . are less than 0 (negative), and actual distances measured from column center line at End I if **D1, D2**, . . . are greater than 0.

6.vii. Floor Load Patterns

If **NFLP** is 0, no load is to be defined for any of the floor elements of this frame. Therefore skip this data section. Otherwise, provide **NFLP** data lines to define the **NFLP** floor load patterns. These loads act on the joints and beams framing the floor element to which they are applied. These loads are applied to the floor elements in Section 6.xvii. below.

Format

Provide data in the following form:

N W WX WY

Description

Variable	Field	Note	Entry
N	1	(1)	Floor load pattern identification number.
W	2	(2)	Uniform load per unit area acting downward.
WX	3	(3)	Uniform load per unit area acting in frame local x-direction.
WY	4		Uniform load per unit area acting in frame local y-direction.

Notes

1. The identification numbers must be input in an ascending, consecutive sequence starting with the number 1.

2. The uniform vertical loads specified here for the floor elements are converted to beam span loads and are applied to the beams at the edges of the floor element. See Figure V-18 for the algorithms used. The algorithms assume that all edges of the floor element are supported on beams. If a beam does not exist at a certain edge the tributary load associated with that edge is lumped at the corners.

3. The uniform lateral loads specified here for the floor elements are converted to joint loads on a tributary area basis.

6.viii. Joint Assignments (Diaphragm and Springs)

If **IJ** is 0, skip this data section completely (including the blank termination line described below). In this case the program assumes that all joints are connected to rigid diaphragm number 1 at the respective stories and that no spring supports exist at the joints.

If **IJ** is 1, then provide in this section as many data lines as needed to assign rigid diaphragm connectivity and the spring support properties at the joints of the structure. Joints that are connected to rigid diaphragm number 1 and have no spring supports need not be assigned. Property assignments for any joint may be repeated. The last values read (or generated) will be used. **End this data section with a blank line.**

Format

Prepare the data in the following form:

NC1 NC2 NSAME SD1 SD2 IDIA ISPR

Description

Variable	Field	Note	Entry
NC1	1	(1)	Column line number of first joint being assigned.
NC2	2	(1)	Column line number of last joint being assigned.
NSAME	3	(2)	Column line number, the joint assignments of which are to be repeated at column lines **NC1** through **NC2**.
SD1	4	(3)	Identification of the first story level associated with the joint being assigned.
SD2	5	(3)	Identification of the last story level associated with the joint being assigned.
IDIA	6	(4)	Diaphragm identification number: < 0 Joint **not** part of any rigid diaphragm at levels **SD1** to **SD2**, however, its mass is tributary to the total mass of rigid diaphragm number **-IDIA** at levels **SD1** to **SD2.** $= 0$ Joint **not** part of any rigid diaphragm at levels **SD1** to **SD2**. > 0 Joint part of rigid diaphragm number **IDIA** at levels **SD1** to **SD2**.
ISPR	7	(5)	Spring property identification number for springs at this joint.

Notes

1. This entry is a column line number. This number must be positive and not greater than **NC**. Also **NC2** may not be less than **NC1**.

2. If **NSAME** is non zero, it is a column line number, the joint diaphragm connectivity and spring support properties of which the user has already assigned in preceding data lines of this data section. The non zero entry for **NSAME** puts the program into a duplication mode. In this mode the joint diaphragm connectivity and spring properties of the joints at all levels on column lines **NC1** to **NC2** are set identical to that of column line **NSAME** as it stands assigned at the time of this entry.

   Subsequent reassignment of joint properties of column line **NSAME** in later data lines will not result in automatic corresponding reassignment of the joint properties on column lines **NC1** or **NC2**, or vice versa.

   In this duplication mode (i.e. when **NSAME** is non zero), the entries for **SD1, SD2, IDIA** and **ISPR** on this data line are meaningless and must not be entered. These entries are only used if **NSAME** is 0.

3. This entry is the alphanumeric story identifier. The story level identifiers have been previously defined in Section 3.

4. This entry specifies the diaphragm number of the rigid diaphragm to which the joints being specified are attached. If the diaphragm number is specified as a positive number the joint is rigidly attached to the diaphragm. The lateral displacements of the joints are then dependent on the lateral displacements of this diaphragm and all mass tributary to the joints become part of the mass on this diaphragm. If the diaphragm number is specified as a negative number the joint is not rigidly attached to the diaphragm, however, any mass tributary to the joints still become part of the mass on this diaphragm. If the joint is independent of all rigid diaphragms, **IDIA** should be set to 0, in this case any mass tributary to this joint will not be part of the analysis.

   All joints are automatically assumed to be rigidly connected to diaphragm number 1 unless this data is specified.

5. This entry refers to the spring property table defined in Section 5.vi. above. This entry may never be negative and must not be greater than **NSP**. Missing elements may be reassigned as having a property set identification number of zero. All springs are external springs (grounded).

6.ix. Column Assignments

This data section is always required. In this section provide as many data lines as needed to define the column property assignment on the column lines of this frame. If the property of any column at any level is not defined, it will be assumed to be 0. Therefore, zero (or dummy) columns need not be defined. Property assignments for any column may be repeated. The last values read (or generated) will be used. **End this data section with a blank line.**

Format

Prepare the data in the following form:

NC1 NC2 NSAME SD1 SD2 ID IPMAJ IPMIN

Description

Variable	Field	Note	Entry
NC1	1	(1)	Column line number of first column line being assigned.
NC2	2	(1)	Column line number of last column line being assigned.
NSAME	3	(2)	Column line number, the assignments of which are to be repeated at column lines **NC1** through **NC2**.
SD1	4	(3)	Identification of the first story level associated with the column being assigned.
SD2	5	(3)	Identification of the last story level associated with the column being assigned.
ID	6	(4)	Column property identification number for columns being assigned.
IPMAJ	7	(5)	Control for major direction pin conditions: = 0 continuity at both ends. = 1 Pin at bottom end of column. = 2 Pin at top end of column. = 3 Pin at both ends of column.
IPMIN	8	(5)	Control for minor direction pin conditions: = 0 continuity at both ends. = 1 Pin at bottom end of column. = 2 Pin at top end of column. = 3 Pin at both ends of column.

Notes

1. This entry is a column line number. This number must be positive and not greater than **NC**.

2. If **NSAME** is non zero, it is a column line number, the column property and pin end condition assignments of which the user has already defined in preceding data lines of this data section. The non zero entry for **NSAME** puts the program into a duplication mode. In this mode the column properties and the pin end conditions of the column elements at all levels on column lines **NC1** to **NC2** are set identical to that of column line **NSAME** as it stands assigned at the time of this entry.

   Subsequent reassignment of properties to column line **NSAME** in later data lines will not result in automatic corresponding reassignment of the properties on column lines **NC1** or **NC2**, or vice versa.

   In this duplication mode (i.e. when **NSAME** is non zero), the entries for **SD1, SD2, ID, IPMAJ** and **IPMIN** on this data line are meaningless and must not be entered. These entries are only used if **NSAME** is 0.

3. This entry is the alphanumeric story identifier. The story level identifiers have been previously defined in Section 3. The story level associated with a column element is the story level at the top of the column.

4. This entry refers to the column section property table defined in Section 5.i. above. This entry may never be negative and must not be greater than **NCP**. Missing elements may be reassigned as having a property set identification number of zero.

5. The pins created at the ends of the columns set the major and minor bending moments at the corresponding ends to 0. The torsional moments are not affected.

6.x. Beam Assignments

If **NB** is 0, there are no bays defined in this frame. Therefore skip this data section completely (including the blank termination line described below). Otherwise, provide as many data lines as needed to define the beam property assignment in the bays of this frame. If the property of any beam in any bay at any level is not defined, it will be assumed to be 0. Therefore, zero (or dummy) beams need not be defined. Property assignments for any beam may be repeated. The last values read or generated will be used. **End this data section with a blank line.**

Format

Prepare the data in the following form:

NB1 NB2 NSAME SD1 SD2 ID IPMAJ IPMIN IPI IPJ

Description

Variable	Field	Note	Entry
NB1	1	(1)	Bay number of first beam bay being assigned.
NB2	2	(1)	Bay number of last beam bay being assigned.
NSAME	3	(2)	Bay number, the assignments of which are to be repeated at bay numbers **NB1** through **NB2**.
SD1	4	(3)	Identification of the first story level at which the beam is being assigned.
SD2	5	(3)	Identification of the last story level at which the beam is being assigned.
ID	6	(4)	Beam section property identification number for beams being assigned.
IPMAJ	7	(5)	Control for major direction pin conditions: = 0 Continuity at both ends. = 1 Pin at End I of beam. = 2 Pin at End J of beam. = 3 Pin at both ends of beam.
IPMIN	8	(5)	Control for minor direction pin conditions: = 0 Continuity at both ends. = 1 Pin at End I of beam. = 2 Pin at End J of beam. = 3 Pin at both ends of beam.
IPI	9	(6)	Code from 0 to 100 to specify partial fixity at End I.
IPJ	10	(6)	Code from 0 to 100 to specify partial fixity at End J.

Notes

1. This entry is a beam bay number. This number must be positive and not greater than **NB**.

2. If **NSAME** is non zero, it is a bay number, the beam properties and pin end condition of which the user has already assigned in the previous data lines of this data section. The non zero entry for **NSAME** puts the program into a duplication mode. In this mode the beam properties and the pin end conditions of the beams at all levels at bays **NB1** to **NB2** are set identical to those of bay **NSAME** as it stands assigned at the time of this entry.

   Subsequent reassignments of member properties of bay **NSAME** in later data lines will not result in automatic corresponding reassignments of the member properties of bays **NB1** to **NB2**, or vice versa.

   In this duplication mode (i.e. when **NSAME** is non zero) the entries for **SD1**, **SD2**, **ID**, **IP**, **IPI** and **IPJ** on this data line are meaningless and must not be entered. These entries are only used if **NSAME** is 0.

3. This entry is the alphanumeric story identifier associated with the level of the beams being defined. The story level identifiers have been previously defined in Section 3.

4. This entry refers to the beam section property table defined in Section 5.ii. above. This entry may never be negative and must not be greater than **NBP**. Missing elements may be reassigned as having a property set identification number of zero.

5. The pins created at the ends of the beams set the major and minor bending moments at the corresponding ends to 0. The torsional moments are not affected.

6. This entry is an integer between 0 and 100 to specify partial fixity at the particular end of the beam. A value of zero represents full fixity. A value of 100 represents an ideal pin condition for both major and minor bending. A pin specification at a particular end through the parameters **IPMAJ** and **IPMIN** overwrites the corresponding **IPI** and **IPJ** values.

6.xi. Floor Assignments

If **NF** is 0, there are no floor bays defined in this frame. Therefore skip this data section completely (including the blank termination line described below). Otherwise, provide as many data lines as needed to define the floor element property assignment in the floor bays of this frame. If the property of any floor element in any floor bay at any level is not defined, it will be assumed to be 0. Therefore, zero (or dummy) floor elements need not be defined. Property assignments for any floor element may be repeated. The last values read or generated will be used. **End this data section with a blank line**.

Format

Prepare the data in the following form:

NF1 NF2 NSAME SD1 SD2 ID

Description

Variable	Field	Note	Entry
NF1	1	(1)	Bay number of first floor bay being assigned.
NF2	2	(1)	Bay number of last floor bay being assigned.
NSAME	3	(2)	Floor bay number, the assignments of which are to be repeated at bay numbers **NF1** through **NF2**.
SD1	4	(3)	Identification of the first story level at which the floor element is being assigned.
SD2	5	(3)	Identification of the last story level at which the floor element is being assigned.
ID	6	(4)	Floor section property identification number for floor element being assigned.

Notes

1. This entry is a floor bay number. This number must be positive and not greater than **NF**.

2. If **NSAME** is non zero, it is a floor bay number, the floor properties of which the user has already assigned in the previous data lines of this data section. The non zero entry for **NSAME** puts the program into a duplication mode. In this mode the floor properties of the floor elements at all levels at bays **NF1** to **NF2** are set identical to those of bay **NSAME** as it stands assigned at the time of this entry.

   Subsequent reassignments of properties of bay **NSAME** in later data lines will not result in automatic corresponding reassignments of the properties of bays **NF1** to **NF2**, or vice versa.

   In this duplication mode (i.e. when **NSAME** is non zero) the entries for **SD1**, **SD2**, and **ID** on this data line are meaningless and must not be entered. These entries are only used if **NSAME** is 0.

3. This entry is the alphanumeric story identifier associated with the level of the floor elements being defined. The story level identifiers have been previously defined in Section 3.

4. This entry refers to the floor section property table defined in Section 5.iii. above. This entry may never be negative and must not be greater than **NFP**. Missing elements may be reassigned as having a property set identification number of zero.

6.xii. Brace Location and Assignments

If **MBR** is 0, there are no diagonal braces in this frame. Therefore skip this data section completely (including the blank termination line described below). Otherwise, provide (or generate) data for a maximum of **MBR** diagonal braces in this frame. If two or more braces are defined in identical location, the last one specified in the data will overwrite earlier definitions. Similarly, if braces are defined with the same brace identification number more than once only the the last brace definition with the particular brace identification is used. **End this data section with a blank line.**

Format

Prepare the data in the following form:

NBR SD1 SD2 IL IU ID IPMAJ IPMIN ISLOPE

Description

Variable	Field	Note	Entry
NBR	1	(1)	Brace identification number.
SD1	2	(2)	Identification of the first story level at which the brace is being defined.
SD2	3	(2)	Identification of the last story level at which the brace is being defined.
IL	4	(3)	Column line number at bottom of this brace.
IU	5	(3)	Column line number at top of this brace.
ID	6	(4)	Brace section property identification number defining properties of this brace.
IPMAJ	7	(5)	Control for major direction pin conditions: = 0 Continuity at both ends. = 1 Pin at lower end. = 2 Pin at upper end. = 3 Pin at both ends.
IPMIN	8	(5)	Control for minor direction pin conditions: = 0 Continuity at both ends. = 1 Pin at lower end. = 2 Pin at upper end. = 3 Pin at both ends.
ISLOPE	9	(6)	Code for multistory braces: = 1 Brace goes down one story level > 1 Brace goes down **ISLOPE** story levels.

Notes

1. This entry is any unique number between 1 and 32767 that identifies this brace.

2. This entry is the alphanumeric story identifier. The story level identifiers have been previously defined in Section 3. The story level associated with a brace element is the story level at the top of the brace.

   When **SD1** is different from **SD2** the brace elements are generated vertically. The brace identification numbers of the generated elements are obtained by repetitively incrementing the brace identification number of the current brace by 1. The braces are generated every **ISLOPE** level below.

3. The **IL** and **IU** entries must be positive numbers not greater than **NC**.

4. This entry refers to the brace section property table defined in Section 5.iv. above. This entry may never be negative and must not be greater than **NDP**. Braces having property set identification numbers of 0 are ignored by the program.

5. The pins created at the ends of the brace set the major and minor bending moments at the corresponding ends to 0. Torsional moments are not affected.

6. It is possible to define multistory braces. If **ISLOPE** is defined as larger than 1, the brace is assumed to go down **ISLOPE** story levels.

   If **ISLOPE** is left as 0 it defaults to a value of 1.

6.xiii. Panel Location and Assignments

If **MP** is 0, there are no panel elements in this frame. Therefore skip this data section completely (including the blank termination line described below). Otherwise, enter (or generate) data for a maximum of **MPAN** panels in this frame. If two or more panels are defined in identical location, the last one specified in the data will overwrite earlier definitions. **End this data section with a blank line.**

Format

Provide the data in the following form:

NW SD1 SD2 IC JC ID

Description

Variable	Field	Note	Entry
NW	1	(1)	Wall identification number for this panel.
SD1	2	(2)	Identification of the first story level associated with the panel being defined.
SD2	3	(2)	Identification of the last story level associated with the panel being defined.
IC	4	(3)	Column line number at End I of this panel.
JC	5		Column line number at End J of this panel.
ID	6	(4)	Panel section property identification number defining properties of this panel.

Notes

1. This entry is a wall identification number between 1 and 32767.

   All the panel elements **at a particular level** having the same wall identification number form an assemblage that defines a "wall" at that level. This identification is important only to control the output. The moment and force responses from the different panels making up the "wall" are combined statically to produce the "wall" output. The moments and forces reported by the program will be for the top and bottom of each "wall" at the center of gravity of the wall, in the wall local coordinate system. The local major direction of the wall is defined by the I-J direction of that panel element of the "wall" which is first defined (or generated) in the panel data input sequence.

   It is important to recognize that a "wall" with a particular ID at a particular level is created by an assemblage of only those panel elements that exist at the particular level and have the particular wall ID. Therefore, panels that have identical wall ID numbers but exist at different levels, will be associated with different "wall" assemblages that correspond to their respective levels.

2. This entry is the alphanumeric story identifier. These identifiers have been previously defined in Section 3. The story level associated with a panel element is the story level at the top of the panel.

3. The **IC** and **JC** entries must be positive numbers not greater than **NC**.

4. This entry refers to the panel section property table defined in Section 5.v. above. This entry may never be negative and must not be greater than **NPP**. Panels having a property identification number of 0 are ignored by the program.

   All panels corresponding to a particular wall need not have the same panel section property identification (or thickness).

6.xiv. Link Location and Assignments

If **MLNK** is 0, there are no links in this frame. Therefore skip this section of data section completely (including the blank termination line described below). Otherwise, provide (or generate) data for a maximum of **MLNK** links in this frame. Two or more link elements can be defined in identical locations. However, if links are defined with the same link identification number more than once, only the last link definition with the particular link identification is used. **End this data section with a blank line.**

Format

Prepare the data in the following form:

NL SD1 SD2 IL IU ID IPMAJ IPMIN ISLOPE

Description

Variable	Field	Note	Entry
NL	1	(1)	Link identification number.
SD1	2	(2)	Identification of the first story level at which the link is being defined.
SD2	3	(2)	Identification of the last story level at which the link is being defined.
IL	4	(3)	Column line number at the bottom end of this link.
IU	5	(3)	Column line number at the top end of this link.
ID	6	(4)	Spring section property identification number defining properties of this link.
IPMAJ	7	(5)	Control for major direction pin conditions: = 0 Continuity at both ends. = 1 Pin at lower end. = 2 Pin at upper end. = 3 Pin at both ends.
IPMIN	8	(5)	Control for minor direction pin conditions: = 0 Continuity at both ends. = 1 Pin at lower end. = 2 Pin at upper end. = 3 Pin at both ends.
ISLOPE	9	(6)	Code for multistory links: = 0 link is horizontal > 0 link goes down **ISLOPE** story levels.

Notes

1. This entry is any unique number between 1 and 32767 that identifies this link.

2. This entry is the alphanumeric story identifier. The story level identifiers have been previously defined in Section 3. The story level associated with a link element is the story level at the top of the link.

   When **SD1** is different from **SD2** the link elements are generated vertically. The link identification numbers of the generated elements are obtained by repetitively incrementing the link identification number of the current link by 1. If **ISLOPE** is greater than 0 the links are generated every **ISLOPE** level below.

3. The **IL** and **IU** entries must be positive numbers not greater than **NC**.

4. This entry refers to the spring section property table defined in Section 5.vi. above. This entry may never be negative and must not be greater than **NSP**. Links having property set identification numbers of 0 are ignored by the program.

5. The pins created at the ends of the link set the major and minor bending moments at the corresponding ends to 0. Torsional moments are not affected.

6. It is possible to define multistory links. If **ISLOPE** is defined as larger than 0, the link is assumed to go down **ISLOPE** story levels.

6.xv. Joint Load Assignments

If **NJLP** is 0, there are no loads to be applied to the joints of this frame. Therefore skip this data section completely (including the blank termination line described below). Otherwise, provide as many data lines as needed to assign joint load patterns to all the loaded joints of this frame. If the loading for any joint at any level is not defined, it is assumed to have no load. Load pattern assignments may be repeated. The last values read or generated will be used. **End this data section with a blank line.**

Format

Prepare the data in the following form:

NC1 NC2 NSAME SD1 SD2 LI LII LIII LA LB LC

Description

Variable	Field	Note	Entry
NC1	1	(1)	Column line number of first joint being assigned.
NC2	2	(1)	Column line number of last joint being assigned.
NSAME	3	(2)	Column line number, the joint loadings of which are to be repeated at column lines **NC1** to **NC2**.
SD1	4	(3)	Identification of the first story level at which the joint loading is being assigned.
SD2	5	(3)	Identification of the last story level at which the joint loading is being assigned.
LI	6	(4)	Joint load pattern identification number defining loads at this joint for load condition I.
LII	7		Joint load pattern identification number defining loads at this joint for load condition II.
LIII	8		Joint load pattern identification number defining loads at this joint for load condition III.
LA	9		Joint load pattern identification number defining loads at this joint for load condition A.
LB	10		Joint load pattern identification number defining loads at this joint for load condition B.
LC	11		Joint load pattern identification number defining loads at this joint for load condition C.

Notes

1. This entry is a column line number. This number must be positive and not greater than **NC**.

2. If **NSAME** is non zero, it is the column line number, the joint loading of which the user has already assigned in preceding data lines of this data section. The non zero entry for **NSAME** puts the program into a duplication mode. In this mode the loading (for all six load conditions I, II, III, A, B and C) of the joints on column lines **NC1** to **NC2** at all levels is set identical to that of joints on column line **NSAME**, as they stands assigned at the time of this entry.

   Subsequent reassignments of the loading of joints on column line **NSAME** in later data lines will not result in automatic corresponding reassignments of the loading on joints of column lines **NC1** or **NC2**, and vice versa.

   In the duplication mode (i.e. when **NSAME** is non zero) the entries for **SD1, SD2, LI, LII, LIII, LA, LB** and **LC** on this data line are meaningless and must not be entered. These entries are only used if **NSAME** is 0.

3. This entry is the alphanumeric story identifier. The story identifiers have been previously defined in Section 3.

4. This entry refers to the joint load pattern table defined in Section 6.v. above. This entry may never be negative and must not be greater than **NJLD**. Load pattern identification numbers of 0 will null previous definitions.

6.xvi. Beam Load Assignments

If **NB** is 0, or **NBLP** is 0, there are no beam bays or beam load patterns defined for this frame. Therefore skip this data section completely (including the blank termination line described below). Otherwise, provide as many data lines as needed to assign beam load patterns to all the loaded beams of this frame. If the loading for any beam in any bay at any level is not defined, it is assumed to have no load. Load pattern assignments may be repeated. The last values read or generated will be used. **End this data section with a blank line.**

Format

Prepare the data in the following form:

NB1 NB2 NSAME SD1 SD2 LI LII LIII

Description

Variable	Field	Note	Entry
NB1	1	(1)	Bay number of the first beam for which loading is being assigned.
NB2	2	(1)	Bay number of the last beam for which loading is being assigned.
NSAME	3	(2)	Bay number, the loading of which is to be repeated at bay numbers **NB1** to **NB2**.
SD1	4	(3)	Identification of the first story level at which the beam loading is being assigned.
SD2	5	(3)	Identification of the last story level at which the beam loading is being assigned.
LI	6	(4)	Beam load pattern identification number defining loads on this beam for load condition I.
LII	7		Beam load pattern identification number defining loads on this beam for load condition II.
LIII	8		Beam load pattern identification number defining loads on this beam for load condition III.

Notes

1. This entry is a beam bay number. This number must be positive and not greater than **NB**.

2. If **NSAME** is non zero, it is a bay number, the loading of which the user has already assigned in preceding data entries of this section. The non zero entry for **NSAME** puts the program into a duplication mode. In this mode the loading (for all conditions I, II and III) of the beams at all levels at bays **NB1** to **NB2** is set identical to that of bay **NSAME** as it stands assigned at the time of this entry.

   Subsequent reassignments of the loading of bay **NSAME** in later data lines will not result in automatic corresponding reassignments of the loading on bays **NB1** to **NB2**, and vice versa.

   In the duplication mode (i.e. when **NSAME** is non zero), the entries for **SD1, SD2, LI, LII and LIII** on this data line are meaningless and must not be entered. These entries are only used if **NSAME** is 0.

3. This entry is the alphanumeric story identifier associated with the level of the beams, the loadings of which are being assigned. The story level identifiers have been previously defined in Section 3.

4. This entry refers to the beam load pattern table defined in Section 6.vi. above. This entry may never be negative and must not be greater than **NBLP**. Load pattern identification numbers of 0 will null previous definitions.

   The beam self weight loading may be added as a uniform load on the beam span in addition to the beam load pattern. This, however, may be used for only one of the vertical load conditions, depending upon the value of **ISLF**.

   Loads applied to bays having zero (0) beam property identifications are ignored.

   The vertical loads applied to floor elements in Section 6.xvii. below are converted to beam loads and are added to the beam loads defined in this section.

6.xvii. Floor Load Assignments

If **NF** is 0, or **NFLP** is 0, there are no floor bays or floor load patterns defined for this frame. Therefore skip this data section completely (including the blank termination line described below). Otherwise, provide as many data lines as needed to assign floor loading patterns to all the loaded floor elements of this frame. If the loading for any floor element in any bay at any level is not defined, it will be assumed to have no load. Loading pattern assignments may be repeated. The last values read or generated will be used. **End this data section with a blank line**.

Format

Prepare the data in the following form:

NF1 NF2 NSAME SD1 SD2 LI LII LIII LA LB LC

Description

Variable	Field	Note	Entry
NF1	1	(1)	Floor bay number of the first floor bay for which loading is being assigned.
NF2	2	(1)	Floor bay number of the last floor bay for which loading is being assigned.
NSAME	3	(2)	Floor bay number, the loading of which is to be repeated at bay numbers **NF1** to **NF2**.
SD1	4	(3)	Identification of the first story level at which the floor loading is being assigned.
SD2	5	(3)	Identification of the last story level at which the floor loading is being assigned.
LI	6	(4)	Floor load pattern identification number defining loads on this floor element for load condition I.
LII	7		Floor load pattern identification number defining loads on this floor element for load condition II.
LIII	8		Floor load pattern identification number defining loads on this floor element for load condition III.
LA	9		Floor load pattern identification number defining loads on this floor element for load condition A.
LB	10		Floor load pattern identification number defining loads on this floor element for load condition B.
LC	11		Floor load pattern identification number defining loads on this floor element for load condition C.

Notes

1. This entry is a floor bay number. This number must be positive and not greater than **NF**.

2. If **NSAME** is non zero, it is a floor bay number, the loading of which the user has already assigned in preceding data entries of this section. The non zero entry for **NSAME** puts the program into a duplication mode. In this mode the loading (for all conditions I, II, III, A, B and C) of the floor elements at all levels at bays **NF1** to **NF2** is set identical to that of bay **NSAME** as it stands assigned at the time of this entry.

   Subsequent reassignments of the loading of bay **NSAME** in later data lines will not result in automatic corresponding reassignments of the loading on bays **NF1** to **NF2**, and vice versa.

   In the duplication mode (i.e. when **NSAME** is non zero), the entries for **SD1, SD2, LI, LII**, **LIII**, **LA**, **LB** and **LC** on this data line are meaningless and must not be entered. These entries are only used if **NSAME** is 0.

3. This entry is the alphanumeric story identifier associated with the level of the floor elements, the loadings on which are being assigned. The story level identifiers have been previously defined in Section 3.

4. This entry refers to the floor load pattern table defined in Section 6.vii. above. This entry may never be negative and must not be greater than **NFLP**. Load pattern identification numbers of zero (0) will null previous definitions.

   The floor element self weight loading may be added as a uniform load on the floor element in addition to the floor loading pattern. This, however, may be used for only one of the vertical load conditions, depending upon the value of **ISLF**.

   Loads applied to bays having zero (0) floor property identifications are ignored.

   For load conditions I, II and III the floor element vertical loads are converted to beam loads as per the algorithms shown on Figure V-18 and are in addition to any beam loads specified. For load conditions A, B or C they are applied to the joints.

   The floor element lateral loads are converted to joint loads on a tributary area basis and are in addition to any lateral loads specified in Section 6.xv. above.

7. Frame Location Data

Provide one line of data to place each of the **NTF** (see Figure V-21) frames that exist in the structure.

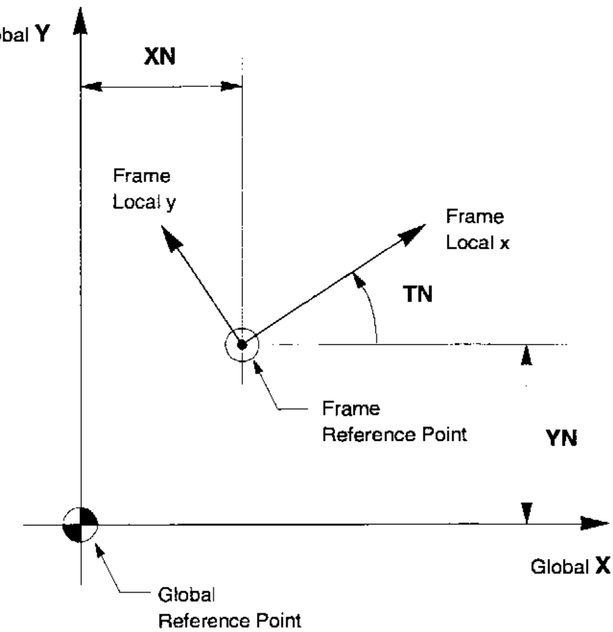
Format

A total of **NTF** data lines must be prepared in the following form:

IF XN YN TN FHED

Description

Variable	Field	Note	Entry
IF	1	(1)	Frame type identification number.
XN	2		Global X-ordinate of the frame local origin.
YN	3		Global Y-ordinate of the frame local origin.
TN	4		Angle (in degrees) between the frame local x-axis and the global X-axis, measured from the global X-axis, counterclockwise positive.
FHED	5		Alphanumeric information to identify the output associated with the frame, up to 32 characters.



Frame Location and Orientation
Figure V-21

Notes

1. This entry refers to the frame type identification number (see Frame Control Data, Section 6.i.b.) of the frame being placed via this frame location data line. Frame type identification numbers may be repeated, but the location data lines must be input such that the frame type identification numbers are in an ascending numerical sequence starting with 1, so that the first of the **NTF** frames that is located must have a frame type identification number of 1 and the last of the **NTF** frames that is located must have a frame type identification number of **NDF**. All of the **NDF** frame types must be referenced by the **NTF** frames.

8. Lateral Static Load Data

This data section is for defining the lateral loads that are to be applied to the building diaphragms for lateral load conditions A, B and C.

If **ILAT** is 0, no lateral loads are applied to the diaphragms. Therefore skip this data section completely. If **ILAT** is 1, i.e. user defined diaphragm lateral loads are to be defined, prepare data as per Section 8.i. below. If **ILAT** is between 11 and 14 or 31, i.e. data for automatic calculation of seismic load is to be defined, prepare data as per Section 8.ii. below. If **ILAT** is between 21 and 24, i.e. data for automatic calculation of wind load is to be defined, prepare data as per Section 8.iii. below.

8.i. User-Defined Lateral Static Loads

Skip this section of data if **ILAT** is not 1. Otherwise, provide as many data lines as needed to define lateral loads on the diaphragms of the building. The diaphragm lateral loads can only be applied to the lateral load conditions A, B and C. The order of input is immaterial. Story identification, diaphragm number and load conditions can all be repeated. If repeated, the loads are additive. **End this data section with a blank line.**

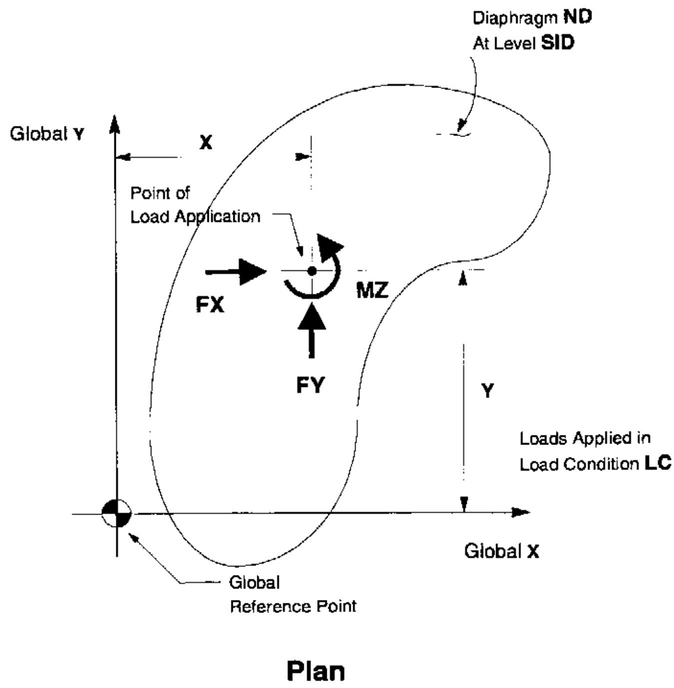
Format

Prepare the data in the following form:

SID ND LC FX FY X Y MZ

Description

Variable	Field	Note	Entry
SID	1	(1)	Identification label for story where load is applied.
ND	2		Identification number of diaphragm where load is applied.
LC	3		Load condition A, B or C.
FX	4		X-direction load.
FY	5		Y-direction load.
X	6		X-coordinate of the point of load application.
Y	7		Y-coordinate of the point of load application.
MZ	8		Moment load about the vertical (z-axis)



User-Defined Diaphragm Lateral Loading
Figure V-22

Notes

1. The lateral loads can be applied at any or all diaphragms at any or all story levels in the three static lateral load conditions A, B or C. All axis references are global. See Figure V-22 for positive directions of forces.

   These lateral loads are in addition to any lateral loads applied at the individual joints.

8.ii. Automatic Lateral Static Seismic Loads

Skip this section of data if **ILAT** is not 11, 12, 13, 14 or 31. Otherwise, prepare the following data as defined in Sections a, b and c below. This data will activate structural lateral load conditions A, B and C. Load condition A will have lateral loads in the global X-direction, load condition B will have lateral loads in the global Y-direction, and load condition C will have torsional loads due to specified additional eccentricities. The building story heights and story mass defined in the story data (Section 3 above) or as calculated by the program (Section 2 above) converted to weights using the gravitational multiplier GRAV defined in Section 1.d above are used to generate these loads.

The lateral loads are applied to each diaphragm with a non-zero mass at each story level. This option should be used with caution if multiple diaphragms per story are present, especially if they are connected with very flexible diaphragms or are not connected to each other. Dynamic analysis is recommended for such situations.

Format

a. Seismic Coefficients

This data is dependent on the Code to be used. Prepare one line of data in the following form to define the site-dependent coefficients:

For the UBC '94 Code without seismic isolation (ILAT = 11)

Z I S

or for the BOCA '93 Code (ILAT = 12)

AV AA S

or for the NBCC '90 Code (ILAT = 13)

VR ZR F I

or for the ATC 3-06 Code (ILAT = 14)

AV AA S

For the UBC '94 Code with seismic isolation (ILAT = 31)

Z N SI

b. Lateral Load Distribution Data

Provide two lines of data, one corresponding to each of the two global directions, to define the direction dependent structural period and structure type data and the extent of the lateral load distributions over the height of the building.

For all code types except UBC '94 Code with seismic isolation prepare the data in the following form:

TX TAX RX STOPX SBOTX

TY TAY RY STOPY SBOTY

For UBC '94 Code with seismic isolation prepare the data in the following form:

TIX BX RIX STOPX SBOTX

TIY BY RIY STOPY SBOTY

c. Additional Story Eccentricities

The generated story loads on any particular diaphragm are applied at the center of mass of that diaphragm in load conditions A and B. If additional eccentricities are required they are specified in this data section. These are then used to apply torsional moments in load condition C.

Prepare one line of data to define the eccentricities corresponding to each story of the building.

The data is to be prepared starting from the top story of the building and progressing in sequence toward the bottom of the building. The data amounts to a total of **NST** data lines in the following form:

EYA EXB

Description

Variable	Field	Note	Entry

a. Seismic Coefficients

For the UBC '94 Code without seismic isolation(ILAT =11)

Variable	Field	Note	Entry
Z	1	(1)	UBC seismic zone factor, Z.
I	2		UBC importance factor, I.
S	3		UBC soil profile based site coefficient, S.

For the BOCA '93 Code (ILAT = 12)

Variable	Field	Note	Entry
AV	1	(2)	BOCA effective peak velocity related acceleration coefficient, A_v.
AA	2		BOCA effective peak acceleration coefficient, A_a.
S	3		BOCA soil profile based site coefficient, S.

For the NBCC '90 Code (ILAT = 13)

Variable	Field	Note	Entry
VR	1	(3)	NBCC zonal velocity ratio, v.
ZR	2		NBCC ratio Z_a/Z_v.
F	3		NBCC foundation factor, F.
I	4		NBCC seismic importance factor, I.

For the ATC 3-06 Code (ILAT = 14)

Variable	Field	Note	Entry
AV	1	(4)	ATC effective peak velocity related acceleration coefficient, A_v.
AA	2		ATC effective peak acceleration coefficient, A_a.
S	3		ATC soil profile characteristic coefficient, S.

Variable	Field	Note	Entry
For the UBC '94 Code with seismic isolation (ILAT =31)			
Z	1	(5)	UBC seismic zone factor, Z.
N	2		UBC fault proximity coefficient, N.
SI	3		UBC site soil profile coefficient for isolated structures, S_I.

b. Lateral Load Distribution Data

For all code types except UBC '94 Code with seismic isolation

X- Direction Loading

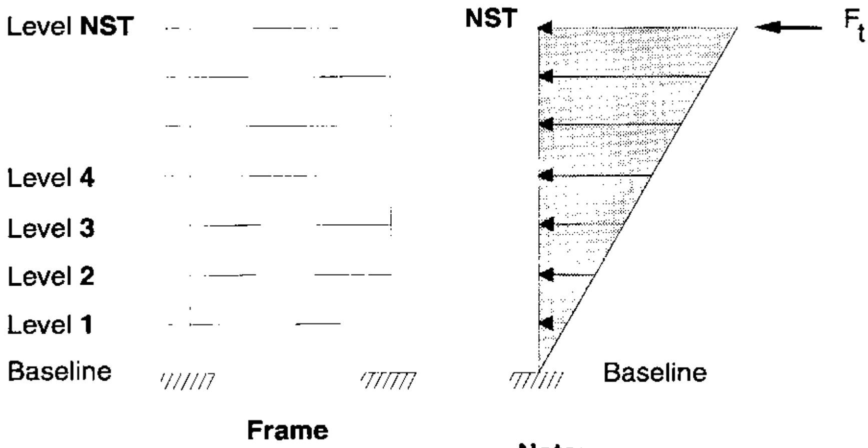
Variable	Field	Note	Entry
TX	1	(6)	Time period of the structural mode predominant in the X-direction.
TAX	2	(1,2,3,4)	UBC Method A fundamental period or BOCA approximate fundamental period or NBCC maximum allowed period or ATC approximate fundamental period associated with the X-direction.
RX	3	(1,2,3,4)	UBC structural system coefficient, R_W or BOCA response modification factor, R or NBCC force modification factor, R or ATC response modification coefficient, R associated with the X-direction.
STOPX	3	(7)	Story identification for the level defining the top of the lateral load distribution for X-direction loads.
SBOTX	4		Story identification for the level defining the bottom of the lateral load distribution for X-direction loads.

Variable	Field	Note	Entry
Y- Direction Loading			
TY	1	(6)	Time period of the structural mode predominant in the Y-direction.
TAY	2	(1,2,3,4)	UBC Method A fundamental period or BOCA approximate fundamental period or NBCC maximum allowed period or ATC approximate fundamental period associated with the Y-direction.
RY	3	(1,2,3,4)	UBC structural system coefficient, R_W or BOCA response modification factor, R or NBCC force modification factor, R or ATC response modification coefficient, R associated with the Y-direction.
STOPY	3	(7)	Story identification for the level defining the top of the lateral load distribution for Y-direction loads.
SBOTY	4		Story identification for the level defining the bottom of the lateral load distribution for Y-direction loads.
For the UBC '94 Code with seismic isolation			
X- Direction Loading			
TIX	1	(8)	Fundamental time period of the isolated structure in the X-direction.
BX	2	(5)	UBC damping related coefficient, B associated with the X-direction.
RIX	3	(1,2,3,4)	UBC structural system coefficient for isolated structures, R_{WI} associated with the X-direction.

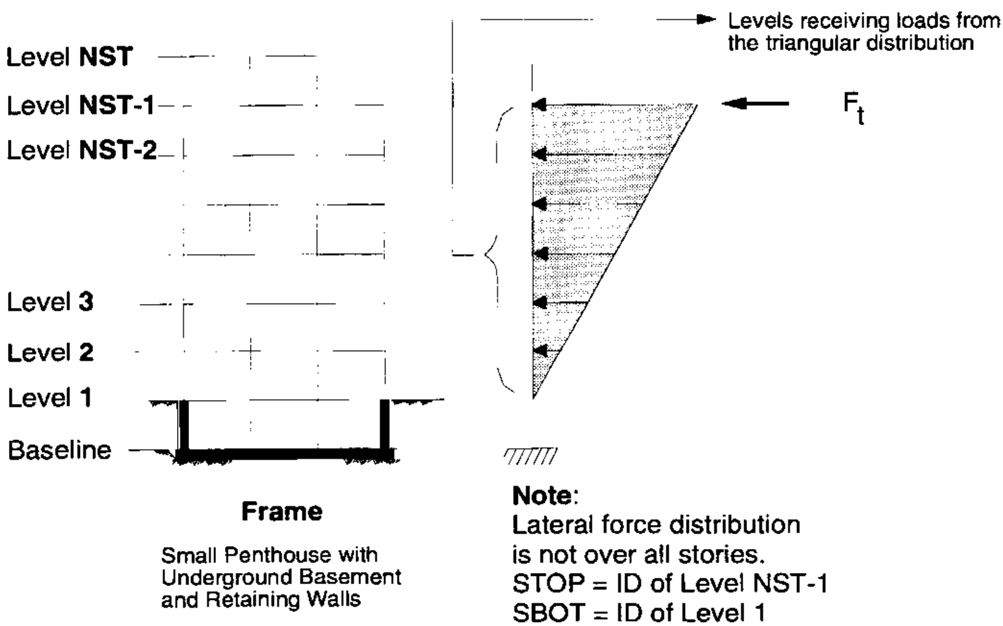
Variable	Field	Note	Entry
STOPX	3	(7)	Story identification for the level defining the top of the lateral load distribution for X-direction loads.
SBOTX	4		Story identification for the level defining the bottom of the lateral load distribution for X-direction loads.
Y- Direction Loading			
TIY	1	(8)	Fundamental time period of the isolated structure in the Y-direction.
BY	2	(5)	UBC damping related coefficient, B associated with the Y-direction.
RIY	3	(1,2,3,4)	UBC structural system coefficient for isolated structures, R_{WI} associated with the Y-direction.
STOPY	3	(7)	Story identification for the level defining the top of the lateral load distribution for Y-direction loads.
SBOTY	4		Story identification for the level defining the bottom of the lateral load distribution for Y-direction loads.

c. Additional Story Eccentricities

Variable	Field	Note	Entry
EYA	1	(9)	Additional eccentricity for X-direction loads.
EXB	2		Additional eccentricity for Y-direction loads.



Note:
Lateral force distribution
is over all stories.
STOP = ID of Level NST
SBOT = Blank (Baseline)



Note:
Lateral force distribution
is not over all stories.
STOP = ID of Level NST-1
SBOT = ID of Level 1

Seismic Lateral Load Distribution for UBC and NBCC Codes
Figure V-23

Notes

1. The seismic load calculation method and the definitions of the various factors are based on Section 1628, Chapter 16 of the Uniform Building Code [13].

2. The seismic load calculation method and the definitions of the various factors are based on Section 1612 of the BOCA National Building Code [15].

3. The seismic load calculation method and the definitions of the various factors are based on Section 4.1.9 of the National Building Code of Canada [17].

4. The seismic load calculation method and the definitions of the various factors are based on Chapter 4 of the ATC 3-06 [14].

5. The seismic load calculation method and the definitions of the various factors are based on Section 1654, Appendix Chapter 16 of the Uniform Building Code [13].

6. **TX** must be provided if the time periods of the structure are not being calculated by the program (i.e. **IDYN** is 0). If **TX** is set to 0 and **IDYN** is not 0, the program will set **TX** equal to the period of the calculated mode having the largest participation factor in the X-direction.

   Similarly, **TY** corresponds to the Y-direction.

   Positive non zero user input values for **TX** and **TY** will always supersede any values generated by the program.

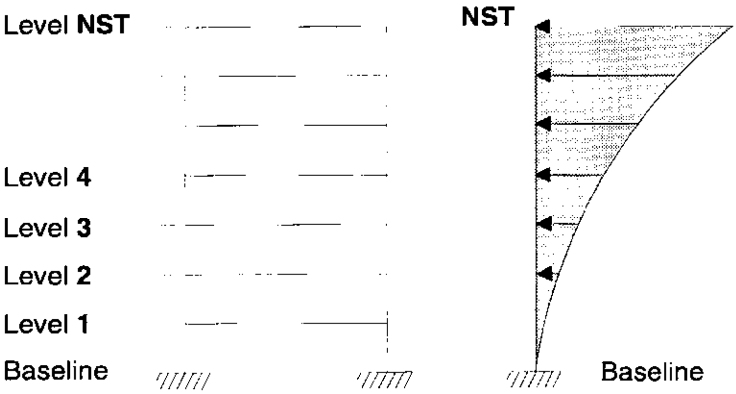
7. It is possible for a structural model to have levels for penthouses, basements or dummy stories for special modeling. The user may not want to include such levels in the overall structural lateral force distribution as defined by the Codes. See Figures V-23 and V-24.

   If there are any levels above **STOPX**, they do not receive any loading from the vertical distribution of lateral loads. The distribution will also not generate any loads for any level below and including level **SBOTX**.

   If **SBOTX** is left blank, **SBOTX** is assumed to be the structural baseline.

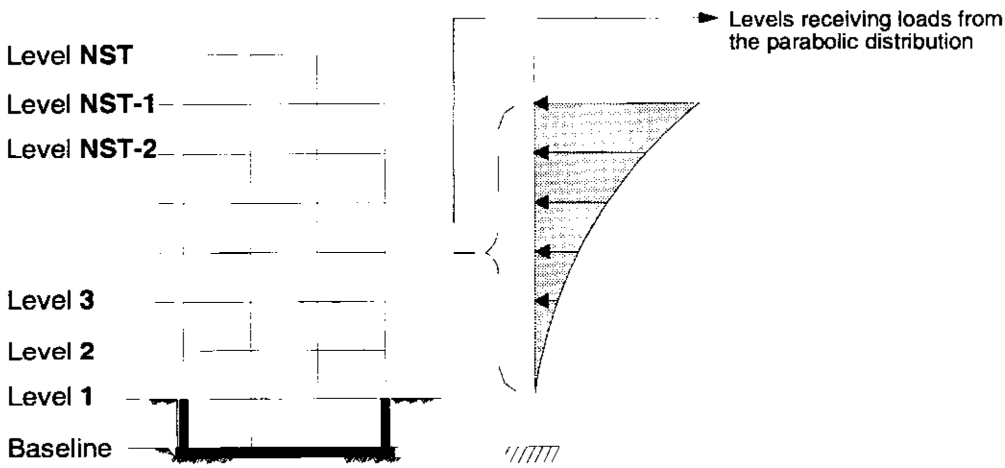
   If **STOPX** and **SBOTX** are both left blank, **STOPX** is assumed to be the top level of the structure and **SBOTX** is assumed to be the structural baseline.

   All the above is true for **STOPY** and **SBOTY** as it corresponds to the Y-direction.



Frame

Note:
Lateral force distribution
is over all stories.
STOP = ID of Level NST
SBOT = Blank (Baseline)

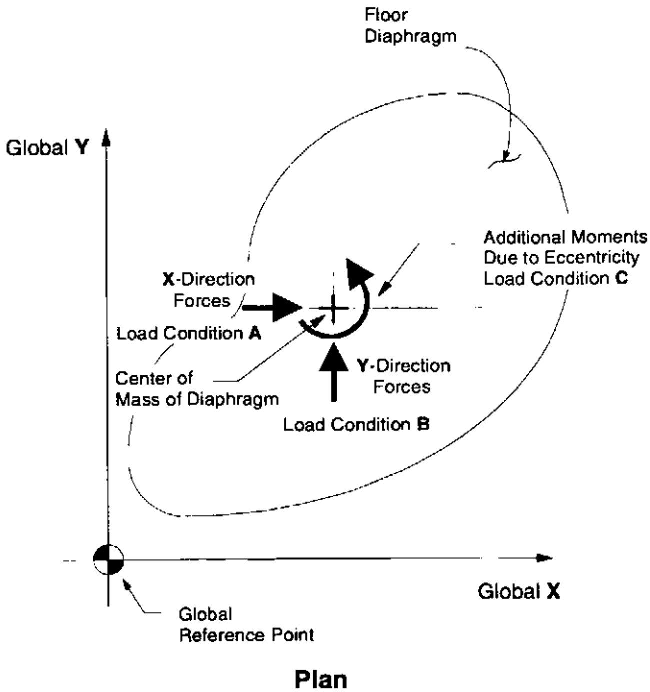


Frame

Small Penthouse with
Underground Basement
and Retaining Walls

Note:
Lateral force distribution
is not over all stories.
STOP = ID of Level NST-1
SBOT = ID of Level 1

Seismic Lateral Load Distribution for BOCA and ATC Codes
Figure V-24



Automated Seismic Lateral Loading
Figure V-25

Any levels that do not get lateral loads from the automatic distribution can still be assigned lateral loads through joint loading assignments.

8. If the fundamental period of the isolated structure in the X direction, **TIX** is not provided (i.e., set to 0.0) then the program calculates this from the effective stiffness of the isolator elements specified and the mass of the structure above and including level **SBOTX** assuming the system to be a single degree of freedom system. Similarly for **TIY** for the Y direction.

9. The loads are applied at the center of mass of each diaphragm for the applicable story levels. If additional eccentricities are specified, the larger of the two torsions is applied to load condition C. See Figure V-25. This option is useful in accounting for accidental torsions. Loading combinations with positive and negative load condition C can be used to account for positive and negative accidental torsions.

8.iii. Automatic Lateral Static Wind Loads

Skip this section of data if **ILAT** is not 21, 22, 23 or 24. Otherwise, prepare the following data as defined in Sections a, b and c below. This data will activate structural lateral load conditions A and B. Load condition A will have wind loads in the global X-direction and load condition B will have wind loads in the global Y-direction. The building story heights defined in the story data (Section 3 above) are used to generate these loads.

All lateral static wind loads are applied to diaphragm 1 at each story level. This automatic option should not be used if multiple diaphragms per story are present. The user defined lateral loads option or joint loading options should be used instead.

Format

a. Wind Pressure Parameters

This data is dependent on the code to be used. Prepare one line of data in the following form to define the wind pressure parameters:

For the UBC '94 Code (ILAT = 21)

V ET I

or for the BOCA '93 Code (ILAT = 22)

V ET I AR

or for the NBCC '90 Code (ILAT = 23)

Q CP

or for the ASCE 7-93 Code (ILAT = 24)

V ET I AR GBAR

b. Wind Exposure Height

Provide two lines of data, one corresponding to each of the two global directions, to define the extent of the structural height that is exposed to the wind. Prepare the data in the following form:

SBOTX

SBOTY

c. Wind Exposure Widths and Centers of Pressure

This data defines the dimensions of the exposed faces, normal to the global X- and Y-directions, and locates the corresponding centers of pressure to determine the magnitude of the wind loads and their corresponding points of application for each story.

Provide one line of data corresponding to each story of the building. The data is to be prepared starting from the top story of the building and progressing in sequence toward the bottom of the building.

The data amounts to a total of **NST** data lines in the following form:

BYA DYA BXB DXB

Description

Variable	Field	Note	Entry
a. Wind Pressure Parameters			
For the UBC '94 Code (ILAT =21)			
V	1	(1)	UBC basic wind speed (between 70 and 130 mph).
ET	2		UBC exposure type (B, C or D).
I	3		UBC importance factor.
For the BOCA '93 Code (ILAT =22)			
V	1	(2)	BOCA basic wind speed (between 70 and 110 mph).
ET	2		BOCA exposure type (A, B, C or D).
I	3		BOCA importance factor.
AR	4		Aspect Ratio (building dimension in X-direction divided by dimension in Y-direction).
For the NBCC '90 Code (ILAT =23)			
Q	1	(3)	NBCC reference velocity pressure, q in kPa.
CP	2		NBCC averaged external pressure coefficient, C_p.
For the ASCE 7-93 Code (ILAT =24)			
V	1	(4)	ASCE basic wind speed, mph.
ET	2		ASCE exposure type (A, B, C or D).
I	3		ASCE importance factor.
AR	4		Aspect Ratio (building dimension in X-direction divided by dimension in Y-direction).

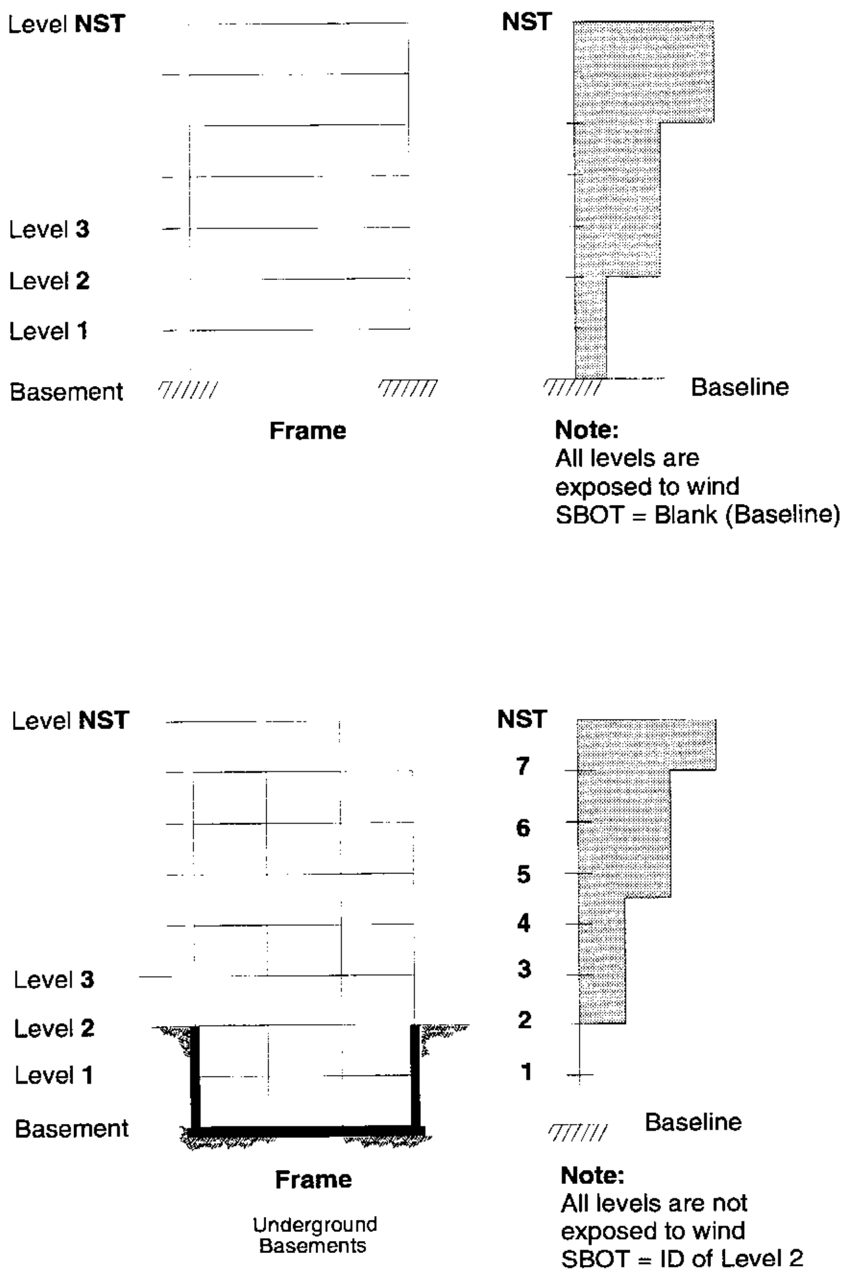
Variable	Field	Note	Entry
GBAR	5		ASCE gust factor for flexible buildings and slender structures.

b. Wind Exposure Height

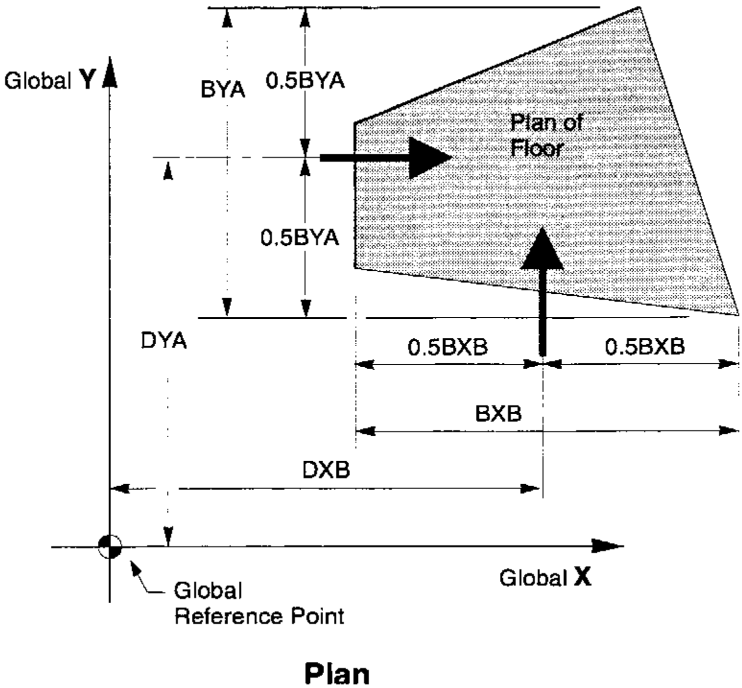
Variable	Field	Note	Entry
SBOTX	1	(5)	Story identification for level corresponding to the ground level for the wind exposure for X-direction wind loads.
SBOTY	1		Story identification for level corresponding to the ground level for the wind exposure for Y-direction wind loads.

c. Wind Exposure Widths and Centers of Pressure

Variable	Field	Note	Entry
BYA	1	(6)	Projected length of the floor level for X-direction loads (projection is on the Y-axis).
DYA	2	(6)	Y-ordinate of the center of the projected length **BYA**.
BXB	4		Projected length of the floor level for Y-direction loads (projection is on the X-axis).
DXB	5		X-ordinate of the center of the projected length **BXB**.



Wind Lateral Load Distribution
Figure V-26



Wind Lateral Loading
Figure V-27

Notes

1. The wind load calculation method and the definitions of the various factors are based on Chapter 16 of the Uniform Building Code [13].

2. The wind load calculation method and the definitions of the various factors are based on Section 1611 of the BOCA National Building Code [15].

3. The wind load calculation method and the definitions of the various factors are based on Section 4.1.8 of the National Building Code of Canada [17].

4. The wind load calculation method and the definitions of the various factors are based on Section 6 of the ASCE Standard 7-93 [16].

5. The part of the structure below the level line associated with the story identification **SBOTX** will receive no wind exposure in the X-direction. See Figure V-26. If **SBOTX** is blank, **SBOTX** is assumed to be the structural baseline.

The same is true for **SBOTY** as it corresponds to the Y-direction.

6. The wind loads are calculated as follows. The wind pressures over the story height (including stepped pressure variations, if any) are established from the code formulas. For the X-direction loads the elevation of a particular story line above the ground is determined using **SBOTX**.

   The wind forces are then calculated as a product of the wind pressure variations and story widths **BYA**. See Figure V-27.

   The loads are then lumped at the story levels on diaphragm number 1 in a statically consistent manner, including the effect of pressure variations within a story height.

   The line of action of the load is in the X-direction through the center of pressure as defined by the entry **DYA**.

   The same is true for **SBOTY**, **BXB** and **DXB** as they correspond to the Y-direction loading.

9. Lateral Dynamic Spectrum Data

This data is for defining the ground acceleration spectrum curves for a dynamic response spectrum analysis.

Skip this section of data if **IDYN** is not 2. Otherwise, prepare the data as defined in Sections a, b and c below.

Format

a. Title Information

Prepare one line of data with title information to identify the response spectrum curve. Up to 50 characters of information may be provided.

b. Control Data

Prepare one line of data of control information in the following form:

ANG ICQC DAMP F1 F2

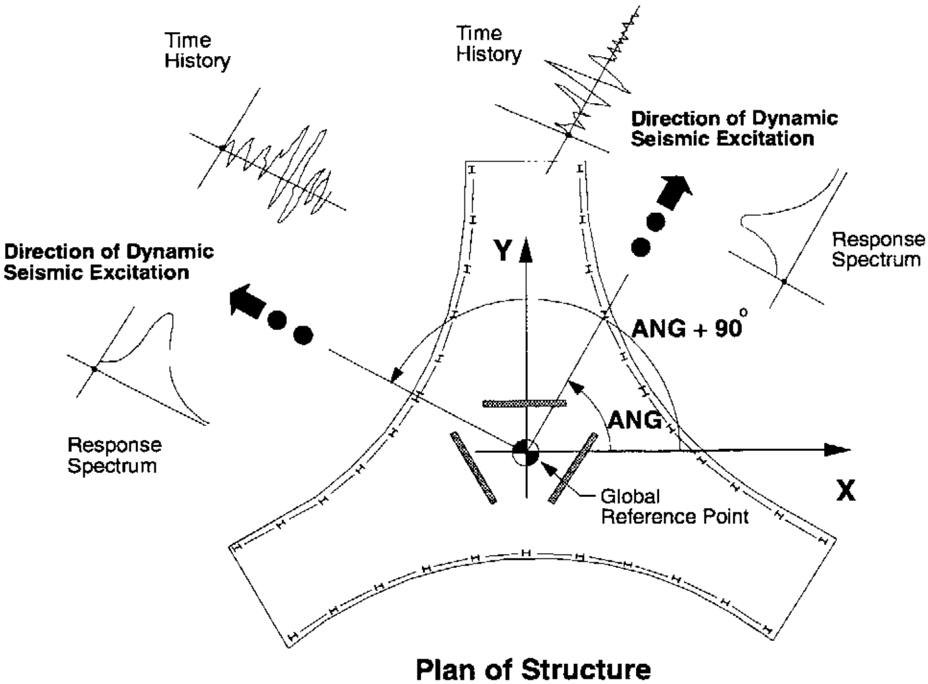
c. Spectrum Data

Prepare two lines of data to specify the response spectra in the following form:

SPECFIL1 SF1 NDAMP1 IPRIN1
SPECFIL2 SF2 NDAMP2 IPRIN2

Description

Variable	Field	Note	Entry
b. Control Data			
ANG	1	(1)	Dircction of earthquake input (degrces).
ICQC	2	(2)	Modal combination technique code: = SRSS SRSS modal combination = CQC CQC modal combination. = GMC GMC modal combination.
DAMP	3	(3)	Damping ratio.
F1	4	(4)	Parameter for GMC.
F2	5	(4)	Parameter for GMC.
c. Spectrum Data			
SPECFIL1 **SPECFIL2**	1	(5)	Filenames containing spectral values.
SF1 **SF2**	2	(6)	Scale factors for spectral values.
NDAMP1 **NDAMP2**	3	(7)	Number of damping values for which spectra file is provided.
IPRN1 **IPRN2**	4		Input responsc spectrum print flag: = 0 Print. = 1 Do not print.



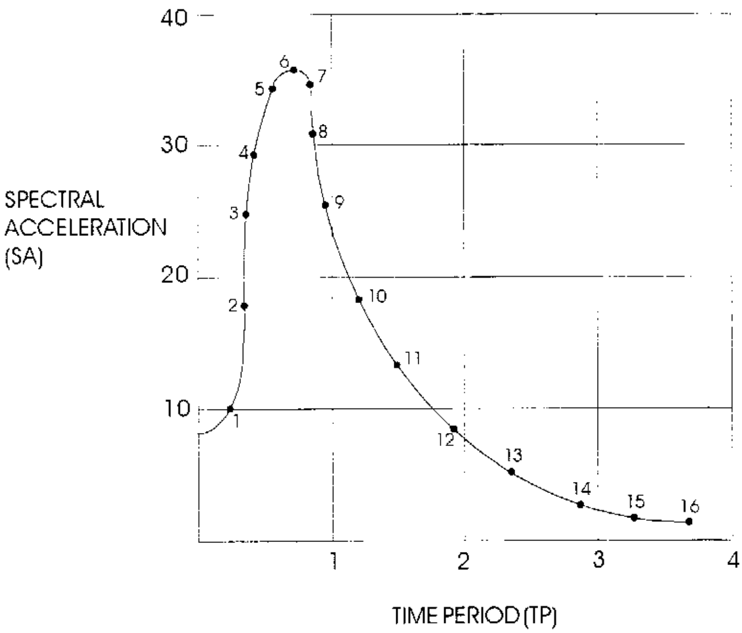
Dynamic Seismic Excitation Direction Convention
Figure V-28

Notes

1. The dynamic excitation can be applied from two different directions in any one run. The directions correspond to the two independent dynamic load conditions D1 and D2. The directions of excitation are all of a lateral nature (in the X-Y plane). No vertical dynamic options are available. **ANG** defines the direction of excitation for D1 measured counter clockwise in degrees from the global x-direction. See Figure V-28. The excitation for D2 is automatically taken at **ANG** + 90°.

2. All modal combinations (for all dynamic load conditions) will be formed by either the square-root-of-the-sum-of-the-squares (SRSS) or the complete-quadratic-combination technique (CQC) or the general modal combination technique (GMC, also known as the Gupta Method. See Reference [10]), depending upon this entry.

3. This damping ratio is additional to any damping ratios calculated by the program for individual modes because of supplemental damping specified. If no supplemen-



Typical Response Spectrum Curve
Figure V-29

tal damping is specified **DAMP** is used as a damping ratio for each mode. The damping is used for two purposes; to select the proper spectra and for the CQC modal combination and GMC modal combination.

4. The general modal combination (GMC or Gupta Method) which accounts for modal correlation between modes with frequencies close to the rigid frequency of an earthquake in addition to modal correlation based on modes with closely spaced frequencies requires these two additional parameters. See Reference [10] for details. ASCE 4 recommends a value of $F1 = SA_{max} / 2\pi SV_{max}$ where SA_{max} and SV_{max} are the maximum ground acceleration and velocity of the earthquake, the program default is 1 Hz. ASCE 4 recommends a value of F2 = Fr where Fr is the rigid frequncy of the earthquake, the program default is 33 Hz.

5. The **SPECFIL1** and **SPECFIL2** are text files containing spectral acceleration values. The **SPECFIL1** contains **NDAMP1** blocks of data one following the other. Each block has a damping ratio on the first line followed by lines with pairs of time period and spectral acceleration values, for this damping ratio and ended by a blank

line. The spectra specified in file **SPECFIL1** is for the seismic excitation to be applied in load condition D1 at an angle **ANG** to the global X-axis. The **SPECFIL2** is similar in format and specifies the seismic excitation to be applied in load condition D2 at an angle **ANG** + 90^{o} to the global X-axis. A typical response spectrum curve is shown in Figure V-29.

If unidirectional motion is to be applied, one of these lines specifying the spectral file should be left blank.

The program interpolates linearly to obtain the acceleration for a given damping and a given time period. If the damping ratio or time period is out of range of values provided in the spectral curves the acceleration associated with the closest damping or time period is used.

The filenames UBC94S1, UBC94S2 and UBC94S3 are reserved names. Their use refers to built in spectral shapes from the UBC 1994 code for soil types S1, S2 and S3, respectively. The spectral shapes are for a single damping ratio of 0.05.

6. These are the multiplying factors to scale the spectrum acceleration values input through the corresponding spectrum file, to amplify or reduce the spectrum intensity or to transform the accelerations into consistent units.

7. These parameters determine the number of curves read from the corresponding spectrum file. If not specified these parameters are assumed to be 1.

10. Lateral Dynamic Time History Data

This data is for defining the ground motion accelerogram and integration parameters for a lateral dynamic time history analysis.

Skip this section of data if **IDYN** is not 3 or 4. Otherwise, prepare the following data as defined in Sections a, b, c, d, e and f below.

Format

a. Title Information

Prepare one line of data with title information to identify the acceleration time history. Up to 50 characters of information may be provided.

b. Control Data

Prepare one line of data of control information in the following form:

ANG NSTEP DT DAMP NDAMP

c. Overriding Damping Data

Provide **NDAMP** data lines, one line each for the dynamic modes for which damping ratios need to be overridden. Prepare the data in the following form:

N D

d. Acceleration History Data

Prepare two lines of data to specify base acceleration histories in the following form:

HISTFIL1 SF1 IHTYP1 HDT1 NPL1 IPRIN1
HISTFIL2 SF2 IHTYP2 HDT2 NPL2 IPRIN2

e. Nonlinear Analysis Iteration Control Data

This data is only required when nonlinear time history analysis is to be made, i.e., **IDYN** = 4. The data specifies the iteration control parameters for nonlinear analysis. Prepare the data in the following form:

DTMAX FTOL ALPHA MAXITR

f. Nonlinear Analysis Initial Load Combo Data

This data is only required when nonlinear time history analysis is to be made, i.e., **IDYN** = 4. The data specifies the static load combination to be used as the initial load in the nonlinear elements at start of the nonlinear dynamic analysis. Prepare the data in the following form:

XI XII XIII XA XB XC

Description

Variable	Field	Note	Entry
b. Control Data			
ANG	1	(1)	Direction of earthquake input (degrees).
NSTEP	2	(2)	Number of output time steps.
DT	3	(2)	Time increment for output sampling.
DAMP	4	(3)	Damping ratio (must be less than 1.0).
NDAMP	5	(3)	Number of modes for which overriding damping ratios are provided in the section below.
c. Overriding Damping Data			
N	1	(3)	Mode number.
D	2	(3)	Damping ratio (must be less than 1.0).
d. Acceleration History Data			
HISTFIL1 **HISTFIL2**	1	(4)	Filenames containing acceleration values.
SF1 **SF2**	2	(5)	Scale factors for accelerations.
IHTYP1 **IHTYP2**	3	(6)	Time history type code: = U Variable time interval. = E Constant time interval.
HDT1 **HDT2**	4	(6)	Time interval spacing for **IHTYP** = E type history (must be greater than zero).
NPL1 **NPL2**	5	(7)	Number of entries per line for history data.

Variable	Field	Note	Entry
IPRN1 **IPRN2**	6		Input time history print flag: = 0 Print. = 1 Do not print.

e. Nonlinear Analysis Iteration Control Data

Variable	Field	Note	Entry
DTMAX	1	(2,8)	Maximum time step for integration.
FTOL	2		Nonlinear force convergence tolerance.
ALPHA	3		Nonlinear force under relaxation factor.
MAXITR	4		Maximum number of iterations allowed before automatic sub stepping.

f. Nonlinear Analysis Initial Load Combo Data

Variable	Field	Note	Entry
XI	1	(9)	Multiplier for static load condition I.
XII	2		Multiplier for static load condition II.
XIII	3		Multiplier for static load condition III.
XA	4		Multiplier for static load condition A.
XB	5		Multiplier for static load condition B.
XC	6		Multiplier for static load condition C.

Notes

1. The direction angle, **ANG** is measured positively counter-clockwise from the global X-direction. See Figure V-29. Time history dynamic excitations are applied concurrently in two horizontal directions. One in the specified direction and the other at **ANG** + 90°.

2. The time span over which the time history analysis is carried out is given by **NSTEP*DT**. Member forces and displacements are calculated after every **DT** seconds, ending up with **NSTEP** + 1 values for each output component. The maximum of the **NSTEP** + 1 values for each component is printed. The time increment **DT** may be any desired sampling value that is deemed fine enough to capture the maximum response values. One-tenth of the time period of the highest mode is usually recommended; however, a larger value may give just as accurate a sampling if the contribution of the higher modes is small.

   For linear time history analysis (**IDYN** = 3) explicit modal integration is used in computing the response, therefore, numerical instability problems based on integration time step are not encountered. The time step used for integration is the smaller of the output time step **DT** and the input time steps for the two directions of the earthquake.

   For nonlinear time history analysis (IDYN = 4) explicit modal integration is also used in computing the response, however, the state of nonlinear elements is updated after each integration time step and it is assumed that the nonlinear element forces vary linearly within a time step. The time step used for integration is the smallest of the output time step **DT**, the input time steps for the two directions of the earthquake, and the value of **DTMAX** specified. This integration time step may further be divided internally in the program if iteration convergence is not achieved in the maximum number of iterations per step specified by **MAXITR**.

3. The damping is represented as a fraction of the critical damping. Therefore, **DAMP** and any **D** value specified must always be less than 1.0.

   DAMP is taken as the modal damping for each mode unless overriding values are provided, in which case **NDAMP** represents the number of modes for which overriding damping data is provided. The value of **D** then becomes the damping ratio for the corresponding mode number **N**.

   These damping ratios are additional to any program calculated damping due to supplemental dampers.

4. The **HISTFIL1** and **HISTFIL2** are text files containing the input acceleration time histories. The acceleration time history represented by file **HISTFIL1** is applied at the base of the structure at an angle **ANG** to the global X-axis. The acceleration time history represented by file **HISTFIL2** is applied concurrently to the base of the structure at an angle **ANG** + 90° to the global X-axis.

   If unidirectional motion is to be applied, one of these lines specifying the time history file should be left blank.

   The format of each file is dependent on the corresponding **IHTYP** and **NPL** parameters as described below in notes 6 and 7.

5. **SF1** and **SF2** are the multiplying factors to scale the time history acceleration values input through files **HISTFIL1** and **HISTFIL2**, respectively. These can be used to amplify or reduce the acceleration intensity or to transform the accelerations into consistent units.

6. **IHTYP** (**IHTYP1** or **IHTYP2**) defines the method that the program uses to read the time history input. If **IHTYP** is "U", the program expects to read pairs of time and associated acceleration values at the rate of **NPL** (**NPL1** or **NPL2**) entries per line. With this method of input **HDT**(**HDT1** or **HDT2**) is ignored by the program.

   If **IHTYP** is "E", the program will read history values (only) at equal intervals along the time axis at the rate of **NPL** entries per line. **HDT** is the time axis spacing for acceleration input values.

   For either "U" and "E" type inputs the first value should be at time equal 0.0.

   Comment lines are possible in the time history files. Any line beginning with a $ sign is ignored by the program.

7. This entry must be even for **IHTYP** = U type history. It is the number of entries (not pairs of entries) on any one line.

8. These entries are used to control the iterations when nonlinear analysis is made. It is recommended that users test sensitivities to these variables to verify the validity of the iteration process.

   DTMAX is the maximum value of the integration time step. If specified as zero it defaults to the output time step **DT**.

   The program uses a test on convergence of the modal contributions of the nonlinear forces as compared to the total modal forces. **FTOL** is the convergence tolerance used. If specified as zero it defaults to 1E-6. Much coarser tolerances may give acceptable results, especially with damper elements.

The program allows for under relaxation of the nonlinear forces during the iteration process. **ALPHA** of zero means no under relaxation. This is also the default value. **ALPHA** must be less than one. Some nonlinear elements, especially nonlinear damper elements may need high value of **ALPHA** for the iteration process to succeed.

The program has automatic sub stepping for nonlinear analysis. When the number of iterations in a particular time step exceed **MAXITR** the program subdivides the step into half. This process continues until the step becomes small enough that other numerical problems may be expected and the program is stopped. The default value of **MAXITR** is set to 10. A smaller value is recommended for hysteretic and gap elements and a larger value is recommended for damping elements. **MAXITR** can be set to a large number if automatic sub stepping is to be suppressed.

9. These entries are used as multipliers of the linear force responses computed for the nonlinear elements for static load cases I, II, III, A, B and C. These responses are then combined algebraically and used as initial values when time history analysis is made.

   This is especially important in computing the starting axial load in isolator elements.

11. Load Case Data

Load cases for the building are defined as combinations of the eight basic load conditions, namely:

- The vertical static load conditions, I, II and III.
- The lateral static load conditions, A, B and C.
- The lateral dynamic load conditions, D1 and D2.

Format

If **NLD** is zero, skip this data section. Otherwise, provide one data line to define each of the **NLD** load cases in the following form:

L LTYP XI XII XIII XA XB XC XD1 XD2

Description

Variable	Field	Note	Entry
L	1	(1)	Load case number.
LTYP	2	(2)	Code for load combination type: = 0 Linear combination, consider all signs. = 1 Linear combination, use absolute values of responses but consider signs of multipliers. = 2 SRSS A and B, combine linearly with others. = 3 SRSS D1 and D2, combine linearly with others.
XI	3	(3)	Multiplier for vertical static load condition I.
XII	4		Multiplier for vertical static load condition II.
XIII	5		Multiplier for vertical static load condition III.
XA	6		Multiplier for lateral static load condition A.
XB	7		Multiplier for lateral static load condition B.
XC	8		Multiplier for lateral static load condition C.
XD1	9		Multiplier for lateral dynamic load condition D1.
XD2	10		Multiplier for lateral dynamic load condition D2.

Notes

1. This number must be input in an ascending, consecutive sequence, starting with the number 1.

2. If this entry is 0, linear combinations are produced and all signs are considered.

   If this entry is 1, linear combinations are produced, except that absolute values of responses are used, although signs of multipliers are considered. This type of combination allows the user to conservatively obtain the net effects of combinations with a minimum number of load cases.

   If this entry is 2, a square root of the sum of the squares (SRSS) combination of the lateral static load conditions A and B responses with the specified multipliers is first made before combining linearly with the other load conditions. The SRSS value is assigned the sign of **XA**. This type of combination is required in some design codes for considering orthogonal effects of seismic excitations.

   If this entry is 3, a SRSS combination of the lateral dynamic load conditions D1 and D2 responses with the specified multipliers is first made before combining linearly with the other load conditions. The SRSS value is assigned the sign of **XD1**. This type of combination is commonly used for dynamic analysis and is required by some design codes to consider orthogonal effects of seismic excitation.

3. Any multiplier entered in the fields associated with inactive load conditions will have a null effect on the load case.

   If **IDYN** is less than 2, none of the lateral dynamic load conditions will be active.

   If **IDYN** is 2 (i.e. response spectrum analysis), the lateral dynamic load condition **D1** contains the responses for base excitation in the direction defined by the parameter **ANG** specified in the response spectrum data and **D2** contains the responses for base excitation in the direction defined by **ANG** + 90°.

   If **IDYN** is 3 or 4 (i.e. time history analysis), only lateral dynamic load condition **D1** is active and contains the response maxima from the time history analysis.

Chapter VI

ETABS Output Files

Depending upon the options that are activated by the ETABS input data, the program will produce up to eight output files containing information produced by the different phases of the analysis.

The names of these output files are built from the input file name with unique file name extensions. For instance, if the name of the ETABS input data filename is ***etabsfile***, the eight output files produced by ETABS will be:

- ***etabsfile***.EKO Echo of the Input Data
- ***etabsfile***.EIG Time Periods and Mode Shapes
- ***etabsfile***.STR Story Displacements and Forces
- ***etabsfile***.DSP Frame Displacements
- ***etabsfile***.FRM Frame Member Forces
- ***etabsfile***.SUM Analysis Summary
- ***etabsfile***.LOG Execution Log
- ***etabsfile***.PST Post Processing Information

The output files with extensions .EKO, .EIG, .STR, .DSP, .FRM, .SUM and .LOG are ASCII files and can be viewed and printed using the BROWSER program or printed from DOS using the **PRINT** command.

The output file with the .PST extension is in binary format that cannot be printed or scanned with an editor. This file forms the interface between ETABS and the ETABS post processors, ETABSOUT, STEELER, CONKER and WALLER.

The following is a short description of the contents of each of the above mentioned ASCII files.

A. The *etabsfile*.EKO File

This output file contains a tabulated and labeled listing of the input data as it has been read and generated from the input data file.

This file also contains additional information about the model that has been calculated by the program from the input data before beginning the structural analysis. This data is helpful in cross-checking the accuracy of the input data. The additional information includes:

- Calculated diaphragm mass data
- Calculated or recovered section properties
- Calculated beam, brace and link element lengths and floor areas
- Calculated wall assemblage areas and centroids
- Summations of frame vertical and lateral loads by levels
- Summations of frame element weights by levels
- Summations of frame element masses by levels
- Recalculated diaphragm masses and centroids including element mass
- Calculated code based wind loads by levels
- Calculated code based equivalent static seismic loads, by levels, if building period is specified.

B. The *etabsfile*.EIG File

This output file contains information from the eigenvalue solution of the building.

This file contains the following information:

- Time periods and frequencies
- Modal participation factors
- Modal direction factors
- Modal effective mass factors
- Structural mode shapes

The structural mode shapes are in the form of components associated with the centers of mass of each diaphragm, at each story level. See Figure VI-1 for the positive sign convention. As all dynamic analysis options in ETABS are of a lateral nature, only the global X- and Y-translation and Z-rotation components are included.

The definition for the various other quantities printed are as follows:

1. The modal participation factors corresponding to the X, Y and Θ_Z directions for a particular mode are defined as:

$$PF_X = \sum M\, \Phi_X \qquad PF_Y = \sum M\, \Phi_Y \qquad PF_{\Theta Z} = \sum M_\Theta\, \Phi_{\Theta Z}$$

   where **M** represents the translational mass, Φ_X represents the X-translational mode shape components, Φ_Y represents the Y-translational mode shape components, M_Θ represents the mass moment of inertia, and $\Phi_{\Theta Z}$ represents the Z-rotational mode shape components.

   All summations are over all diaphragms and over all stories.

2. The modal direction factors identify the predominant direction of excitation associated with each of the modes. The factors are percentages associated with the X- and Y-translational and Z-rotational directions. The sum of the three values will always add up to 100.

The factors for a particular mode are given by:

$$\%X = 100 \sum M \Phi_X^2 \quad \%Y = 100 \sum M \Phi_Y^2 \quad \%\Phi_Z = 100 \sum M_\Theta \Phi_{\Theta z}^2$$

where

$$\sum M \Phi^2 = \sum M \Phi_X^2 + \sum M \Phi_Y^2 + \sum M_\Theta \Phi_{\Theta Z}^2 = \mathbf{1.0}$$

Again all summations are over all diaphragms and over all stories.

3. The modal effective mass factors identify the percentage of the total mass that is mobilized by each of the modes when the structure is subjected to a constant unity response spectrum applied in a particular global direction.

   The factors for a particular mode are calculated from the participation factors as follows:

$$EM_X = 100 \frac{(PF_X)^2}{\sum M} \quad EM_Y = 100 \frac{(PF_Y)^2}{\sum M} \quad EM_{\Theta Z} = 100 \frac{(PF_{\Theta Z})^2}{\sum M_{\Theta Z}}$$

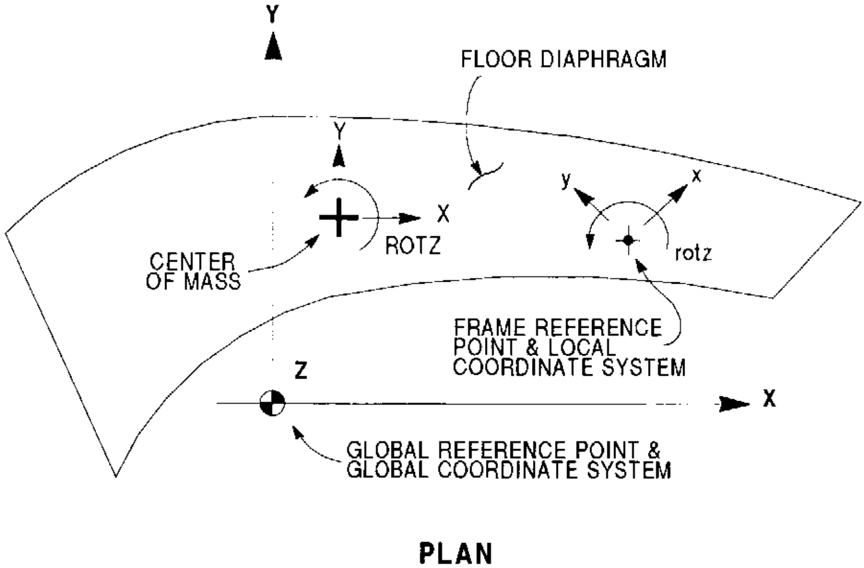
   This information is helpful in determining the number of modes that need to be included in the implementation of a dynamic analysis. It is recommended that the cumulative mass percentage from all the modes that are included in the analysis should not be less than 90% for any direction of excitation. If all modes are included, the sum of the mass percentages from each direction will equal 100%.

C. The *etabsfile*.STR File

This output file contains structural level information associated with the analysis of the structure.

Depending upon the analytical options that are activated, this file can contain the following information:

- Calculated centers of cumulative mass for each diaphragm. The cumulative mass for a particular diaphragm represents the summation of the masses of all



Positive Directions for Structural and Frame Lateral Displacements
Figure VI-1

the diaphragms (with the same diaphragm number) from all levels above and including the particular diaphragm.

- Calculated centers of rigidity for each diaphragm. Lateral translational loads applied at center of rigidity of a rigid floor diaphragm will produce displacements containing only translational (X and/or Y) components, with no rotation, provided that there are no loads applied on any of the other diaphragms.

- Calculated code based equivalent static seismic loads at each level. If the building period is not specified the loads are based upon the calculated periods.

- Total modal damping for linear dynamic analysis including the effect of supplemental dampers. This assumes proportional damping.

- Response spectrum values used in the analysis at the different modal periods and damping.

- Maximum values of input energy, kinetic energy, strain energy, energy loss due to viscous damping (modal and supplemental damping), energy loss due to hysteresis and friction and the error in energy balance are all reported with their corresponding times of occurrence for linear and nonlinear time history analyses.

- Lateral story displacements at the centers of mass of each diaphragm at each story. The displacements are in the global coordinate system. See Figure VI-1 for the positive sign convention.

- Lateral loads (inertia loads for dynamic analysis) at the centers of mass of each diaphragm at each story. The loads are in the global coordinate system.

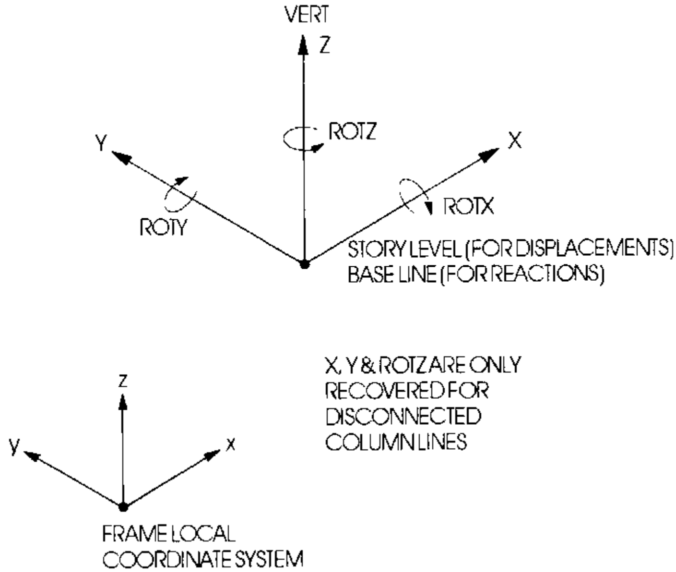
- Lateral story shears at the centers of mass of each diaphragm at each story. The story shears are cumulative lateral loads summed from the top story downwards for each diaphragm number separately. Story shears for the total building are also given as the summation of lateral loads of all diaphragms above a certain story. The story shears are in the global coordinate system.

- For each frame in the structure, lateral story displacements and lateral inter-story drift ratios for each diaphragm at each story at the point in the plan of the structure that corresponds to the frame reference point. The displacements and drift ratios are in the frame local coordinate system. See Figure VI-1.

  The inter-story drift ratio for a diaphragm at a story is defined as the difference between the lateral story displacements at that diaphragm at that story and the story below, divided by the story height associated with that story. Missing diaphragms that result in taller inter-story heights are automatically taken into consideration.

- For each frame in the structure, lateral story shears for each diaphragm at each story at the point in the plan of the structure that corresponds to the frame reference point. The story shears are cumulative lateral loads summed from the top story downwards for each diaphragm number separately. The story shears are in the frame local coordinate system.

  The story displacements, story loads and story shears at the structural and frame levels are produced for static load conditions I, II, III, A, B and C if static lateral loads are specified (i.e., **ILAT** is not 0). They are produced for dynamic load conditions D1 and D2 for response spectrum analysis (i.e., **IDYN** = 2) and represent individual maxima computed from modal responses. These values for response spectrum are always reported as positive. They are produced for dynamic load condition D1 for time history analysis (i.e., **IDYN** = 3 or 4). In this case both maximum and minimum values are reported with their corresponding times of occurrence.



Positive Directions for Frame Joint Displacements and Reactions
Figure VI-2

D. The *etabsfile*.DSP File

This output file contains the frame displacements for each of the **NLD** load cases.

For each joint in each frame, the total (not inter-story) displacements are output in the frame local coordinate system. See Figure VI-2 for the positive sign convention.

All output is in the same sequence as the frame location data (see Chapter V, Section 7). Output associated with the first frame is printed first, starting with the top story and progressing downward.

If **NLD** is 0, this file will be empty.

E. The *etabsfile*.FRM File

This output file contains all member forces for each element type for each of the **NLD** load cases.

Depending upon the types of elements used in the model this file can contain the following information:

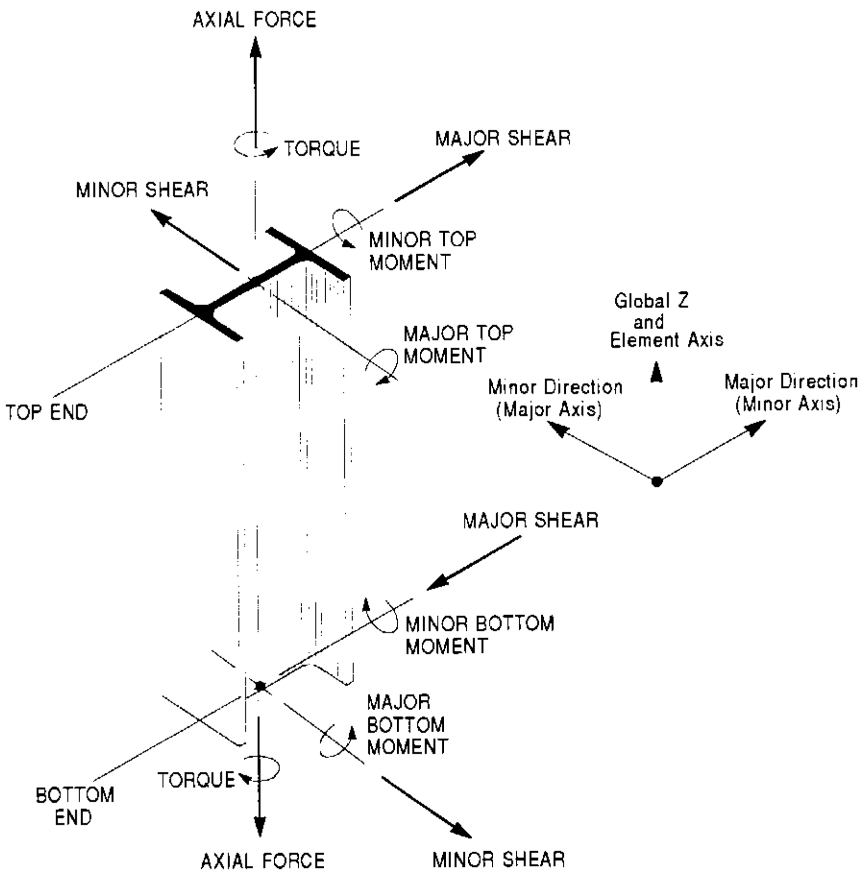
- For each column element, major and minor moments are output at each end of the column, at the outer faces of the beams framing into the column. Axial force, shear forces and torsion are constant over the column height and are therefore only printed once, i.e., at the top end. See Figure VI-3 for the positive sign convention.

- For each beam element, major and minor moments are output at each end of the beam at the outer faces of the columns framing into the beam. If the frame contains any beam span loading data or any floor element loading data the beam bending moments and major shear forces are output at three additional stations, namely the 1/4, 1/2 and 3/4 clear span points. Axial forces, minor shear forces and torsions that are constant over the beam length are only printed once, i.e. at End I of the beam. See Figure VI-4 for the positive sign convention.

  Axial forces in horizontal beams will only be recovered if at least one of the joints at the end of the beam is released from the rigid diaphragm and the beam axial properties are specified. In many cases, such as in braced frames, the structural behavior can cause large axial forces to be generated in a beam, but these forces are not recovered by the program if both ends of a horizontal beam connect to a rigid diaphragm, because the rigid diaphragm assumption forces the axial deformations in the beam to be zero. In reality, the beams have axial forces and deformations, and situations involving heavy axial action should be closely examined.

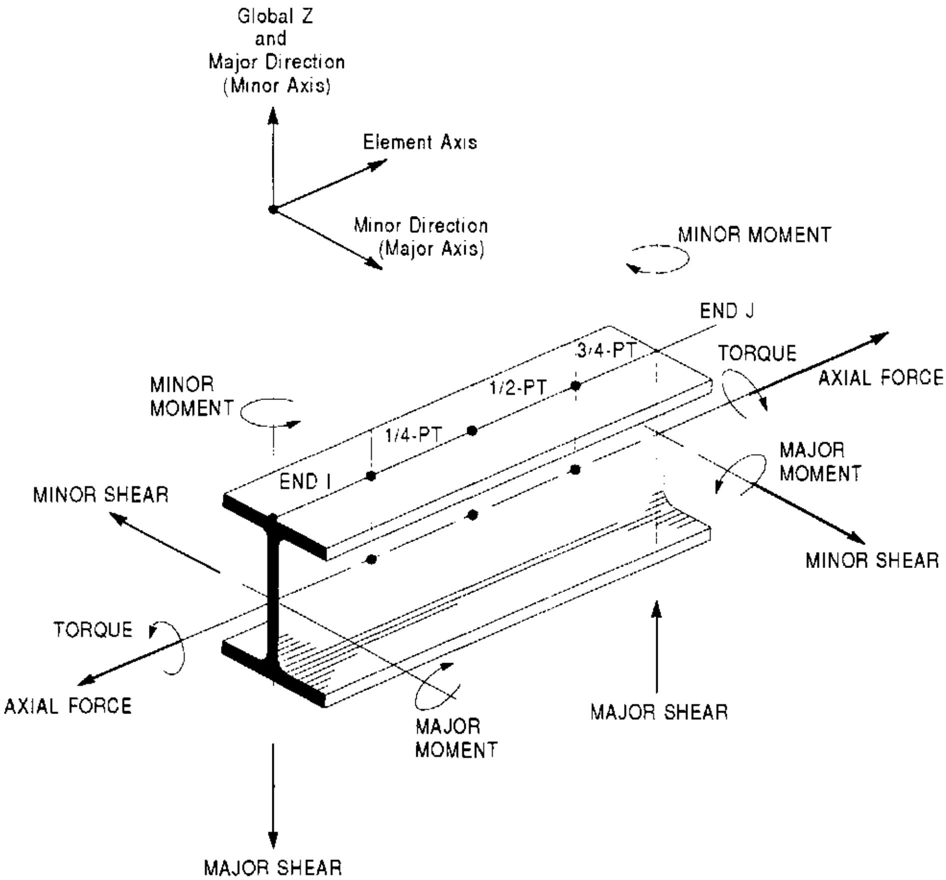
- For each floor element, nodal forces are printed at each node in the floor local coordinate system. See Figure V-12 for the floor local coordinate system.

- For each brace element, major and minor moments are output at each end of the brace. Axial forces, shear forces and torsions are constant over the length of the brace and are therefore only printed once, i.e., at the top end. See Figure VI-5 for the positive sign convention.



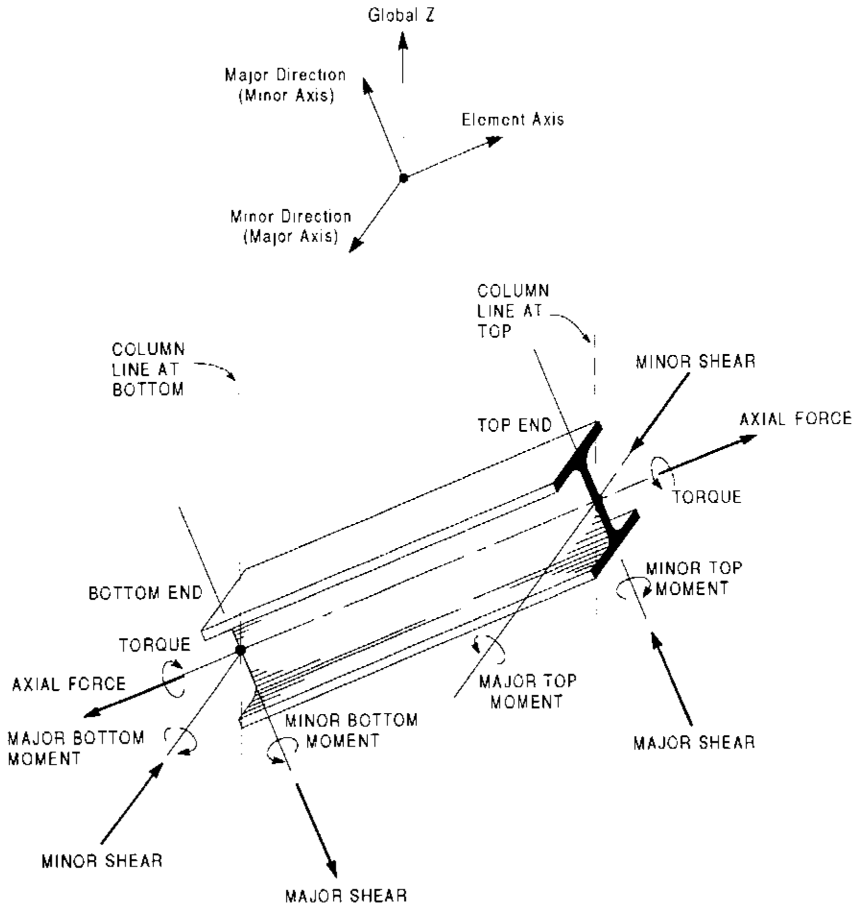
NOTE: Positive moments cause compression on the positive face, i.e. the element face on the side of the positive axis.

Column Member Forces
Figure VI-3



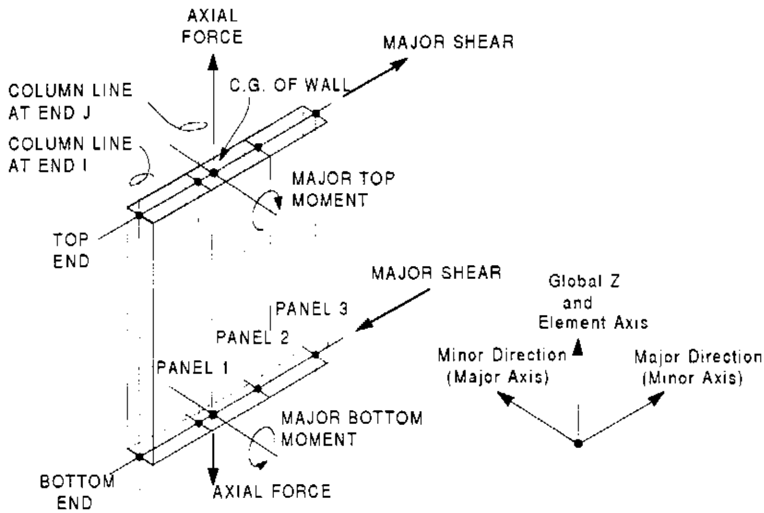
NOTE: Positive moments cause compression on the positive face, i.e. the element face on the side of the positive axis.

Beam Member Forces
Figure VI-4

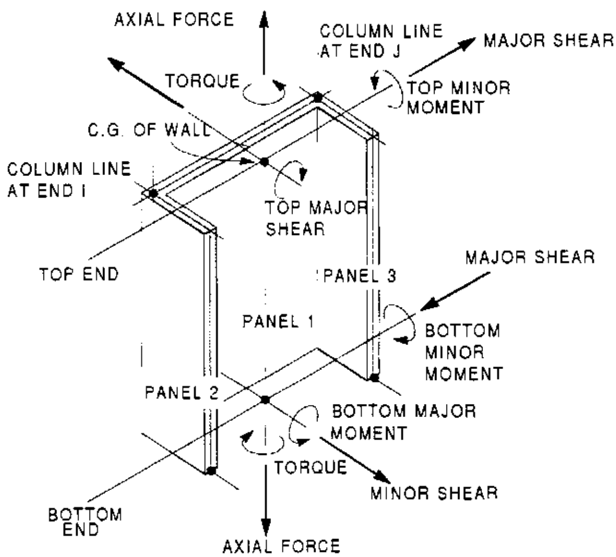


NOTE: Positive moments cause compression on the positive face. i.e. the element face on the side of the positive axis.

Brace and Link Member Forces
Figure VI-5



(a) 2-D WALL (PLANAR)



(b) 3-D WALL

NOTE: Positive moments cause compression on the positive face, i.e. the element face on the side of the positive axis.

Wall Assemblage Member Forces
Figure VI-6

- For each wall section (assemblage of panel elements), major and minor moments are output at each end of the assemblage at the floor levels. Axial forces, shear forces and torsions are constant over the height of the wall and are therefore only printed once, i.e., at the top end. See Figure VI-6 for the positive sign convention.

  The wall section major direction is defined by the orientation of the panel element that is first in the input (or generated) sequence among the panel elements that define the particular wall section at the particular level.

- For each link element, major and minor moments are output at each end of the link. Axial forces, shear forces and torsions are constant over the length of the link and are therefore only printed once, i.e., at the top end. See Figure VI-5 for the positive sign convention.

- For each spring element, major and minor moments, major and minor shear forces, axial force, and torsional moment are printed once, in the coordinate system of the column at that location.

All output is in the same sequence as the frame location data (see Chapter V, Section 7). Output associated with the first frame is printed first, starting with the top story and progressing downward.

If **NLD** is 0, this file will be empty.

F. The *etabsfile*.SUM File

This file contains a summary of the frame results. The results are based on the **NLD** load cases and are arranged by frames. For each frame the file contains the following:

- Maximum and minimum total displacements

- Maximum and minimum element forces for each property type

- Baseline reactions at each column line

- Summation of reactions at baseline.

All displacements and reactions are in the frame local coordinate system. See Figure VI-2 for the positive sign convention. For positive sign convention of member forces see Figures VI-3 to VI-6.

All output is in the same sequence as the frame location data (see Chapter V, Section 7).

If **NLD** is 0, this file will be empty. Also, the baseline reactions and summations are not included in the file if any nonlinear element contributes to the reactions.

G. The *etabsfile*.LOG File

This file contains the progress messages that are displayed on the screen during execution. All warning and error messages are written to this file. The user should check this file if the execution is prematurely terminated.

H. The *etabsfile*.PST File

This file is the ETABS postprocessing file. The file contains information pertaining to the building geometry and loading and the analytical results from the corresponding ETABS analysis. It forms the interface between ETABS and the ETABS postprocessors.

References

1. **Wilson, E. L. and Habibullah, A.**

SAP90 - A Series of Computer Programs for the Finite Element Analysis of Structures, Computers and Structures, Inc., Berkeley, California,1992.

2. **Bathe, K. J., Wilson, E. L. and Peterson, F. E.**

SAPIV - A Structural Analysis Program for Static and Dynamic Response of Linear Systems, EERC, Report No. 73-11, June 1973, Revised April 1974.

3. **Clough, R. W., King, I. P. and Wilson, E. L.**

Structural Analysis of Multistory Buildings, Journal of the Structural Division, ASCE, Vol. 89, No. 8, 1963.

4. **Wilson, E. L. and Dovey, H. H.**

Three-Dimensional Analysis of Building Systems - TABS, EERC, Report No. 72-8, University of California, Berkeley, California, 1972.

5. **Wilson, E. L., Hollings, J. P. and Dovey, H. H.**

ETABS, Three-Dimensional Analysis of Building Systems - Extended Version, EERC, Report No. 75-13, University of California, Berkeley, California, 1975.

6. **Wilson, E. L. and Habibullah, A.**

   A Program for Three-Dimensional Static and Dynamic Analysis of Multistory Buildings, Structural Mechanics Software Series, Vol. II, University Press of Virginia, 1978.

7. **Wilson, E. L., Dovey, H. H. and Habibullah, A.**

   Theoretical Basis for CTABS80: A Computer Program for Three-Dimensional Analysis of Building Systems, Technical Report K-81-2, Computers / Structures International, Oakland, California, 1981.

8. **Wilson, E. L. and Habibullah, A.**

   Static and Dynamic Analysis of Multi-Story Buildings Including P-Delta Effects, Earthquake Spectra, Vol. 3, No. 2, Earthquake Engineering Research Institute, Oakland, California, 1987.

9. **Wilson, E. L., Kiureghian, A. D. and Bayo, E.**

   A Replacement for the SRSS Method in Seismic Analysis, Earthquake Engineering and Structural Dynamics, Vol. 9, 1981.

10. **Gupta, A. K.**

    Response Spectrum Method, Blackwell Scientific Publications, Ltd., 1990.

11. **Wilson, E. L.**

    An Efficient Computational Method for the Base Isolation and Energy Dissipation Analysis of Structural Systems, ATC 17-1, Proceedings of Seminar on Seismic Isolation, Passive Energy Dissipation, and Active Control, Applied Technology Council, Redwood City, California, 1993.

12. **Ibrahimbegovic, A. and Wilson, E. L.**

    Simple Numerical Algorithms for the Mode Superposition Analysis of Linear Structural Systems with Nonproportional Damping, Computers and Structures, Vol. 33, No. 2, 1989.

13. **International Conference of Building Officials**

    Uniform Building Code, Whittier, California, 1994.

14. Applied Technology Council, ATC 3-06

Tentative Provisions for the Development of Seismic Regulations for Buildings, June 1978.

15. Building Officials and Code Administrators International, Inc.

The BOCA National Building Code/1993, Country Club Hills, Illinois, 1993.

16. American Society of Civil Engineers

Minimum Design Loads for Buildings and Other Structures - ASCE 7-93, New York, New York, 1993.

17. National Research Council of Canada

National Building Code of Canada 1990, Ottawa, Canada 1990.

18. Wen, Y. K.

Method for Random Vibration of Hysteretic Systems, Journal of the Engineering Mechanics Division, ASCE, Vol. 102, No.EM2, 1976.

19. Park, Y. J., Wen, Y. K. and Ang, A. H-S.

Random Vibration of Hysteretic Systems Under Bi-Directional Ground Motions, Earthquake Engineering and Structural Dynamics, Vol. 14, 1986.

20. Nagarajaiah, S., Reinhorn, A. M. and Constantinou, M. C.

3D-BASIS: Nonlinear Dynamic Analysis of Three-Dimensional Base Isolated Structures: Part II, Technical Report NCEER-91-0005, National Center for Earthquake Engineering Research, State University of New York at Buffalo, Buffalo, New York, 1991.

21. Zayas, V. and Low, S.

A Simple Pendulum Technique for Achieving Seismic Isolation, Earthquake Spectra, Vol. 6, No. 2, Earthquake Engineering Research Institute, Oakland, California, 1990.

Index

J

L

M

N

O