EVALUATION OF SOFTWARE FOR ANALYSIS AND DESIGN OF REINFORCED CONCRETE STRUCTURES

by

Andrew D. Betaque

Thesis submitted to the Graduate Faculty of the

Virginia Polytechnic Institute and State University

in partial fulfillment of the requirements for the degree of

MASTER OF SCIENCE

in

CIVIL ENGINEERING

APPROVED:

K. B. Rojiani, Chairman

R. M. Barker

D. A. Garst

December, 1992

Blacksburg, Virginia

Evaluation of Software for Analysis and Design of Reinforced Concrete Structures

(ABSTRACT)

A study was performed to compare the linear static analysis and concrete design capabilities of two structural analysis and design programs - MicasPlus and STAAD III. Four structures were considered including a four span continuous beam, a two story 3-D frame, a seven story 3-D frame, and a fourteen story 3-D frame. The study compared the accuracy of the programs as well as their functionality. In the evaluation of the accuracy of analysis results, factors such as support reactions, nodal displacements, and element end forces were compared. The evaluation of the concrete design capabilities was based on comparing the reinforcement recommended by the two programs for beams and columns. For beam design, steel reinforcing bar selections for positive moment, negative moment, and shear reinforcing were compared. For column design, the cross sectional area of steel chosen by each program was compared. Factors considered in the evaluation of functionality included: user interface, ease of use, ease of learning, quality of output, documentation, flexibility, and analysis and design capabilities.

Acknowledgements

The author extends his sincere appreciation to Dr. Rojiani for his guidance and assistance with this thesis.

The author thanks Professor Barker and Professor Garst for being on his thesis committee.

The author thanks Guner Eruren, Mike Parscale, and Wallace Wyeth of Cromwell A/E for help with and use of computers and other equipment.

The author thanks his wife, Betsy Betaque, for her patience and moral support.

The author thanks his parents, Norm and Arleen Betaque, for a lifetime of moral and financial support without which this thesis would not have been possible.

Table of Contents

List of Illustrations

List of Tables

Chapter 1

Introduction

1.1 Computers in Structural Engineering

Computers play an important role in structural engineering. They allow a faster and easier alternative to manual computations for the analysis and design of structures. One advantage to using computers in structural engineering is that a more complex analysis can be performed than by hand. The computer can quickly analyze three-dimensional multistory frames as well as simpler two dimensional frames. The use of computers allows factors such a torsion and axial shortening to be accounted for that were previously often neglected. Another advantage to using computers is that many load cases and load combinations can be considered without significantly increasing analysis and design time. This allows the engineer to quickly test all possible loadings to ensure that the building will be safe. With the use of modern structural analysis and design programs, a large building can be designed in a matter of hours or days. However, the engineer must decide if it is wise to unconditionally trust the computer generated results.

Computers cannot be used without great caution however. There are two possible sources of error when using a computer program for structural analysis and design. The first possible source of error is an improperly modeled structure. It is important for the structural engineer to be aware of the fact that a computer analyzes a mathematical model of a real structure. The mathematical model must behave similarly to the real structure in order for a computer program to accurately predict the behavior of a structure. The second possible source of error is the computer program itself. It must not be assumed by the engineer that a program is correct. Studies in the past have shown that several well known structural engineering programs may differ from one another in their analysis and design results. It should be noted that the use of a computer program does not relieve the engineer of responsibility for the structural integrity of a structure. It is therefore the responsibility of the engineer to verify any computer generated results.

An important issue to consider when discussing computer programs is the functionality of the software. Functionality refers to the less technical aspects of the program such as the documentation, the ease of use, flexibility, and the quality of the program. Functionality is an important factor in the selection of a particular computer program for two reasons. First, it is likely that an engineer will spend many hours each day using the program. Therefore, the functionality of a program can greatly affect the productivity of the engineering office. Second, a program with poor functionality may have a higher potential for errors. The most important reason that a program should have good functionality is safety. A program must make it simple for the engineer to verify the input and to assure that the computer is modeling the same structure that the engineer intended to model. Without this, a large structure would be very difficult to model with any degree of confidence.

1.2 Objectives of Study

The objectives of this study are to verify the analysis and concrete design results and to evaluate the functionality of MicasPlus (Intergraph Corporation) and STAAD III (Research Engineers, Inc.) for both their analysis and concrete design capabilities. It is not possible to completely verify each program. However, it can be determined if there is an agreement in the results produced by each program. The results of this study should help to determine if a further investigation into the quality of either of these two programs is necessary or desired.

The verification process is based on a comparison of the analysis and design results generated by MicasPlus and STAAD III for four concrete structures. The analysis results for the continuous beam are also compared to a third structural analysis program created by Holzer (Holzer, 1985). MicasPlus column design results are compared to hand calculations instead of STAAD III results due to an error in the STAAD program which makes its column design unreliable. The four structures considered include a four span continuous beam, a two story 3-D frame, a seven story 3-D frame, and a fourteen story 3-D frame. Nodal displacements, member end forces, and support reactions are compared in order to verify the accuracy of the analysis modules of the programs. The verification of the design modules is based on the required amount of steel selected for beams and columns.

The evaluation of the functionality will be the result of both objective and subjective reasoning. Subjective factors include ease of use, documentation, and overall quality of the program. Objective factors include features, capabilities and limitations of the program.

1.3 Organization of Chapters

Chapter 1 is an introduction to this study. Its purpose is to familiarize the reader with the role of computers in structural engineering. It also describes the objective of this study.

Chapter 2 contains a literature review of related studies that have been done in the past. It describes these studies and relates their importance to this study.

Chapter 3 contains an overview of MicasPlus and STAAD III. It describes the features, capabilities, and hardware requirements of both programs.

Chapter 4 contains a description of the structures that are compared. It describes the geometry, loading, and the member sizes used for the analysis of each structure.

Chapter 5 contains the results of this study. It contains tables of the analysis and design results generated by MicasPlus and STAAD III for selected members.

Chapter 6 compares the functionality of MicasPlus and STAAD III.

Chapter 7 summarizes the study and presents the conclusions.

Chapter 2

Literature Review

2.1 Literature Review

As computer technology advances, structural engineering software becomes more complex. The increase in software complexity has benefits as well as drawbacks. The benefits include more accurate modeling of structures, faster execution times, and a wider range of structures which can be modeled. The biggest drawback is that of verifying the accuracy and quality of the programs. Verification is done to "determine that (the software) performs its intended functions correctly, to ensure that it performs no unintended functions, and to measure its quality and reliability" (Wallace and Fujii, 1989). In addition to the need to verify the accuracy of a program is the need to evaluate it. This includes rating the functionality, the features, and the general quality of the program. The following pages will describe some aspects of software verification and evaluation. Also, a review of several relevant studies that have been done in the past is presented.

Independent verification of a program is necessary for several reasons. First, although verification should be considered an important part of the development of any program, this is not always the case. Approximately 20-30% of the development time of a program should be allotted to the test phase (Priest, 1988). This is expensive so it is often the first thing that is neglected when a company tries to cut costs. Second, a program is usually tested module by module, while little testing is done on the program as a whole. This is because it is much more expensive to fix an error after the program is completed than when it is in the design phase and because no techniques exist for thoroughly testing a program after completion (Priest, 1988). Third, the structural engineering company, not the software company, is liable for any structure that is built using the software.

Evaluation of a program is also necessary. Even the cheapest structural program is expensive. Therefore, it should be determined that a program performs the necessary functions and is easy to use. Several questions should be asked when evaluating software. How many functions does the software perform? Is the software easy to use? Is it a broad program that can handle a wide range of situations? (Machover, 1989). These are just a few of the questions that must be answered.

One verification and evaluation study was done by Kumar (1991). The objective of the study was to evaluate the concrete analysis and design capabilities of STAAD III structural engineering software which is marketed by Research Engineers, Inc. This was done by comparing the analysis and design results generated by STAAD III to another source of results. The analysis results generated by STAAD III were compared to those of a frame analysis program written by Holzer (1985). The design results were compared to hand calculations.

Two example problems were considered in the study including a simple one story frame and a multistory frame with multiple bays. Many loading conditions were applied including gravity, wind and earthquake loads. Many aspects of the results were compared including the effect of diaphragms in a concrete structure, effect of joint size, nodal displacements, forces and member selection.

The results of the verification were as follows. Results from STAAD III analysis and Holzer's analysis program were almost identical. The results of the concrete design from STAAD III were very close to those obtained from hand calculations. However, STAAD III does not design all values necessary for concrete design including development lengths, anchorage requirements, minimum shear reinforcement, sizing and spacing of ties in a column, and several others (Kumar, 1991).

The results of the functionality evaluation were as follows. As would be expected, STAAD III has many advantages as well as some disadvantages over hand calculations. The most notable advantages are the following: STAAD III runs on a PC, it is reliable, it is convenient, it saves much time over hand calculations. The major disadvantages are the following: STAAD III requires a lot of memory on a PC, it is not user friendly, and large structures cannot be designed with STAAD III on a PC (Kumar, 1991).

Another software evaluation and verification study was performed by White (1991). The purpose of White's study was to evaluate and verify the analysis and steel design capabilities of two programs including Intergraph's MicasPlus and GTSTRUDL. For the comparison, three structures were analyzed and two were designed including both braced and unbraced structures. The structures considered were existing industrial buildings consisting entirely of wide flange sections and concrete slabs. One four story braced frame, one four story moment resisting frame, and a 10 story braced frame were considered.

Each structure was analyzed and designed using both MicasPlus and GTSTRUDL. Many different load combinations were considered. Nodal displacements, force resultants, maximum moments, and several other factors were compared in order to test the analysis capabilities of MicasPlus. Member selection and unity checks were used as the basis of comparison for the design capabilities of the two programs (White, 1991).

White concluded the following. MicasPlus is reliable and accurate for linear static analysis and design of small three dimensional steel frame structures (less than 200 members). However, as the size of the building increases, the reliability of MicasPlus decreases. Results obtained by MicasPlus and GTSTRUDL for the support reactions of the 10 story building did not agree closely (within 15%). Nodal displacements also differed substantially (within 10%). However, reactions above 20 kips and displacements over 1 inch improved agreement to 8%. White concluded that the functionality of MicasPlus is very good (White, 1991).

Very few studies have been performed on testing and validating structural engineering software. This is somewhat surprising given the importance of the reliability of a structural program. This thesis will contribute to the studies done by Kumar and White by comparing MicasPlus and STAAD III for analysis and concrete design capabilities.

Chapter 3

Overview of MicasPlus and STAAD III

3.1 Overview of Chapter

This chapter provides an introduction to MicasPlus and STAAD III. It describes the purpose, features, and hardware requirements of both programs.

3.2 Introduction to MicasPlus

MicasPlus (marketed by Intergraph Corporation) is a structural engineering program used for the analysis and design of two and three dimensional structures. MicasPlus is composed of three modules including MicasPlus ModelDraft, MicasPlus Analysis, and MicasPlus Design. The three modules are integrated through the use of a database. This allows the three modules to exchange data in order to work as a unit. The advantage to this is that a model created in ModelDraft may be used in MicasPlus Analysis and MicasPlus Design. Changes made in one module will be reflected in the other modules.

MicasPlus is intended to analyze and design two and three-dimensional structures. This is accomplished through the creation of a geometric model of a real structure using MicasPlus ModelDraft or MicasPlus Analysis. MicasPlus Analysis then breaks the structure into finite elements for analysis. Using finite element analysis, MicasPlus has the ability to perform linear static analysis, limited non-linear static analysis, and dynamic analysis of the geometric model. Results of an analysis may be transferred to MicasPlus Design via the database. MicasPlus Design is capable of performing steel design and concrete design.

MicasPlus requires extensive hardware and software. Minimum hardware requirements include an Intergraph Clipper based workstation with at least 12 megabytes of RAM and 200 megabytes of hard drive space. Minimum software requirement include Intergraphs MicroStation, MicasPlus Analysis, and MicasPlus Design. Intergraphs Intersect is desirable but not required.

The most important feature of MicasPlus is its use of a computer aided design

(CAD) environment for its user interface. A CAD environment allows the user to "draw" the structure on the computer screen using a mouse or digitizer rather than using an alphanumeric interface to enter the properties of the structure. The main advantage to this type of interface is that the user can visually verify or edit the geometry and loading of the structure. A CAD interface also helps MicasPlus to perform many postprocessing functions such as the creation of shear and moment diagrams, drawing deflected shapes of the structure, and labeling reactions. MicasPlus refers to the CAD environment as the "graphical interface".

MicasPlus uses an Intergraph CAD program called MicroStation to provide its CAD capabilities. MicasPlus and MicroStation use graphical representations of buttons (called "icons") located along the top and right edges of the screen to activate most of the functions. For example, a column can be drawn on the screen by placing an arrow (which is controlled by the mouse or digitizer) on top of the appropriate icon and pressing the mouse button. The user will then be prompted for information such as the location, orientation, and size of the column. Most of this information can also be entered using the mouse. Most modeling, pre-processing, and post-processing functions available in MicasPlus are performed in a similar manner.

3.3 MicasPlus ModelDraft

MicasPlus ModelDraft is one of the three modules that form MicasPlus. Its main purpose is to aid in the creation of a geometric model of a structure. It also performs material reporting, drawing extractions, and drawing production functions. Traditionally, it is necessary to create one model for use in drawing composition and another for use in analysis. This is not necessary with ModelDraft. The diverse capabilities of ModelDraft allow the same model to be used for both purposes.

3.3.1 Model Creation in ModelDraft

The most important feature of ModelDraft is the ability to create three dimensional models. It does this through the use of the graphical interface and simple icon menu-driven commands that allow the user to place different elements together to form the model of the structure. Examples of element placement commands include place beam, place column, and place brace.

There are four types of elements that can be placed when creating a model. They are: members, arc elements, area elements, and volume elements. Members are linear elements that represent either a column, beam, or brace. Arc elements represent curved structural elements. Area elements are 2-D elements used to represent slabs or walls. Finally, volume elements are 3-D elements used to model solids. Members are the most commonly placed elements and will be discussed briefly in order to provide the reader with an understanding of the general process involved in placing a member.

There are three steps involved in placing a member in ModelDraft. They are: place a member line, assign a cross section, and define the orientation of the cross section. First, a member line must be placed. A member line defines the location of the member to be placed. ModelDraft provides tools to aid the placement and manipulation of member lines. The next step is to assign a cross section to the member. It is this step where the user must specify a beam, column, or a brace. The final step is to provide the orientation of the member in space. These steps are performed by providing input to ModelDraft either by using the keyboard to respond to prompts or by using the mouse.

Other features of element placement include ModelDraft's ability to place holes and it's Parametric Modeling Language (PML). The PML is used to place multiple components in a structure that differ only slightly from one another. For example, a group of components that differ only in their length could be modeled using PML.

ModelDraft allows the user to place grids, which are used to ease modeling. A grid is a series of lines on the screen that are used to simplify the placement of members lines. The basic form of a grid is a set of parallel horizontal lines with another set of parallel horizontal lines running perpendicular to these. This type of grid may be used to represent the location of floor beams, for example. The main advantage to using a grid is that the location of members may then be entered into the computer using the mouse, rather than typing in node coordinates with the keyboard. Complex grids lines can be generated, if necessary, using editing tools provided by ModelDraft. These tools allow grids to be copied, deleted, scaled, and otherwise manipulated.

Another ability of ModelDraft is the ability to manipulate views. This allows the user to see the model in an isometric view, top view, left view, right view, or front view. In addition to these predefined views, the user has the ability to create his or her own views and recall them later. Views may be rotated, also. With the use of this feature, a model can be rotated in order to view it from any orientation. This is often useful when placing members or preparing to plot. It is also a useful tool to help determine if the model being developed on the screen is actually the desired model.

3.3.2 Model Manipulation in ModelDraft

ModelDraft provides extensive tools for manipulating elements. These commands provide the user with the ability to copy, delete, move, and modify member lines or fully-defined members.

The copy command has many variations. It is used to make one or more exact copies of a member or member line. In addition to the basic form of the copy command, ModelDraft provides advanced copy techniques. For example, one function combines a member rotation with the basic copy command. Mirror copies may be performed in order to aid the creation of a symmetrical structures. Another variation of the copy command allows a member to be copied between two skewed lines, with the length of the member being adjusted automatically.

The delete command is used to delete member cross sections, entire members, or a grid set. This allows for the easy removal of incorrectly placed members. The delete command can be dangerous, however, because there is no "undo" command in MicasPlus.

The move commands allow members to be moved. Either a single member or a group of members can be moved. There are options to determine whether or not secondary members are moved along with the chosen member or members. A feature called "member associativity" allows members that are connected to the member to be moved to be changed accordingly. For example, if a story in a building is to be made taller, the floor may be moved and the columns will adjust accordingly.

The modify command allows the user to change the buoyancy status, class, element status, material, grade, or user attribute of an element or group of elements. This command changes attributes that have been set previously.

3.3.3 Material Reporting

ModelDraft provides a Linear Member Material Take-Off (MTO) option. The MTO provides methods to format, edit, generate, and place material reports for each linear member and arc element in an entire project, or only a selected group. ModelDraft provides a group of options that allow the user to create and edit the format by which the MTO is presented. Many options are available to customize the MTO. An MTO may be placed in the design file as a permanent text element. An MTO may be generated separately for area and/or volume elements in a similar fashion.

3.3.4 Drawing Extractions

The drawing extraction utility is used to create engineering drawings from a 3-D model. The is one of the most important features of ModelDraft since many of the other functions that ModelDraft performs are available in either MicasPlus Analysis or MicasPlus Design. ModelDraft provides several methods for choosing the views to be extracted, assuring that the desired view can be obtained.

Once the desired view is obtained, there are functions available for manipulating the views. As would be expected, there are options for revising, deleting, and listing view set. In addition to these, there is an option which updates the extracted views when a change is made to the model.

3.3.5 Drawing Production

Once a view is extracted, it can be annotated in order to produce construction drawings. This feature is implemented in a completely different environment since the procedure of creating construction drawings is very different from modeling a structure. This environment is called Drawing Composition Environment.

When creating a drawing, the first step is to recall an extracted view. Once the view is recalled, the user can use the Drawing Composition Environment to scale, move, rotate, and dimension the views. ModelDraft automatically places a drawing border, a number, scale, and a name. These options can be turned off, however.

3.4 MicasPlus Analysis

MicasPlus Analysis is a module in MicasPlus that performs both 2-D and 3-D analysis of structures. It does this by performing finite element analysis on a geometric model of the structure that is created in either MicasPlus Analysis or MicasPlus ModelDraft. Analysis capabilities include linear static analysis, limited non-linear static analysis, and dynamic analysis of structures.

MicasPlus Analysis uses a graphical interface as described previously. The graphical interface allows a structure to be drawn on the screen with a mouse rather than using numbers and letters to describe the structure. MicasPlus Analysis uses the graphical interface to display the model geometry, member orientation, member end releases, boundary conditions, and many other parameters of the model. This is not only convenient, but also a valuable tool for discovering errors. When an element is placed, the user can visually verify that it is being placed as desired.

MicasPlus Analysis has many of the same member placement and manipulation functions that are available in ModelDraft. This allows the user to create and edit models in MicasPlus Analysis rather than switching between MicasPlus Analysis and ModelDraft. Any changes made to the model in MicasPlus Analysis will be reflected in ModelDraft and MicasPlus Design.

In addition to the features mentioned above, MicasPlus Analysis has the following features: many load types, extensive postprocessing abilities, and the ability to interface with many other Intergraph Corporation products.

The basic process involved in analyzing a model involves: choosing a model type, placing members, loading the members, applying boundary conditions to the structure, and performing an analysis. Following is a brief description of these steps.

3.4.1 Model Types

MicasPlus analysis has six different model types available. The model type determines which degrees of freedom (d.o.f.) are active. The maximum number of degrees of freedom is six (three translational and three rotational). The following list briefly describes each model type.

Plane Stress - 2-D model with (X,Y) d.o.f. active

Thin Plate - 2-D model with (Z,RX,RY) d.o.f. active

Axisymmetric Solid - 2-D model with (X,Y) d.o.f active

Solid - 3-D model with (X,Y,Z) d.o.f. active

Thin Shell - 3-D model with (X,Y,Z,RX,RY,RZ) d.o.f. active

Plane Stress Special - 2-D model with (X,Y,RZ) d.o.f. active

While choosing Thin Shell for all models would work, it would greatly slow down the analysis process for certain models.

Within each model type there are two type of elements. They are line elements and finite elements. Line elements represent a beam or truss element with two nodes. Finite elements represent any of the finite elements in the MicasPlus Analysis finite element library.

3.4.2 Member Types

There are two types of members in MicasPlus Analysis. These are: analytic elements and physical members. An analytic element is the smallest unit used in MicasPlus. An analytic element is equal to a finite element in the analysis process. A physical member represents a single member in the real structure that is being modeled. When a physical member is placed, it is separated into one or more analytical elements by MicasPlus Analysis in order to perform the analysis. For example, a floor beam supporting two purlins would be modeled as one physical member but would be divided into three analytic elements by MicasPlus Analysis. However, MicasPlus Analysis knows that it is actually one physical member and adjusts connectivity accordingly. Also, if a physical member is moved, MicasPlus Analysis knows to move the entire member, not just one analytic section.

There are three different types of physical members in MicasPlus Analysis. These are: beam, column, and brace. To MicasPlus Analysis, there is no difference in a beam, column, or brace. However, the distinction between these types of members is important in the design stage. When placing a beam, column, or brace, the user is prompted for information regarding the location of the member, element properties, cardinal point, and the orientation of the member. Also, information can be provided for end releases, rigid end offsets, shear stiffness, and other properties.

There are many more analytic elements than there are physical elements. Available analytic elements include line element, surface element, and solid element. The line element type includes such elements as gap elements, hook elements, rod elements, and spring elements. These are elements that differ in their capacity to handle tension and compression. Surface and solid elements are elements chosen from the finite element library of MicasPlus analysis.

3.4.3 Loads

MicasPlus Analysis provides a wide selection of load types. In addition to the expected loading conditions such as nodal, concentrated, distributed, and wind loads, MicasPlus allows the following loads: body loads, inertia loads, edge loads (on a finite element), ice loads, initial condition loads (initial stress, strain, etc), surface loads, and thermal loads. Load cases with combinations of any number of structural loads, each multiplied by a scaler, must be created in order to perform the analysis.

There is a wide variety of commands available to place and manipulate loads. With these commands, the user may place and alter loads on the model. The loads are represented graphically on the screen as either lines or arrows. It is possible to determine the direction and relative magnitude of a load by looking at the screen, but further effort is required to determine the exact magnitude of the load.

3.4.4 Boundary Conditions

Placing boundary conditions is simple in MicasPlus Analysis. There are two options in the boundary conditions menu. These options allow the user to constrain the end of a member in any of the applicable degrees of freedom or to specify an initial displacement for the member end.

Related to the topic of boundary conditions is the subject of end releases. The end releases of a member allow certain degrees of freedom to be released at each end of the member in order to simulate a truss element or a pinned connection, etc. When a degree of freedom is released, no stiffness between members exist and no forces can be transmitted for that particular degree of freedom.

3.4.5 Analysis Process

The analysis process consist of toggling options on or off in order to customize the analysis. Options that can be controlled during this step range from choosing the analysis type (such as static, dynamic, etc) to choosing what output is desired. Once all options are set as desired, the analysis can begin. An analysis cannot be interrupted once it begins; therefore, it is important that all options be set as desired before beginning.

3.4.6 Postprocessing Capabilities

MicasPlus analysis has extensive postprocessing capabilities including the ability to generate shear and moment diagrams, draw deformed shapes of the structure under loading, and label the structure with analysis results. The postprocessing capabilities of MicasPlus Analysis are built around a system called the Post-Data Management system (PDM). Vectors and matrices containing data concerning the results of an analysis are stored within the PDM.

One of the most important postprocessing features of MicasPlus Analysis is the ability to create shear, moment, axial load, torsion, and loading diagrams for each member. This is helpful in checking a structure to determine if it was modeled correctly. For example, by checking the moment diagram it can be determined if the member end conditions are correct.

Another postprocessing capability of MicasPlus Analysis is the ability to generate the deformed shape of the structure under loading. A deformed shape can be generated for any load case or load combination. It can be created in any view of the structure. A deformed shape plot is another tool that is useful in determining if the structure was modeled correctly.

MicasPlus Analysis can place labels on the model of various analysis results including support reactions, member end reactions, nodal deflections, and many other quantities. The labels are color coded to indicate their magnitude. For example, all support reactions under 300 kips may be displayed in yellow, reactions from 301 kips to 500 kips in blue, reactions from 501 kips to 700 kips in red, etc. This too, is helpful in checking the model for correctness.

3.5 MicasPlus Design

MicasPlus Design is a module that uses the results of MicasPlus Analysis to design either a single member, marked group, or an entire structure. MicasPlus Design is divided into MicasPlus Steel Design and MicasPlus Concrete Design. The following sections will discuss MicasPlus Concrete Design only, although many of the concepts are similar for MicasPlus Steel Design.

MicasPlus Concrete Design has many features. Some of the major features include: ability to design members based on several different codes, alphanumeric or graphic interface, single or multiple member design, design tables, torsion design, control of printed output, material takeoff, and many graphics output capabilities. MicasPlus design also has the capability to interact with MicasPlus Analysis. The member sizes and section properties that are obtained during design can be sent to the structural database and the analysis results can be updated using these new member properties. This provides an interactive analysis and design environment.

3.5.1 General Information

MicasPlus Concrete Design has features that are very useful during the design process. One feature is the concept of the physical member. The best way to illustrate the concept of the physical member is with an example. Consider a floor beam that has several other beams framing into it. The beam will be divided into several analytic elements for the purpose of analysis. However, the entire beam is considered to be one physical member for design purposes, and MicasPlus Design will choose one cross section that satisfies the requirements for all of the analytic elements because it realizes that one member will be used in the actual construction of the structure.

Another feature that assures reasonable design results is the concept of mark groups. As described previously, a mark group is a group of members that have the same design. There are two types of mark groups. The first type of mark group assures that all member in a mark group have the same dimensions. The second type requires the dimensions and the bar selection be the same for all members in a mark group. The concept of mark groups is valuable to assure reasonable member selections.

3.5.2 Design Parameters

There are five categories of design parameters that can be altered in MicasPlus Concrete Design. They are:

- Strength and deflection check flags
- Bracing parameters
- Member design parameters
- Process control options
- Table assignments

The strength and deflection check flags are a group of design items that may be turned on or off. The group includes strong axis bending check, weak axis bending check, shear check, torsion check, and deflection check. If an item is turned off, MicasPlus Concrete Design will not check for that particular code requirement when designing the member.

Bracing parameters allow the specification of bracing conditions for a member or group of members. Bracing conditions include unbraced length and a check flag for bracing against sidesway about the major and minor axes.

The design parameters are divided into two groups. These groups are code independent parameters and code dependent parameters. Code independent parameters include such parameters as design type, member orientation, design end offsets, effective length factor calculation basis, and others. Code Dependent Parameters include parameters such as method for determining Beta and Cm, moment magnifiers, and crack width calculations.

Member processing control options determine several functions. First, a member can be toggled to be active or inactive, thus determining whether or not it will be processed. Second, two features including analysis property update and independent processing can be toggled on or off. Analysis property updates determines if the analysis will be updated based on design results. Independent processing determines if independent results will be calculated in addition to mark group results. Finally, the basis for a material take off can be specified.

Table assignment options allow control over the selection of which construction materials, such as reinforcing bars, are available for design purposes. The table assignment options also allow the user to specify the order in which the parts are tried. For example, the user can specify that #3, #4, and #5 bars are available for the design of a particular beam.

3.5.3 Member Parts Library

The member parts library in MicasPlus Concrete Design is a library of available bar sizes and patterns that can be selected/used for design. The member parts library consists of the concrete bar library table, the column bar pattern table, and the shear/torsion stirrup table. Many choices for bar patterns and stirrup types are available in this library. In addition to the member parts library, the material properties and shape of the elements must be specified before design of a beam or column is possible. Deflection limits may also be specified if desired.

3.5.4 Effective Length Factor

There are three available methods in MicasPlus Concrete Design for determining the effective length factor including direct user entry, selection from a user editable table, and automatic calculation by the program. The direct user entry method is used for unusual cases or if the moment magnification is being calculated by the P-Delta method or from a geometrically non-linear analysis. The user editable table lists different combinations of end release situations and the corresponding k. The calculation by program method uses one of two equations (depending on whether the member is braced or unbraced) to determine the value of the k factor for design.

3.5.5 Reports and Post Processing

MicasPlus Design has simple commands for printing or reviewing information that is generated during the design process. These commands include Print Input, Print Results, and Print Material Takeoff. The Print Input command prints member data, design tables, design groups, and load data. This command has two options: Design Definition Report Selection and Print Design Definition. Design Definition Report allows the user to specify the desired output from choices which include material tables, deflection limit tables, load case type, etc. The Print Design Definition tutorial allows the user to select the output destination. The Print Results command is responsible for printing the results of a design run. It works much the same way as Print Input, allowing the user to specify the desired output information and location. The Material Takeoff command lists the volume of concrete, weight of reinforcing bars, and area of forms required to build the structure.

Generation of graphic results of a design is possible in MicasPlus Concrete Design. However, this can be done more thoroughly using MicasPlus ModelDraft. Because the process is similar in ModelDraft, Analysis, and Concrete Design, it will not be discussed here.

3.6 STAAD III

STAAD III is one of three modules that form ISDS structural engineering software. However, these three modules are commonly referred to as STAAD III rather than ISDS. STAAD III will be used in this thesis to refer to all three modules which are STAAD III, STAADPL, and STAAD - DRAFT. STAAD III is an integrated structural analysis and design program that performs structural analysis, design, graphics, and drafting.

3.6.1 STAAD III Module

STAAD III is the module that performs structural analysis and design. It is capable of performing static analysis, dynamic analysis, and P-delta analysis of structures. Structures can include frame, plate, and shell elements. STAAD III perform steel, concrete, and timber design.

The STAAD III interface is primarily alphanumeric. Models are entered by using a line editor that is contained in STAAD III. The user must create an input file to describe the structure to STAAD III. All information about the structure is entered in this file including member data, member location, model type, location of nodes, load cases and combinations, units, etc.

A wide range of structures may be analyzed using one of the four model types available in STAAD III. The four model type include PLANE, SPACE, TRUSS 2D/3D, and FLOOR structures. A PLANE structure has all members, loads, and reactions restricted to one plane. A SPACE structure has all degrees of freedom available and allows three dimensional structures with loads applied in any plane. A TRUSS structure has only axial forces present and may be either two dimensional or three dimensional. A FLOOR structure has no horizontal loads applied and may be either two dimensional or three dimensional.

Many different types of loads are available in STAAD III. Nodal loads may be applied at any free joint in the structure. Three types of member loads may be applied including uniformly distributed loads, concentrated loads, and linearly varying loads. An "area load" may be applied to a plane that represents, for example, a floor in a structure. STAAD III will determine the tributary area for each member under the area load and will determine the appropriate load to be applied to the member. Other loads may be applied including fixed end member load, prestress member load, temperature load, support displacements, and moving loads. In addition, STAAD III provides a UBC seismic load generator as well as a wind load generator.

STAAD III performs finite element analysis. Available elements include plane stress, plane strain, plate bending, out of plane shear, and flat shell elements.

Either three or four noded elements may be used. A wide range of loading conditions is available for finite elements including joint loads, concentrated loads, uniform pressure loads, partial uniform pressure loads, linearly varying pressure loads, and temperature loads. Output consists of membrane forces, bending moments, principle stresses, the maximum shear stress, and the orientation of the principle plane.

3.6.2 STAAD III Concrete Design

STAAD III perform basic concrete design. It is capable of designing beams, columns, and slabs/walls. Beams must be either rectangular, trapezoidal, T-shaped, or prismatic. Columns must be either rectangular, square, or circular. It is assumed that an equal number of reinforcing bars will be placed on each face of the column. Stirrups are always U-shaped. Slabs and walls are modeled as finite elements. The size and shape of a member must be specified by the user before the design takes place. STAAD III will choose reinforcement for the specified section.

Slenderness effects are calculated by either a P-delta method or the moment magnifier method. The P-delta method is an analysis type which accounts for the effect of axial load on the deflected shape of the member. The moment magnifier method allows the user to specify a factor by which moments will be magnified for the purposes of design.

Design output consists of bar size and number for positive and negative moment regions, plus bar sizes and spacing of bars for shear. STAAD III will specify the location of this reinforcement as well as whether or not anchorage is necessary. Also, column interaction diagrams will be produced if desired.

3.6.3 STAADPL

STAADPL is a module that provides the graphical capabilities for STAAD III. STAADPL performs three important functions. First, it allows visual verification of the model by creating a plot of the structure. This is important since the input file can be quite lengthy and difficult to debug. Second, it provides graphical input generation capabilities. This is an alternate method for creating an input file. It has the ability to define structure geometry, element properties, boundary conditions, etc. Third, STAADPL provides graphical post-processing functions. This allows STAAD III to label the structure with output results.

3.6.4 STAAD - DRAFT

STAAD - DRAFT is a drafting program which provides basic drafting and editing tools. STAAD - DRAFT may be used with STAAD III or may be used as a stand alone drafting program. STAAD - DRAFT provides basic drafting tools such as drawing and editing lines, arc, circles, etc. It can also used to dimension drawings. STAAD - DRAFT works interactively with the other modules in STAAD III.

Chapter 4

Description of Test Structures

4.1 Overview of Test Structures

Four structures were analyzed and designed in both MicasPlus and STAAD III. Included in the test structures are a continuous beam having a rectangular cross-section, a two story multibay frame, a seven story frame, and a fourteen story frame. The material properties were as follows: Concrete compressive strength, fc' = 4000 psi, modulus of elasticity, E=3,605,000 psi, and Poisson's ratio, v=0.3. Figure 1 shows the global coordinate system that was used. The following sections describe the structures and the applied loading.

The procedure used to determine initial member sizes for each structure was as follows. First, each structure was designed using MicasPlus. The section sizes obtained from the design were then used for analyzing the structure in both MicasPlus and STAAD III. MicasPlus was used to obtain member sizes instead of STAAD III because STAAD III does not choose cross sections; it only selects reinforcement for a given cross section. MicasPlus sizes the cross section as well as the reinforcement for each member.

For the multistory structures, the MicasPlus mark group option was used to choose initial member sizes for use in analysis. For design purposes, the mark groups were removed so that MicasPlus was able to choose reinforcement for each member independently. This allowed a fair comparison between MicasPlus and STAAD III design results.

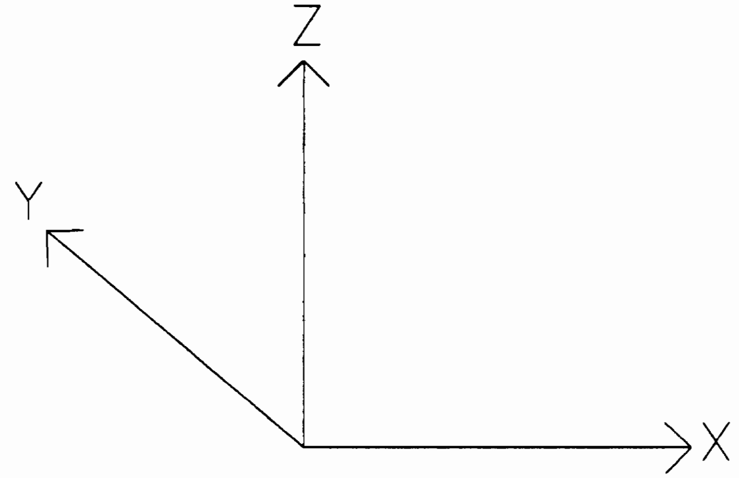


Figure 1: Coordinate System

4.2 Continuous Beam

The first structure considered was a two dimensional, four span, continuous beam. The following sections describe the geometry, boundary conditions, and loading conditions of the structure.

4.2.1 Cross Section

The beam was taken from a typical interior bay in a building. It has a 18 x 26 inch rectangular cross section as shown in Figure 2.

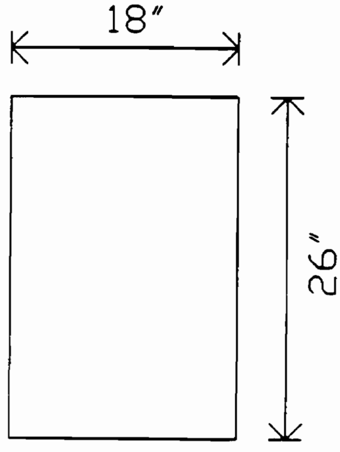


Figure 2: Cross Section of Continuous Beam

4.2.2 Boundary Conditions

The four-span concrete beam is supported in the following manner. The beam is pinned on each exterior end. The interior supports of the beam are roller supports, allowing no vertical translation. In order to model these support conditions, certain degrees of freedom are fixed. For the pinned joints, A and E (see Figure 3), translation in both directions is fixed (X,Z). In addition, torsional rotation about the axis of the member is prevented (RX). At joints B, C, and D, translation in the Z direction is fixed, as well as torsional rotation about the axis of the member.

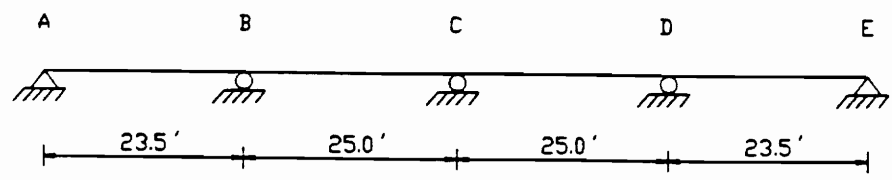


Figure 3: Continuous Beam Boundary Conditions

4.2.3 Loading

The loading on the beam consists of a uniform dead load and a uniform live load which are applied to the entire length of the beam. The dead load has a magnitude of 2 k/ft plus the self weight of the beam and the live has a magnitude of 3 k/ft. In addition, a concentrated live load of 20 kips is placed at the midpoint of each span as shown in Figure 4. The only load combination considered is:

$$1.4D + 1.7L$$

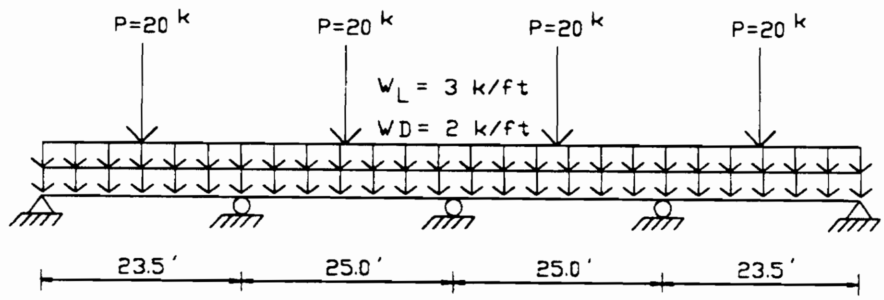


Figure 4: Loading on Continuous Beam

4.3 Two Story Frame

The second structure considered was a three dimensional two story frame shown in Figure 5. All beam cross sections are rectangular and all columns are square. For the two story, seven story, and fourteen story structures, all moment magnifiers were set equal to 1 and all columns were therefore designed as short columns. Also, all columns had tied reinforcement rather than spirals.

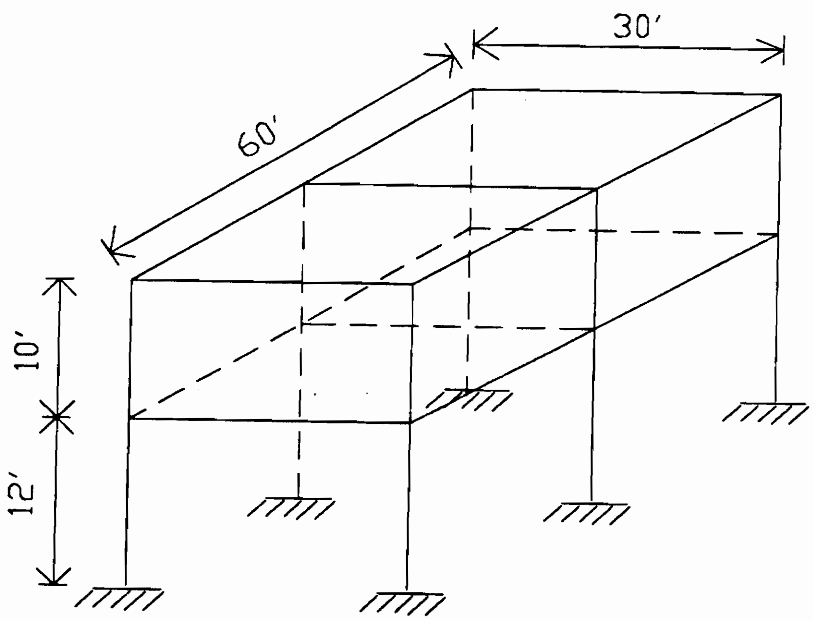


Figure 5: Two Story Multibay Frame

4.3.1 Geometry

The building is two stories high. The first story is 12 feet and the second story is 10 feet. The building consist of two 30 foot by 30 foot bays. Therefore, the building is 30 feet wide by 60 feet long. A plan view of the structure is show in Figure 6. The front view and side views are shown in Figures 7 and 8. The member numbers are shown in Figure 9.

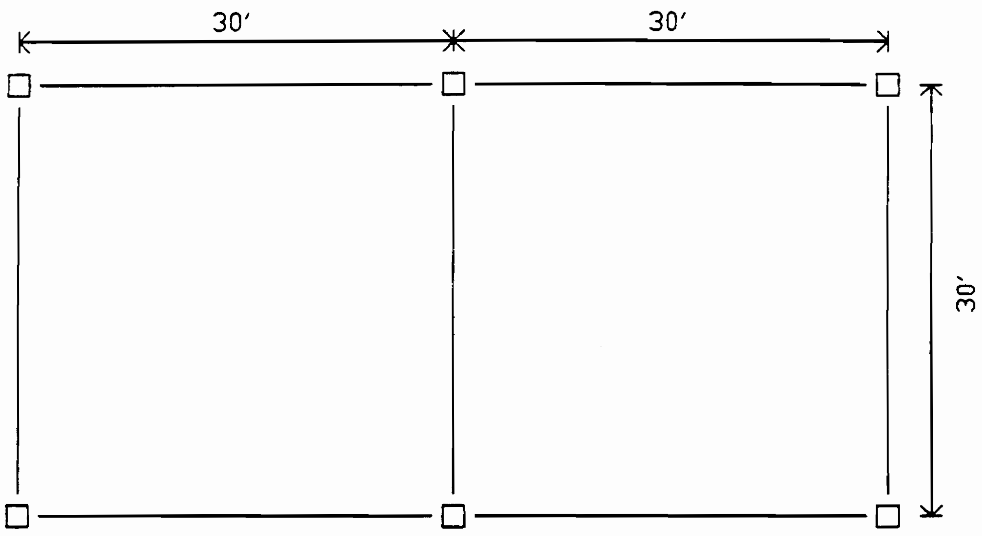


Figure 6: Plan View of Two Story Frame

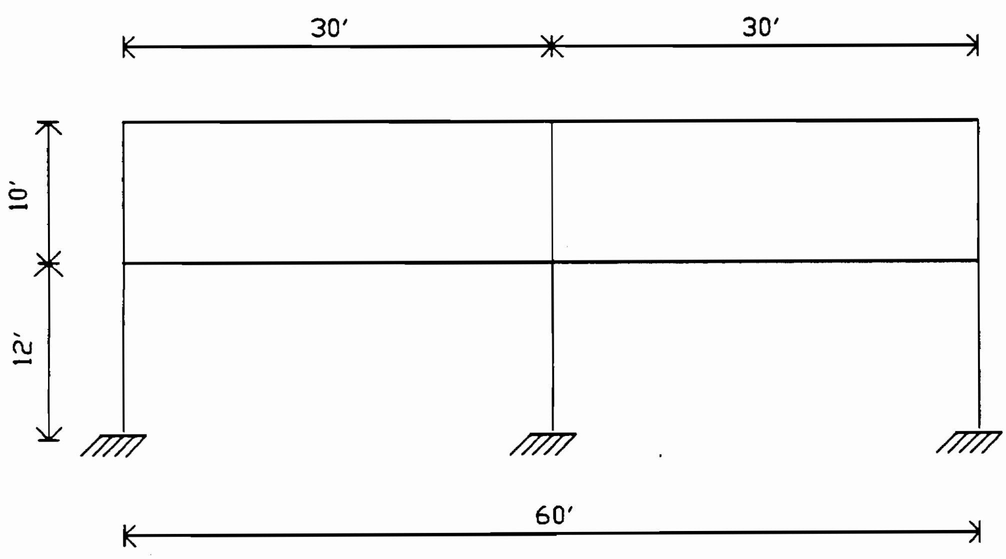


Figure 7: Front View of Two Story Frame

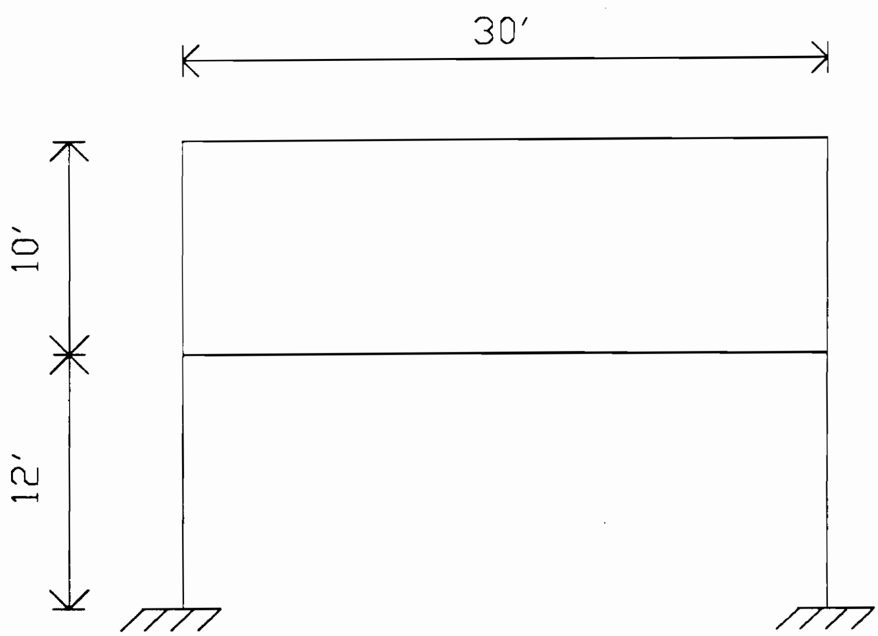


Figure 8: Side View of Two Story Frame

4.3.2 Member Cross-Sections and Sizes

As stated above, the member cross sections were obtain from MicasPlus Design. Using typical area loads of 50 psf live load and 120 psf dead load applied to the floor and roof, resultant forces on the spandrel beams were determined. An additional dead load of 5.5 kips was applied to member SB1 at each of the two concentrated load locations. Each beam cross section was chosen based on these forces. The beams sizes chosen by MicasPlus are shown in Figure 10. An

18 x 18 inch square cross section was chosen for all columns.

Mark groups were used to determine the initial size of the members. Beam groups SB1, SB2, and B2 (see Figure 9) each formed a mark group. All columns formed another mark group.

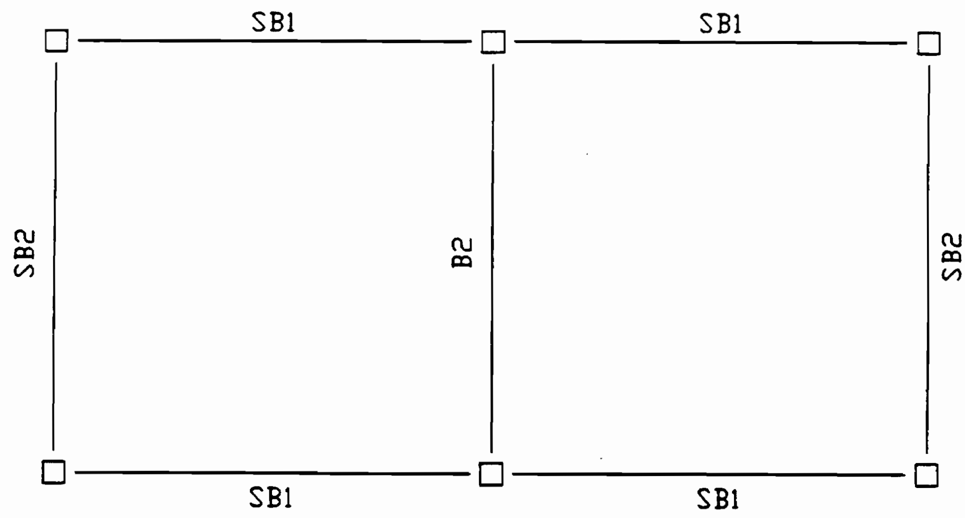


Figure 9: Nomenclature for Floor and Roof Beams

4.3.3 Support Conditions and Member Releases

All supports are fixed. All joints are rigid. There are no member end releases in the entire structure. The beams do not resist torsion. All beams are unbraced. It is a moment resisting frame.

4.3.4 Loading

The loads acting on the structure are shown in Figure 11. The concentrated loads on beam SB1 are support reactions from beams that were not considered in the analysis or design because only the primary frame was considered. A wind load of 25 psf was applied to the left wall as well as the front wall. The wind loads were discretized and applied at joints using an appropriate tributary area. The following load combinations were considered:

$$1.4D + 1.7L$$

$$0.75(1.4D + 1.7L + 1.7W(\text{left}))$$

$$0.75(1.4D + 1.7L + 1.7W(\text{front}))$$

$$0.9D + 1.3W(\text{front})$$

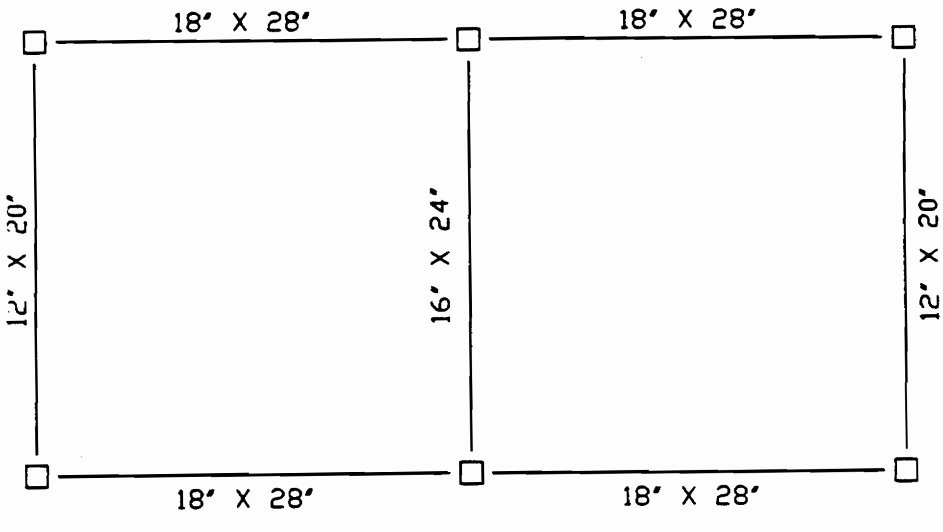


Figure 10: Beam Sizes Two Story Frame

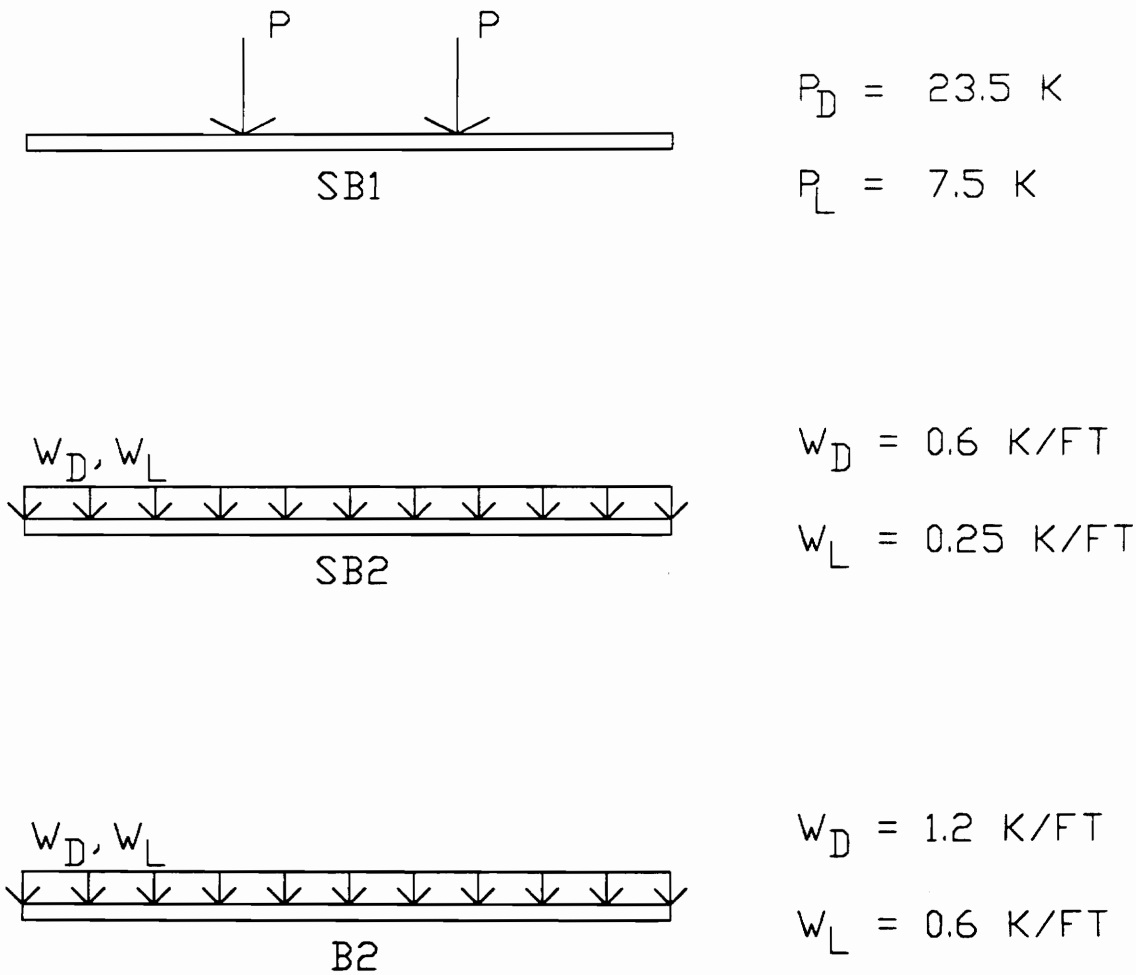


Figure 11: Loading on Two Story Frame

4.4 Seven Story Concrete Frame

The third test structure is a seven story, six bay building. It is similar to the two story test structure except for the size and number of stories. Again, all beams are rectangular and all columns are square.

4.4.1 Geometry of the Building

The building has a rectangular plan and has six bays. The building is 75 feet long by 40 feet wide. Each story is 10 feet high except for the first story which is 16 feet high. Figure 12 is a plot of the structure which was generated using STAADPL.

4.4.2 Members

In order to provide a realistic design, the mark group option on MicasPlus was used for the design of this structure. The mark group option assures that all members in a given mark group will have a common design. There were four mark groups for the columns and four mark groups for the beams.

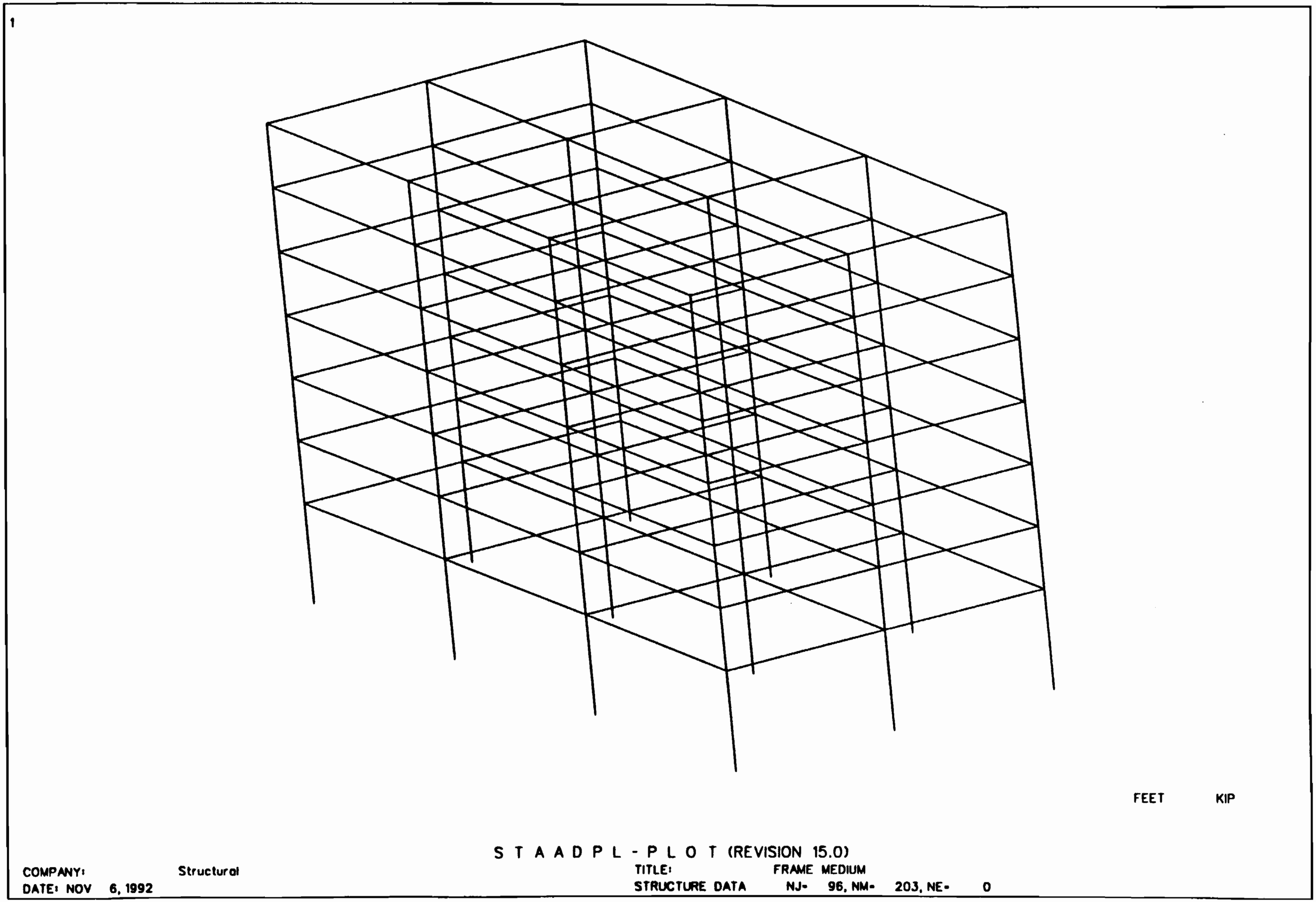


Figure 12: STAADPL Plot of Seven Story Building

The column mark groups were subdivided in the following manner. The exterior and interior columns were placed in separate mark groups. Also, all the columns on the first three floors were placed in different mark groups than the columns on the top four floors. This arrangement resulted in four column mark groups: bottom level interior, bottom level exterior, top level interior, and top level exterior.

There were four mark groups for the beams. These are shown in Figure 13.

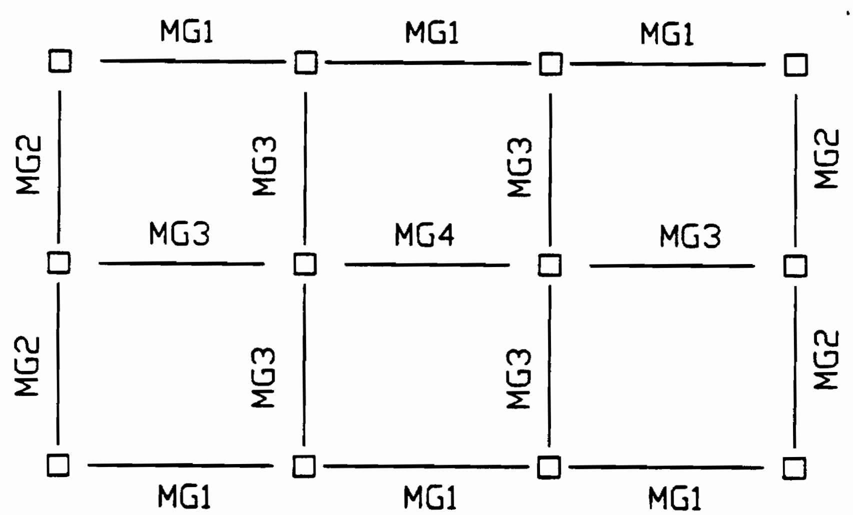


Figure 13: Mark Groups for the Beams of the Seven Story Building

The following member sizes were chosen for the seven story structure: all columns were 16 x 16 inches, except for the interior columns on the first three floors which were 26 x 26 inches. The beam sizes are shown in Figure 14.

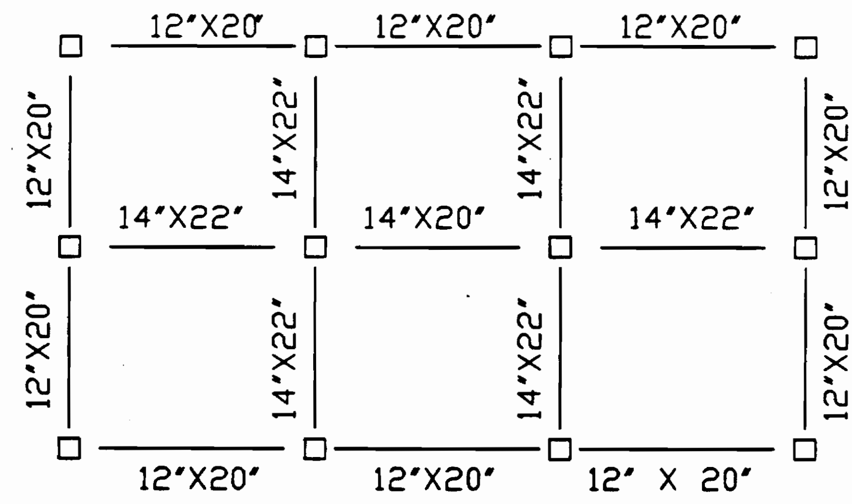


Figure 14: Beam Sizes for All Floors in Seven Story Building

4.4.3 Support Conditions and Member Releases

All supports are fixed. There are no member end releases. All beams and columns are unbraced except at the nodes. It is a moment resisting frame.

4.4.4 Loading

The loading on the members was determined by placing a 100 psf live load and a 115 psf dead load (not including self weight of the members) on each floor and the roof of the structure. For each member, an equivalent uniformly distributed load was determined and applied to the member. The uniformly distributed loads ranged in magnitude from 0.75 kips/foot to 2 kips/foot, depending on the tributary area of the member. Also, a wind load caused by a 100 mph wind was applied to the front and left side of the building in accordance with the UBC-88 (UBC, 1988) wind loading provisions. The following load combinations were considered:

$$1.4D + 1.7L$$

$$0.75(1.4D + 1.7L + 1.7W(\text{left}))$$

$$0.75(1.4D + 1.7L + 1.7W(\text{front}))$$

$$0.9D + 1.3W(\text{front})$$

4.5 Fourteen Story Frame

The final structure considered in the verification study was a fourteen story concrete frame. It was similar to the seven story frame except for its geometry. Therefore, all supports are fixed and there are no member end releases. The loading on all beams consists of uniformly distributed dead and live loads.

4.5.1 Geometry of the Building

The fourteen story frame is square in plan (see Figure 15). It has nine 26 X 26 foot bays, arranged with three bays on each side. Each story is 10 feet high.

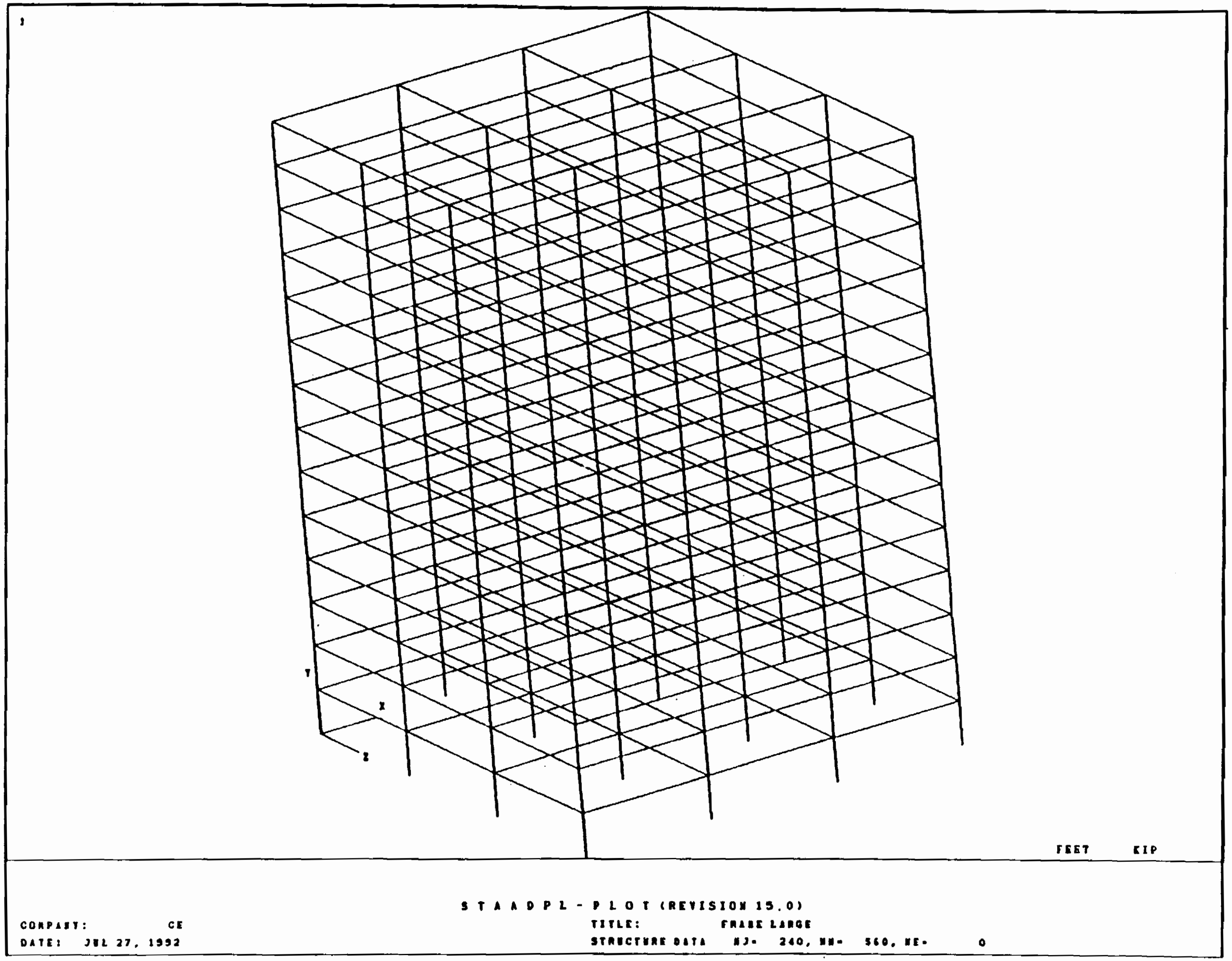


Figure 15. STAADPL Plot of Fourteen Story Frame

4.5.2 Members

The exterior spandrel beams are 14 X 24 inches (See Figure 16). The interior beams are 18 X 36 inch rectangular beams.

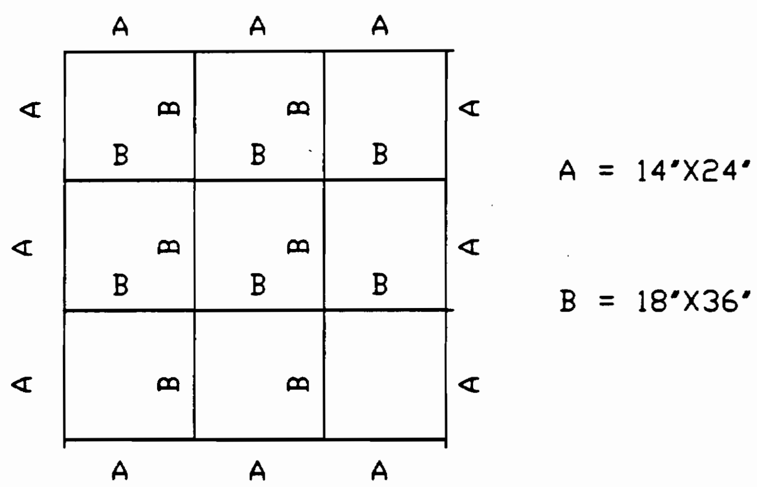


Figure 16. Floor Beams in Fourteen Story Frame

Many different column sizes were used. The exterior columns on floors 1 - 4 are 52 inches square. The exterior columns on floors 5 - 8 are 40 inches square. The exterior columns on floors 9 - 12 are 28 inches square. The exterior columns on floors 13 - 14 are 16 inches square. The interior columns on floors 1 - 4 are 32 inches square. The interior columns on floors 5 - 8 are 24 inches square. The interior columns on floors 9 - 14 are 20 inches square.

There are eight column mark groups. The fourteen levels are divided into four groups which are the first four levels, the second four, the third four, and the last three levels. Within each level, the columns are divided into two mark groups: the exterior columns and the interior columns. This makes a total of eight mark groups.

There are two beam mark groups. The exterior beams form one group and the interior beams form another group.

4.5.3 Loading

The loading was determined by applying a 130 psf live load and a 110 psf dead load to each floor and the roof. The loads were applied to the beams as an equivalent uniform live load. In addition to this, a body load was applied to the entire structure. Also, a wind load caused by a 100 mph wind was applied to the front of the building in accordance with the UBC-88 (UBC, 1988) wind provisions. The following load combinations were considered:

$$1.4D + 1.7L$$

$$0.75(1.4D + 1.7L + 1.7W(\text{left}))$$

$$0.75(1.4D + 1.7L + 1.7W(\text{front}))$$

$$0.9D + 1.3W(\text{front})$$

Chapter 5

Results

5.1 General

In this chapter a comparison of the results obtained from MicasPlus and STAAD III for the analysis and design of the structures described in Chapter 4 is presented. For analysis, the values compared include support reactions, nodal displacements, and member end forces. The quantities considered in the comparison of design results for beams include the area of steel chosen for the top longitudinal reinforcing steel located at the left end of the beam as well as the top longitudinal reinforcing steel located at the right end of the beam. The positive moment resisting reinforcing steel located at mid-span is compared as well. Also, the recommended spacing of #4 stirrups at the left end of the beam and at the right end of the beam is compared. For the design of columns, the area of longitudinal reinforcement chosen by MicasPlus was compared to the area chosen by hand calculations. The STAAD III column design results were not used because they are somewhat questionable. Discussions with the technical support department at Research Engineers, Inc. confirmed this.

The results presented in this chapter are for certain key locations. The nodes and elements presented in the tables were chosen to represent a large variation of loading conditions and locations throughout the structure. Whenever possible, nodes and elements with relatively large displacements or reactions were used in order to decrease the effect of round-off errors. The location of the nodes and elements in the test structures is shown in Appendix A.

The percentage difference is not shown on the tables for some results that are small in magnitude. This is because differences in results for small values is not important from a practical standpoint since these will not affect the design of the structure. Large percentage difference in these small values is due to round-off errors and due to the fact that the two programs run on different hardware platforms.

5.2 Continuous Beam

Tables 1 through 3 show a comparison of the analysis results generated by MicasPlus and STAAD III for the continuous beam model. Results for all nodes and elements were compared for three load cases and one load combination. Nodes 1 and 5 are exterior nodes supporting the beam (see Appendix A, Fig. A-1). Nodes 2 and 4 are interior nodes. Node 3 is the center support. Elements 1 and 4 are exterior beams, while elements 2 and 3 are the interior beams.

Table 1 shows support reactions generated by MicasPlus and STAAD III. As would be expected, both programs indicate that the response of the structure is symmetric. The largest percentage difference in the support reactions computed by MicasPlus and STAAD III is at nodes 1 and 5 for load case 3. This is because the magnitude of the reaction is small at these nodes, so a difference of only .02 kips results in the highest percentage difference. Practically speaking, however, there is no difference between the support reactions produced by MicasPlus and STAAD III for the continuous beam.

Table 2 shows nodal rotations computed by MicasPlus and STAAD III for the continuous beam. Again, a symmetric response was obtained. The largest percentage difference between the results from the two programs is less than 1.5%. This difference is probably due to round-off error in the two programs as

Table 1. Support reactions for continuous beam

Load Case 1 (uniform dead load)

Node no.	MicasPlus Fy (kips)	Staad III Fy (kips)	Percentage Difference
1	22.72	22.74	0.09%
2	68.24	68.17	0.10%
3	59.37	59.46	0.15%
4	68.24	68.17	0.10%
5	22.72	22.74	0.09%

Load Case 2 (uniform live load)

Node no.	MicasPlus Fy (kips)	Staad III Fy (kips)	Percentage Difference
1	27.40	27.43	0.11%
2	82.30	82.21	0.11%
3	71.61	71.71	0.14%
4	82.30	82.21	0.11%
5	27.40	27.43	0.11%

Load Case 3 (concentrated live load)

Node no.	MicasPlus Fy (kips)	Staad III Fy (kips)	Percentage Difference
1	6.73	6.75	0.30%
2	24.12	24.08	0.17%
3	18.29	18.34	0.27%
4	24.12	24.08	0.17%
5	6.73	6.75	0.30%

Load Comb 1 (1.4D + 1.7L)

Node no.	MicasPlus Fy (kips)	Staad III Fy (kips)	Percentage Difference
1	89.83	89.94	0.12%
2	276.45	276.15	0.11%
3	235.95	236.33	0.16%
4	276.45	276.15	0.11%
5	89.83	89.94	0.12%

Table 2. Nodal displacements for continuous beam

Load Case 1 (uniform dead load)

Node	MicasPlus	Staad III	Percentage Difference
no.	Rz (rad)	Rz (rad)	
1	-0.00113	-0.00114	0.88%
2	0.00022	0.00022	0.00%
3	0	0	0.00%
4	-0.00022	-0.00022	0.00%
5	0.00113	0.00114	0.88%

Load Case 2 (uniform live load)

Node	MicasPlus	Staad III	Percentage Difference
no.	Rz (rad)	Rz (rad)	
1	-0.00136	-0.00138	1.47%
2	0.00027	0.00026	-
3	0	0	0.00%
4	-0.00027	-0.00026	-
5	0.00136	0.00138	1.47%

Load Case 3 (concentrated live load)

Node	MicasPlus	Staad III	Percentage Difference
no.	Rz (rad)	Rz (rad)	
1	-0.00059	-0.0006	1.69%
2	0	0	0.00%
3	0	0	0.00%
4	0	0	0.00%
5	0.00059	0.0006	1.69%

Load Comb1 (1.4D + 1.7L)

Node	MicasPlus	Staad III	Percentage Difference
no.	Rz (rad)	Rz (rad)	
1	-0.0049	-0.00496	1.22%
2	0.00099	0.00098	1.01%
3	0	0	0.00%
4	-0.00099	-0.00098	1.01%
5	0.0049	0.00496	1.22%

are many of the other differences. Table 3 shows element end forces for each element in the continuous beam. There is no significant difference between the values produced by MicasPlus and those produced by STAAD III.

Table 4 shows the design results produced by MicasPlus and STAAD III for the continuous beam. Steel areas chosen by the two programs were compared for longitudinal and shear reinforcing. The difference in the values produced by the two programs for longitudinal steel were consistently less than 2.31%, except for the case of the top longitudinal reinforcing at the left end of element 3. In this case, there was a difference of 7.83%, which is still not very significant. There is no combination of reinforcing bars that will provide an area of steel between 5.53 in^2 and 6.00 in^2 which are the areas computed by STAAD III and MicasPlus, respectively. If MicasPlus determined a required steel area of 5.54 in^2, for example, then it would increment to 6.00 in^2 of steel. Thus, from a practical viewpoint the results given by MicasPlus and STAAD III are closer than it may first appear.

The stirrup spacing determined by MicasPlus agrees closely with the spacing requirements determined by STAAD III. In every case, STAAD III is less conservative than MicasPlus by 0.1 inch.

In order to correctly interpret the design results, it is important to understand the

Table 3. Element end forces for continuous beam

Load Case 1 (uniform dead load)

El.	Nd.	MicasPlus		Staad III		% Difference	
no.	no.	Fy (kips)	Mz (k-ft)	Fy (kips)	Mz (k-ft)	Fy	Mz
1	1	22.72	0.00	22.74	0.00	0.09%	0.00%
2	2	32.50	153.00	32.46	152.39	0.12%	0.40%
3	3	29.69	117.83	29.73	118.31	0.13%	0.41%
4	4	35.74	153.01	35.71	152.39	0.08%	0.41%

Load Case 2 (uniform live load)

El.	Nd.	MicasPlus		Staad III		% Difference	
no.	no.	Fy (kips)	Mz (k-ft)	Fy (kips)	Mz (k-ft)	Fy	Mz
1	1	27.40	0.00	27.43	0.00	0.11%	0.00%
2	2	39.20	184.53	39.14	183.79	0.15%	0.40%
3	3	35.80	142.11	35.86	142.68	0.17%	0.40%
4	4	43.10	184.53	43.07	183.79	0.07%	0.40%

Load Case 3 (concentrated live load)

El.	Nd.	MicasPlus		Staad III		% Difference	
no.	no.	Fy (kips)	Mz (k-ft)	Fy (kips)	Mz (k-ft)	Fy	Mz
1	1	6.73	0.00	6.75	0.00	0.30%	0.00%
2	2	10.86	76.75	10.83	76.43	0.28%	0.42%
3	3	9.14	55.37	9.17	55.64	0.33%	0.49%
4	4	13.27	76.75	13.25	76.43	0.15%	0.42%

Load Comb 1 (1.4D + 1.7L)

El.	Nd.	MicasPlus		Staad III		% Difference	
no.	no.	Fy (kips)	Mz (k-ft)	Fy (kips)	Mz (k-ft)	Fy	Mz
1	1	89.83	0.00	89.94	0.00	0.12%	0.00%
2	2	130.59	658.39	130.40	655.71	0.15%	0.41%
3	3	117.97	500.69	118.16	502.77	0.16%	0.42%
4	4	145.86	658.39	145.75	655.71	0.08%	0.41%

Table 4. Design results for continuous beam

a. Beam longitudinal reinforcing - top

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	in^2	in^2	in^2	in^2		
1	-	7.80	-	7.62	-	-2.31%
2	7.80	6.00	7.62	5.53	-2.31%	-7.83%
3	6.00	7.80	5.53	7.62	-7.83%	-2.31%
4	7.80	-	7.62	-	-2.31%	-

b. Beam longitudinal reinforcing- bottom

Element Number	MicasPlus	STAAD III	% Differ.
	Midspan	Midspan	
	in^2	in^2	
1	5.00	5.00	0.00%
2	3.08	3.08	0.00%
3	3.08	3.08	0.00%
4	5.00	5.00	0.00%

c. Stirrup spacing

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	inches	inches	inches	inches		
1	11.7	5.6	11.8	5.7	0.85%	1.79%
2	6.9	8.5	7	8.6	1.45%	1.18%
3	8.5	6.9	8.6	7	1.18%	1.45%
4	5.6	11.7	5.7	11.8	1.79%	0.85%

process by which MicasPlus and STAAD III choose reinforcing for a member. There is a difference in how MicasPlus and STAAD III or hand calculations choose reinforcing. In MicasPlus, it is necessary for the user to specify what reinforcing bars can be used for the design of each member. Therefore, if the user chooses #4, #5, #6, #7, and #8 bars, MicasPlus will first check to see if there is a combination of #4 bars that would satisfy the requirements. If not, it will then try #5 bars, and so on. Once a satisfactory combination is found, MicasPlus ends design for that member. This causes conservative results because MicasPlus does not check to see if fewer, larger bars would provide a steel area closer to the desired value.

When designing columns, MicasPlus uses a similar approach. The user must specify whether bars or patterns have precedence. If patterns have precedence, then the program will proceed to the next column reinforcing pattern before proceeding to the next bar size. If the bars have precedence, then the program will proceed to the next bar size before proceeding to the next reinforcing pattern. Again, this can potentially yield conservative results because MicasPlus terminates design of a member when it finds the first acceptable combination of bars rather than checking for closer combinations.

STAAD III does not allow the user to specify either the bar preferences or the pattern. An advantage to this is that STAAD III chooses the lightest combination that is adequate. The disadvantage is that the user has no control on how STAAD III will design the member.

5.3 Two Story Frame

The comparison of the analysis results produced by MicasPlus and STAAD III is presented in Tables 5, 6, and 7. Table 5 shows support reaction results at nodes 1, 5, 7, and 11. Nodes 1, 7, and 11 are located at the bottom of a first story corner columns (see Appendix A, Fig. A-2). Node 5 is located at the bottom of a first story exterior column. Nodal displacements for nodes 4, 8, 13, and 18 are shown in Table 6. Nodes 4 and 8 are located at the first level. Nodes 13 and 16 are located at the roof level. Element end forces for elements 1, 10, 21, and 24 are shown in Table 7. Element 1 is a first story corner column. Element 10 is a second story edge column. Elements 21 and 24 are exterior beams located on the roof level. These nodes and elements were chosen since they have large displacements and forces. They were also chosen because they are located at a wide range of locations throughout the building.

There is no significant difference in values of the support reactions, nodal displacements, and element end forces computed by MicasPlus and STAAD III for the two story frame. Support reactions agree within a maximum of 0.46%. Nodal

Table 5. Support reactions for two story frame

Load Combination 1

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	10.87	8.43	196.30	10.88	8.41	196.35	0.09%	0.24%	0.03%
5	0.00	15.36	443.59	0.00	15.35	443.48	0.00%	0.07%	0.02%
7	-10.87	8.43	196.30	-10.88	8.41	196.35	0.09%	0.24%	0.03%
11	-10.87	-8.43	196.30	-10.88	-8.41	196.35	0.09%	0.24%	0.03%

Load Combination 2

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	5.69	6.33	146.14	5.70	6.31	146.18	0.18%	0.32%	0.03%
5	-2.83	11.52	332.69	-2.83	11.51	332.60	0.00%	0.09%	0.03%
7	-10.51	6.33	148.31	-10.52	6.31	148.35	0.10%	0.32%	0.03%
11	-10.51	-6.33	148.31	-10.52	-6.31	148.35	0.10%	0.32%	0.03%

Load Combination 3

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	8.15	2.17	145.31	8.16	2.16	145.35	0.12%	0.46%	0.03%
5	0.00	4.35	328.87	0.00	4.34	328.79	0.00%	0.23%	0.02%
7	-8.15	2.17	145.31	-8.16	2.16	145.35	0.12%	0.46%	0.03%
11	-8.16	-10.38	149.13	-8.17	-10.35	149.18	0.12%	0.29%	0.03%

Load Combination 4

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	5.32	-0.43	93.51	5.33	-0.43	93.54	0.19%	0.00%	0.03%
5	0.00	-0.80	205.03	0.00	-0.81	204.97	0.00%	-	0.03%
7	-5.32	-0.43	93.51	-5.33	-0.43	93.54	0.19%	0.00%	0.03%
11	-5.33	-7.94	97.41	-5.33	-7.92	97.44	0.00%	0.25%	0.03%

Table 6. Nodal displacements for two story frame

Load Combination 1

Node no.	MicasPlus			STAAD III			% Difference		
	Tx (inch)	Ty (inch)	Tz (inch)	Tx (inch)	Ty (inch)	Tz (inch)	Tx	Ty	Tz
4	-0.00463	0.00542	-0.0239	-0.00466	0.00543	-0.0239	0.65%	0.18%	0.00%
8	0.00463	-0.00542	-0.0239	0.00466	-0.00543	-0.02386	0.65%	0.18%	0.17%
13	0.00679	0.00718	-0.0335	0.00681	0.00718	-0.0335	0.29%	0.00%	0.00%
18	0	-0.00753	-0.0771	0	-0.00756	-0.0771	0.00%	0.40%	0.00%

Load Combination 2

Node no.	MicasPlus			STAAD III			% Difference		
	Tx (inch)	Ty (inch)	Tz (inch)	Tx (inch)	Ty (inch)	Tz (inch)	Tx	Ty	Tz
4	0.0243	0.00407	-0.0178	0.0241	0.00408	-0.0178	0.82%	0.25%	0.00%
8	0.0302	-0.00407	-0.018	0.0301	-0.00408	-0.018	0.33%	0.25%	0.00%
13	0.0432	0.00538	-0.025	0.043	0.00539	-0.025	0.46%	0.19%	0.00%
18	0.0377	-0.00565	-0.0578	0.0375	-0.00567	-0.0578	0.53%	0.35%	0.00%

Load Combination 3

Node no.	MicasPlus			STAAD III			% Difference		
	Tx (inch)	Ty (inch)	Tz (inch)	Tx (inch)	Ty (inch)	Tz (inch)	Tx	Ty	Tz
4	-0.00349	0.0707	-0.0181	-0.00351	0.0703	-0.0181	0.57%	0.57%	0.00%
8	0.00346	0.0636	-0.0177	0.00348	0.0632	-0.0177	0.58%	0.63%	0.00%
13	0.0051	0.1226	-0.0248	0.00512	0.1221	-0.0248	0.39%	0.41%	0.00%
18	0	0.1335	-0.0584	0	0.1329	-0.0584	0.00%	0.45%	0.00%

Load Combination 4

Node no.	MicasPlus			STAAD III			% Difference		
	Tx (inch)	Ty (inch)	Tz (inch)	Tx (inch)	Ty (inch)	Tz (inch)	Tx	Ty	Tz
4	-0.00228	0.0704	-0.0118	-0.00229	0.07	-0.0118	0.44%	0.57%	0.00%
8	0.00225	0.0666	-0.0113	0.00226	0.0662	-0.01131	0.44%	0.60%	0.09%
13	0.00333	0.1227	-0.0159	0.00335	0.1222	-0.0159	0.60%	0.41%	0.00%
18	0	0.1387	-0.0368	0	0.1381	-0.0368	0.00%	0.43%	0.00%

Table 7. Element end forces for two story frame

Load Combination 1

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
1	1	196.30	-43.60	-34.29	196.35	-43.83	-34.33	0.03%	0.53%	0.12%
10	6	224.22	0.00	-255.42	224.16	0.00	-256.54	0.03%	0.00%	0.44%
21	18	34.25	-0.04	435.72	34.39	-0.05	435.59	0.41%	-	0.03%
24	13	34.49	0.00	187.63	34.53	0.01	187.86	0.12%	-	0.12%

Load Combination 2 (kips or kip-ft)

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
1	1	146.14	-16.04	-25.72	146.18	-16.22	-25.75	0.03%	1.12%	0.12%
10	6	168.15	5.84	-191.57	168.11	5.85	-192.40	0.02%	0.17%	0.43%
21	18	26.27	-0.03	323.08	26.37	-0.03	322.98	0.38%	0.00%	0.03%
24	13	25.87	0.00	140.72	25.90	0.00	140.89	0.12%	0.00%	0.12%

Load Combination 3 (kips or kip-ft)

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
1	1	145.31	-32.68	7.65	145.35	-32.85	7.58	0.03%	0.52%	0.92%
10	6	167.01	0.00	-186.63	166.97	0.00	-187.46	0.02%	0.00%	0.44%
21	18	25.63	2.63	326.70	25.73	2.61	326.60	0.39%	0.76%	0.03%
24	13	27.09	1.00	130.48	27.12	1.01	130.65	0.11%	1.00%	0.13%

Load Combination 4 (kips or kip-ft)

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
1	1	93.51	-21.33	18.54	93.54	-21.45	18.48	0.03%	0.56%	0.32%
10	6	104.40	0.00	-103.37	104.38	0.00	-103.82	0.02%	0.00%	0.44%
21	18	16.70	2.72	214.23	16.77	2.70	214.17	0.42%	0.74%	0.03%
24	13	16.76	1.02	74.69	16.78	1.02	74.79	0.12%	0.00%	0.13%

displacements also agree closely, differing by a maximum of 0.82%. Element end forces agree almost as well as the support reactions and the nodal displacements, with a maximum difference of 1.12%. Most of the values for element end forces agree within 0.5%. Many of them agree within 0.10%. These differences are insignificant in the design of a member.

The design results produced by MicasPlus and STAAD III for beams 14, 17, 20, and 25 are shown in Table 8. Beams 14 and 17 are exterior beams located on the first floor of the two story structure. Beams 20 and 25 are exterior beams located on the roof. The steel area required given by MicasPlus and STAAD III agreed almost exactly for top steel longitudinal reinforcing and stirrup spacing. The largest difference was 2.22%. The values for the bottom reinforcing did not agree as closely. The percentage difference ranged from 4.44% to 9.22%. However, these are not large differences. It is interesting to note that MicasPlus gave conservative results compared to STAAD III for most cases.

Table 8 also presents a comparison of the steel areas provided by MicasPlus and those obtained by hand calculations for columns 1, 3, 9, and 13. Column 1 is a corner column on the first floor. Column 3 is an exterior column on the first floor. Columns 9 and 13 are corner columns on the top floor. The MicasPlus results agreed closely with hand calculations. The first floor columns agreed within 5.38%, with MicasPlus being more conservative than those obtained by hand

Table 8. Design results for two story frame

a. Beam longitudinal reinforcement - top

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	in^2	in^2	in^2	in^2		
14	2.48	4.20	2.48	4.20	0.00%	0.00%
17	4.20	2.48	4.20	2.48	0.00%	0.00%
20	1.80	4.20	1.76	4.20	-2.22%	0.00%
25	3.16	3.16	3.16	3.16	0.00%	0.00%

b. Beam longitudinal reinforcement - bottom

Element Number	MicasPlus	STAAD III	% Differ.
	Midspan	Midspan	
	in^2	in^2	
14	2.17	2.00	-7.83%
17	2.17	2.37	9.22%
20	2.48	2.37	-4.44%
25	2.40	2.20	-8.33%

c. Stirrup spacing

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	inches	inches	inches	inches		
14	12.80	12.70	12.80	12.80	0.00%	0.79%
17	12.70	12.80	12.80	12.80	0.79%	0.00%
20	12.80	12.70	12.80	12.80	0.00%	0.79%
25	8.70	8.70	8.80	8.80	1.15%	1.15%

d. Column reinforcement

Element Number	MicasPlus	Hand	% Differ.
	Area steel	Area steel	
	in^2	in^2	
1	3.72	3.52	-5.38%
3	3.72	3.52	-5.38%
9	12.00	12.00	0.00%
13	12.00	12.00	0.00%

calculations. The second floor columns agreed exactly.

5.4 Seven Story Frame

A comparison of the analysis results produced by MicasPlus and STAAD III for the seven story frame is shown in Tables 9, 10, and 11. Table 9 shows support reactions at nodes 1, 5, 6, and 12. Nodes 1, 5, 6, and 12 are support nodes located at the base of the building (see Appendix A, Fig. A-3). Table 10 shows nodal displacements for nodes 16, 25, 40, 50, 61, 78, 84, and 96. Nodes 16, 25, 40, 50, 61, 78, 84, and 96 are nodes on each of the seven floors, except for nodes 78 and 84 which are both located on the sixth level. Table 11 shows element end forces for elements 5, 30, 60, 88, 118, 130, 159, and 202. These elements represent interior and exterior beams and columns located at each floor.

There is almost no difference in the support reactions, nodal displacements, and element end forces computed by MicasPlus and STAAD III. Of the quantities that are presented in the tables, there were only two cases in which the difference in the results was more than 1.00%. In both cases the quantities compared were of very small magnitude. Therefore the difference is insignificant and was most likely caused by round-off error.

Table 9. Support reactions for seven story frame

Load Combination 1

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	-4.22	-2.72	511.67	-4.22	-2.72	511.88	0.00%	0.00%	0.04%
5	1.15	0.00	1946.42	1.15	0.00	1946.18	0.00%	0.00%	0.01%
6	0.10	3.49	922.99	0.10	3.48	922.97	0.00%	0.29%	0.00%
12	4.22	2.72	511.67	4.22	2.72	511.88	0.00%	0.00%	0.04%

Load Combination 2

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	-7.35	-2.05	394.76	-7.35	-2.05	394.92	0.00%	0.00%	0.04%
5	-16.59	0.00	1457.00	-16.60	0.00	1456.83	0.06%	0.00%	0.01%
6	-4.87	2.61	691.23	-4.87	2.61	691.22	0.00%	0.00%	0.00%
12	-1.04	2.04	372.75	-1.05	2.03	372.91	0.96%	0.49%	0.04%

Load Combination 3

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	-3.18	-9.89	409.42	-3.18	-9.89	409.56	0.00%	0.00%	0.03%
5	0.87	-35.42	1459.81	0.86	-35.46	1459.63	1.15%	0.11%	0.01%
6	0.08	-4.79	649.18	0.08	-4.78	649.14	0.00%	0.21%	0.01%
12	3.16	-5.83	358.09	3.16	-5.83	358.26	0.00%	0.00%	0.05%

Load Combination 4

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	STAAD III Fx (kips)	STAAD III Fy (kips)	STAAD III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	-1.48	-9.01	217.45	-1.48	-9.00	217.51	0.00%	0.11%	0.03%
5	0.42	-36.12	690.65	0.42	-36.15	690.57	0.00%	0.08%	0.01%
6	0.04	-6.30	289.82	0.04	-6.29	289.78	0.00%	0.16%	0.01%
12	1.46	-7.02	165.12	1.46	-7.02	165.21	0.00%	0.00%	0.05%

Table 10. Nodal displacements for seven story frame.

Load Combination 1

Node no.	MicasPlus Tx (inch)	MicasPlus Ty (inch)	MicasPlus Tz (inch)	STAAD III Tx (inch)	STAAD III Ty (inch)	STAAD III Tz (inch)	% Difference Tx	% Difference Ty	% Difference Tz
16	0.00202	0.00227	-0.1914	0.00203	0.00228	-0.1914	0.50%	0.44%	0.00%
25	-0.00054	-0.00016	-0.1628	-0.00055	-0.00017	-0.1629	-	-	0.06%
40	0.00014	0.0002	-0.3794	0.00014	0.00021	-0.3794	0.00%	-	0.00%
50	0.0006	0	-0.4332	0.0006	0	-0.4331	0.00%	0.00%	0.02%
61	-0.00016	-0.00008	-0.277	-0.00016	-0.00009	-0.2772	0.00%	-	0.07%
78	0.00094	-0.00127	-0.5341	0.00094	-0.00128	-0.5341	0.00%	0.79%	0.00%
84	-0.00322	-0.00139	-0.2959	-0.00323	-0.0014	-0.296	0.31%	0.72%	0.03%
96	0.0114	0.00463	-0.3048	0.01147	0.00465	-0.305	0.61%	0.43%	0.07%

Load Combination 2

Node no.	MicasPlus Tx (inch)	MicasPlus Ty (inch)	MicasPlus Tz (inch)	STAAD III Tx (inch)	STAAD III Ty (inch)	STAAD III Tz (inch)	% Difference Tx	% Difference Ty	% Difference Tz
16	0.2137	0.0017	-0.1433	0.2131	0.00171	-0.1433	0.28%	0.59%	0.00%
25	0.3271	-0.00015	-0.1255	0.3259	-0.00015	-0.1255	0.37%	0.00%	0.00%
40	0.4273	0.00015	-0.2842	0.4257	0.00015	-0.2842	0.37%	0.00%	0.00%
50	0.5127	0	-0.332	0.5106	0	-0.3319	0.41%	0.00%	0.03%
61	0.5765	-0.00005	-0.2126	0.5741	-0.00005	-0.2127	0.42%	0.00%	0.05%
78	0.6193	-0.00095	-0.4002	0.6166	-0.00096	-0.4002	0.44%	-	0.00%
84	0.6177	-0.00102	-0.2169	0.6151	-0.00103	-0.217	0.42%	-	0.05%
96	0.6478	0.00347	-0.2236	0.6451	0.00348	-0.2237	0.42%	0.29%	0.04%

Load Combination 3

Node no.	MicasPlus Tx (inch)	MicasPlus Ty (inch)	MicasPlus Tz (inch)	STAAD III Tx (inch)	STAAD III Ty (inch)	STAAD III Tz (inch)	% Difference Tx	% Difference Ty	% Difference Tz
16	0.00148	0.3235	-0.1525	0.00148	0.3219	-0.1525	0.00%	0.49%	0.00%
25	-0.0005	0.5735	-0.1299	-0.00051	0.571	-0.1299	-	0.44%	0.00%
40	0.00007	0.6997	-0.3003	0.00007	0.6961	-0.3003	0.00%	0.51%	0.00%
50	0.00045	0.8763	-0.3249	0.00045	0.8719	-0.3248	0.00%	0.50%	0.03%
61	-0.00005	0.9817	-0.219	-0.00005	0.9767	-0.2191	0.00%	0.51%	0.05%
78	0.00064	1.1034	-0.3819	0.00064	1.0976	-0.3819	0.00%	0.53%	0.00%
84	-0.00228	1.0517	-0.2104	-0.0023	1.0461	-0.2105	0.88%	0.53%	0.05%
96	0.00864	1.0903	-0.2171	0.00866	1.0847	-0.2172	0.23%	0.51%	0.05%

Load Combination 4

Node no.	MicasPlus Tx (inch)	MicasPlus Ty (inch)	MicasPlus Tz (inch)	STAAD III Tx (inch)	STAAD III Ty (inch)	STAAD III Tz (inch)	% Difference Tx	% Difference Ty	% Difference Tz
16	0.00066	0.3289	-0.0782	0.00066	0.3273	-0.0782	0.00%	0.49%	0.00%
25	-0.00028	0.5848	-0.0685	-0.00029	0.5822	-0.0685	-	0.44%	0.00%
40	0.00001	0.7134	-0.1528	0.00001	0.7096	-0.1528	0.00%	0.53%	0.00%
50	0.0002	0.8935	-0.1565	0.00021	0.889	-0.1564	-	0.50%	0.06%
61	0.00002	1.001	-0.1143	0.00002	0.9958	-0.1144	0.00%	0.52%	0.09%
78	0.00026	1.1256	-0.1732	0.00026	1.1197	-0.1732	0.00%	0.52%	0.00%
84	-0.00098	1.0728	-0.0981	-0.00099	1.0673	-0.0981	-	0.51%	0.00%
96	0.00403	1.1098	-0.1013	0.00404	1.10405	-0.1014	0.25%	0.52%	0.10%

Table 11. Element end forces for seven story frame.

Load Combination 1

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
5	5	1946.42	-4.82	0.00	1946.18	-4.80	0.00	0.01%	0.41%	0.00%
30	25	-436.36	75.19	53.46	-436.55	75.32	53.58	0.04%	0.17%	0.22%
60	38	-631.69	105.51	0.00	-631.58	105.65	0.00	0.02%	0.13%	0.00%
88	49	-291.26	76.90	57.54	-291.40	77.00	57.65	0.05%	0.13%	0.19%
118	50	381.47	115.73	0.00	381.41	115.96	0.00	0.02%	0.20%	0.00%
130	62	0.30	0.02	65.88	0.31	0.02	65.74	-	0.00%	0.21%
159	75	5.02	0.01	-127.67	5.05	0.01	-127.98	0.60%	0.00%	0.24%
202	95	-31.78	0.00	-186.77	-31.87	0.00	-187.13	0.28%	0.00%	0.19%

Load Combination 2

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
5	5	1457.00	195.56	0.00	1456.83	195.57	0.00	0.01%	0.01%	0.00%
30	25	-335.43	73.40	40.27	-335.57	73.47	40.36	0.04%	0.10%	0.22%
60	38	-482.37	99.85	0.00	-482.29	99.97	0.00	0.02%	0.12%	0.00%
88	49	-222.20	71.38	43.39	-222.30	71.43	43.48	0.05%	0.07%	0.21%
118	50	288.59	97.40	0.00	288.54	97.57	0.00	0.02%	0.17%	0.00%
130	62	0.19	0.83	48.85	0.19	0.82	48.74	0.00%	1.20%	0.23%
159	75	3.84	0.79	-96.26	3.86	0.79	-96.50	0.52%	0.00%	0.25%
202	95	-26.91	0.00	-136.94	-26.98	0.00	-137.22	0.26%	0.00%	0.20%

Load Combination 3

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
5	5	1459.81	-3.62	382.70	1459.63	-3.60	382.77	0.01%	0.55%	0.02%
30	25	-346.06	56.68	73.27	-346.19	56.79	73.30	0.04%	0.19%	0.04%
60	38	-473.76	79.13	48.07	-473.68	79.24	48.05	0.02%	0.14%	0.04%
88	49	-226.97	58.09	67.98	-227.07	58.17	68.02	0.04%	0.14%	0.06%
118	50	286.09	86.80	28.07	286.04	86.97	28.06	0.02%	0.20%	0.04%
130	62	3.36	-1.04	73.96	3.37	-1.04	73.84	0.30%	0.00%	0.16%
159	75	0.52	-1.63	-80.74	0.54	-1.63	-80.99	-	0.00%	0.31%
202	95	-23.84	-3.59	-140.07	-23.90	-3.57	-140.34	0.25%	0.56%	0.19%

Load Combination 4

El	Node	MicasPlus			STAAD III			% Difference		
no.	no.	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial (kips)	My (k-ft)	Mz (k-ft)	Axial	My	Mz
5	5	690.65	-1.81	390.20	690.57	-1.80	390.27	0.01%	0.55%	0.02%
30	25	-180.69	26.45	53.53	-180.74	26.50	53.41	0.03%	0.19%	0.22%
60	38	-227.49	35.09	49.01	-227.45	35.14	48.99	0.02%	0.14%	0.04%
88	49	-116.12	27.17	46.27	-116.16	27.20	46.27	0.03%	0.11%	0.00%
118	50	138.03	38.61	28.62	138.01	38.68	28.61	0.01%	0.18%	0.03%
130	62	3.31	-1.07	50.64	3.32	-1.07	50.58	0.30%	0.00%	0.12%
159	75	-1.49	-1.67	-31.08	-1.48	-1.67	-31.22	0.67%	0.00%	0.45%
202	95	-10.63	-3.66	-62.28	-10.65	-3.64	-62.40	0.19%	0.55%	0.19%

The design results for nine beams are presented in Table 12. Beams 42, 53, and 54 are located on the second floor. Beam 42 is an exterior beam running in the transverse direction and is connected to a corner column. Beam 53 is an exterior beam located along one longitudinal side at midpoint. Beam 54 is an interior beam. Beams 100, 111, and 112 are located in similar places, except they are on the fourth floor. Beams 187, 198, and 199 are located in the same relative position but on the seventh floor. The results agreed within 8.33% except for one case in which MicasPlus gave a smaller area of steel. The difference in the results for this case was 19.35%. This was for the bottom longitudinal reinforcement for beam 198 where the amount of steel required was quite small. For most cases, MicasPlus and STAAD III gave almost exactly the same amount of longitudinal steel for both the top and the bottom. As was the case for the previous structures, the spacing of the stirrups computed by both programs agreed very closely. The difference in the stirrup spacing at any given location was always less than 0.2 inches.

The design results for nine columns of the seven story building are compared in Table 12d. Columns 1, 4, and 8 are all located on the ground floor. Column 1 is a corner column. Column 4 is an exterior edge column, and column 8 is an interior column. Columns 59, 62, and 66 are in similar locations on the third floor. Columns 146, 149, and 153 are in similar locations on the sixth floor. There is no difference in the column design results obtained by MicasPlus and those obtained

Table 12. Design results for seven story frame

a. Beam longitudinal reinforcement - top

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	in^2	in^2	in^2	in^2		
42	2.40	1.24	2.20	1.32	-8.33%	6.45%
53	2.40	2.40	2.20	2.20	-8.33%	-8.33%
54	5.08	5.08	5.08	5.08	0.00%	0.00%
100	1.76	1.00	1.76	0.93	0.00%	-7.00%
111	2.40	2.40	2.20	2.20	-8.33%	-8.33%
112	5.08	5.08	5.08	5.08	0.00%	0.00%
187	1.24	1.00	1.32	1.00	6.45%	0.00%
198	2.40	2.40	2.20	2.20	-8.33%	-8.33%
199	5.08	5.08	4.68	4.68	-7.87%	-7.87%

b. Beam longitudinal reinforcement - bottom

Element Number	MicasPlus	STAAD III	% Differ.
	Midspan	Midspan	
	in^2	in^2	
42	0.80	0.80	0.00%
53	1.24	1.32	6.45%
54	2.20	2.20	0.00%
100	0.80	0.80	0.00%
111	1.24	1.32	6.45%
112	2.20	2.37	7.73%
187	1.00	0.93	-7.00%
198	1.24	1.00	-19.35%
199	2.20	2.37	7.73%

Table 12, continued.

c. Stirrup spacing

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	inches	inches	inches	inches		
42	8.7	8.8	8.8	8.8	1.15%	0.00%
53	8.7	8.7	8.8	8.8	1.15%	1.15%
54	8.6	8.6	8.8	8.8	2.33%	2.33%
100	8.8	8.8	8.8	8.8	0.00%	0.00%
111	8.7	8.7	8.8	8.8	1.15%	1.15%
112	8.6	8.6	8.8	8.8	2.33%	2.33%
187	8.8	8.8	8.8	8.8	0.00%	0.00%
198	8.7	8.7	8.8	8.8	1.15%	1.15%
199	8.6	8.6	8.8	8.8	2.33%	2.33%

d. Column reinforcement

Element Number	MicasPlus	Hand	% Differ.
	Area steel	Area steel	
	in^2	in^2	
1	3.16	3.16	0.00%
4	18.00	18.00	0.00%
8	27.00	27.00	0.00%
59	3.16	3.16	0.00%
62	6.24	6.24	0.00%
66	9.00	9.00	0.00%
146	3.16	3.16	0.00%
149	3.16	3.16	0.00%
153	3.16	3.16	0.00%

by hand calculations. This is not surprising since the forces were large and therefore a large area of steel was required. As the required area of steel increases, the possibility for variation in design decreases.

5.5 Fourteen Story Frame

Tables 13, 14, and 15 show a comparison of analysis results obtained by MicasPlus to those obtained by STAAD III. Table 13 shows the support reactions for nodes 1, 6, 12, and 14. Nodes 1, 6, 12, and 14 are support nodes which are located at the ground level (see Appendix A, Fig. A-4). Table 14 shows the nodal displacement for twelve nodes in the fourteen story structure. The nodes used for the nodal displacement comparisons were chosen based on their relatively large displacements. They are located at varying elevations throughout the building. Table 15 shows element end forces for twelve elements. Elements used in element end force comparisons were chosen based on large end reactions. The elements presented in the table are elements located at varying elevations throughout the building.

Analysis results for the fourteen story frame obtained by MicasPlus agree closely with those obtained by STAAD III. Following is a summary of the results.

Table 13 compares the support reactions obtained by MicasPlus and STAAD III for the fourteen story frame. Results agree within 0.05% for the vertical reactions. Results for forces in the X and Y directions also agree closely (within 1.5%). However, the X and Y reactions are insignificant in magnitude when compared to the vertical reactions.

Table 14 compares nodal reactions for the fourteen story frame obtained by MicasPlus and STAAD III. The nodal displacements in all directions and all load combinations agree closely (within 1%).

Table 15 compares the element end forces obtained by MicasPlus with those obtained by STAAD III. Axial loads for all load combinations differ by less than 1%. Values for moments in the Y and Z directions differ by 2.3% in several cases, but generally agree within 1%.

Design results obtained by MicasPlus agree closely to those obtained by STAAD III in some cases. In other cases, there is a substantial difference. Following is a summary of Table 16, which compares the design results for the fourteen story frame obtained by MicasPlus with those obtained by STAAD III.

The elements used to compared design results were chosen to represent elements with diverse loading conditions. Elements 57 and 60 are an exterior edge beam

Table 13. Support reactions for fourteen story frame.

Load Combination 1

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	Staad III Fx (kips)	Staad III Fy (kips)	Staad III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	-16.92	-16.82	1652.54	-16.94	-16.94	1653.25	0.12%	0.71%	0.04%
6	-1.57	-1.57	6093.17	-1.61	-1.61	6094.83	-	-	0.03%
12	1.22	27.15	2781.72	1.23	27.38	2780.84	0.82%	0.85%	0.03%
14	27.15	-1.22	2781.72	27.38	-1.23	2780.84	0.85%	0.82%	0.03%

Load Combination 2

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	Staad III Fx (kips)	Staad III Fy (kips)	Staad III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	-12.65	-25.67	1292.45	-12.74	-25.72	1292.87	0.71%	0.19%	0.03%
6	-1.20	-62.67	4588.17	-1.22	-62.79	4589.69	-	0.19%	0.03%
12	0.88	4.10	1970.57	0.89	4.34	1969.81	-	-	0.04%
14	20.38	-17.24	2086.94	20.55	-17.23	2086.36	0.83%	0.06%	0.03%

Load Combination 3

Node no.	MicasPlus Fx (kips)	MicasPlus Fy (kips)	MicasPlus Fz (kips)	Staad III Fx (kips)	Staad III Fy (kips)	Staad III Fz (kips)	% Difference Fx	% Difference Fy	% Difference Fz
1	-5.23	-18.50	606.30	-5.27	-18.50	606.40	0.76%	0.00%	0.02%
6	0.45	-63.13	1949.43	0.46	-63.23	1950.41	-	0.16%	0.05%
12	0.33	-8.49	770.33	0.34	-8.36	769.96	-	1.53%	0.05%
14	8.10	-17.02	888.99	8.17	-17.00	888.78	0.86%	0.12%	0.02%

Table 14. Nodal displacements for fourteen story frame.

Load Combination 1

Node no.	MicasPlus Tx (inch)	MicasPlus Ty (inch)	MicasPlus Tz (inch)	STAAD III Tx (inch)	STAAD III Ty (inch)	STAAD III Tz (inch)	% Difference Tx	% Difference Ty	% Difference Tz
52	0.00094	-0.00094	-0.1489	0.00095	-0.00095	-0.149	-	-	0.07%
75	0.0008	0.0008	-0.2664	0.0008	0.0008	-0.2665	0.00%	0.00%	0.04%
104	0.0003	0.00041	-0.5314	0.0003	0.00041	-0.5312	0.00%	0.00%	0.04%
119	-0.00027	0.00027	-0.5078	-0.00027	0.00027	-0.508	0.00%	0.00%	0.04%
163	0.00014	-0.00016	-0.8734	0.00011	-0.00016	-0.8729	-	0.00%	0.06%
172	-0.00016	-0.00014	-0.8734	-0.00016	-0.00008	-0.8729	0.00%	-	0.06%
180	0.00023	-0.00023	-0.5661	0.00023	-0.00023	-0.5664	0.00%	0.00%	0.05%
194	0.00133	0.00011	-0.9854	0.00135	0.00011	-0.9848	-	0.00%	0.06%
212	0.00394	-0.00394	-0.6142	0.00396	-0.00396	-0.6145	0.51%	0.51%	0.05%
223	-0.00283	-0.00124	-1.0175	-0.00285	-0.00125	-1.0169	0.71%	0.81%	0.06%
226	-0.00948	-0.00581	-1.0335	-0.00945	-0.00583	-1.0329	0.32%	0.34%	0.06%
229	-0.00581	-0.00948	-1.0335	-0.00583	-0.00945	-1.0329	0.34%	0.32%	0.06%

Load Combination 2

Node no.	MicasPlus Tx (inch)	MicasPlus Ty (inch)	MicasPlus Tz (inch)	STAAD III Tx (inch)	STAAD III Ty (inch)	STAAD III Tz (inch)	% Difference Tx	% Difference Ty	% Difference Tz
52	0.000725	0.229	-0.1069	0.00073	0.22749	-0.107	0.69%	0.66%	0.09%
75	0.0006	0.3063	-0.1989	0.0006	0.3041	-0.199	0.00%	0.72%	0.05%
104	0.00026	0.5213	-0.3785	0.00026	0.5175	-0.3783	0.00%	0.73%	0.05%
119	-0.00021	0.6163	-0.3792	-0.00021	0.6118	-0.3793	0.00%	0.73%	0.03%
163	0.0001	0.9186	-0.6546	0.0001	0.9114	-0.6543	0.00%	0.78%	0.05%
172	-0.00016	0.8806	-0.6274	-0.00016	0.873	-0.6271	0.00%	0.86%	0.05%
180	0.00025	0.9832	-0.4107	0.00026	0.9754	-0.411	-	0.79%	0.07%
194	0.00098	1.0287	-0.7395	0.001	1.0204	-0.7391	-	0.81%	0.05%
212	0.00295	1.0591	-0.4464	0.00297	1.0505	-0.4467	0.68%	0.81%	0.07%
223	-0.00211	1.0599	-0.7626	-0.00213	1.0513	-0.7621	0.95%	0.81%	0.07%
226	-0.00714	1.0723	-0.7757	-0.00712	1.0635	-0.7753	0.28%	0.82%	0.05%
229	-0.00448	1.0623	-0.804	-0.00449	1.0528	-0.8036	0.22%	0.89%	0.05%

Load Combination 3

Node no.	MicasPlus Tx (inch)	MicasPlus Ty (inch)	MicasPlus Tz (inch)	STAAD III Tx (inch)	STAAD III Ty (inch)	STAAD III Tz (inch)	% Difference Tx	% Difference Ty	% Difference Tz
52	0.0003	0.234	-0.0445	0.0003	0.2324	-0.0445	0.00%	0.68%	0.00%
75	0.00024	0.3119	-0.0828	0.00024	0.3097	-0.0828	0.00%	0.71%	0.00%
104	0.00012	0.5313	-0.1476	0.00012	0.5274	-0.1475	0.00%	0.73%	0.07%
119	-0.00009	0.6283	-0.1568	-0.00009	0.6237	-0.1569	0.00%	0.73%	0.06%
163	0.00003	0.9367	-0.2748	0.00002	0.9293	-0.2746	-	0.79%	0.07%
172	-0.00009	0.8979	-0.2471	-0.00009	0.8901	-0.2469	0.00%	0.87%	0.08%
180	0.00015	1.0026	-0.1707	0.00015	0.9946	-0.1708	0.00%	0.80%	0.06%
194	0.00041	1.0488	-0.3107	0.00042	1.0404	-0.3105	-	0.80%	0.06%
212	0.00119	1.0817	-0.1858	0.0012	1.0729	-0.1859	0.84%	0.81%	0.05%
223	-0.00083	1.0813	-0.3196	-0.00084	1.0724	-0.3194	-	0.82%	0.06%
226	-0.00276	1.096	-0.3257	-0.00276	1.087	-0.3255	0.00%	0.82%	0.06%
229	-0.00188	1.0876	-0.3546	-0.00188	1.078	-0.3544	0.00%	0.88%	0.06%

Table 15. Element end forces for fourteen story frame

Load Combination 1

El no.	Node no.	MicasPlus Axial (kips)	MicasPlus My (k-ft)	MicasPlus Mz (k-ft)	STAAD III Axial (kips)	STAAD III My (k-ft)	STAAD III Mz (k-ft)	% Difference Axial	% Difference My	% Difference Mz
1	1	1652.54	57.92	57.92	1653.25	59.21	59.21	0.04%	2.23%	2.23%
6	6	6093.17	13.67	13.67	6094.83	13.67	13.67	0.03%	0.00%	0.00%
42	18	2568.78	199.44	1.03	2567.91	200.04	1.09	0.03%	0.30%	-
66	45	-1.85	-0.09	234.76	-1.82	-0.09	234.99	1.62%	0.00%	0.10%
81	33	1415.20	122.63	122.63	1415.86	122.72	122.72	0.05%	0.08%	0.08%
121	49	1294.45	120.24	120.24	1295.08	120.45	120.45	0.05%	0.17%	0.17%
200	96	3.63	-0.13	-249.02	3.63	-0.13	-249.39	0.00%	0.00%	0.15%
321	145	-696.80	128.06	128.06	-697.19	128.25	128.25	0.06%	0.15%	0.15%
403	163	771.09	122.77	-31.74	770.46	122.26	-31.95	0.08%	0.41%	0.66%
415	175	771.09	-122.77	-31.74	770.46	-122.26	-31.95	0.08%	0.42%	0.66%
513	217	9.64	0.00	-214.50	9.70	0.00	-214.55	0.63%	0.00%	0.02%
560	236	38.23	0.62	170.04	38.33	0.62	169.84	0.26%	0.00%	0.12%

Load Combination 2

El no.	Node no.	MicasPlus Axial (kips)	MicasPlus My (k-ft)	MicasPlus Mz (k-ft)	STAAD III Axial (kips)	STAAD III My (k-ft)	STAAD III Mz (k-ft)	% Difference Axial	% Difference My	% Difference Mz
1	1	1292.45	43.58	228.87	1292.87	44.55	229.16	0.03%	2.23%	0.13%
6	6	4588.17	10.33	1029.35	4589.65	10.33	1028.27	0.03%	0.00%	0.10%
42	18	1927.25	149.67	131.85	1926.65	150.12	131.60	0.03%	0.30%	0.19%
66	45	0.34	-0.84	244.81	0.36	-0.85	244.90	-	1.19%	0.04%
81	33	1105.32	92.59	154.62	1105.70	92.67	154.45	0.03%	0.09%	0.11%
121	49	1008.93	90.99	133.35	1009.30	91.15	133.32	0.04%	0.18%	0.02%
200	96	2.80	1.96	-184.41	2.81	1.95	-184.67	0.36%	0.51%	0.14%
321	145	-535.05	97.60	124.93	-535.30	97.75	124.94	0.05%	0.15%	0.01%
403	163	577.52	91.58	4.87	577.04	91.19	4.58	0.08%	0.43%	-
415	175	577.52	-91.58	4.87	577.04	-91.19	4.58	0.08%	0.43%	-
513	217	7.23	-0.19	-160.63	7.27	-0.19	-160.67	0.55%	0.00%	0.02%
560	236	28.19	0.79	130.31	28.26	0.85	130.18	0.25%	-	0.10%

Load Combination 3

El no.	Node no.	MicasPlus Axial (kips)	MicasPlus My (k-ft)	MicasPlus Mz (k-ft)	STAAD III Axial (kips)	STAAD III My (k-ft)	STAAD III Mz (k-ft)	% Difference Axial	% Difference My	% Difference Mz
1	1	606.30	18.00	206.93	606.40	18.40	206.63	0.02%	2.22%	0.14%
6	6	1949.43	3.99	1042.99	1950.41	3.98	1041.88	0.05%	0.25%	0.11%
42	18	819.19	59.50	133.90	818.98	59.68	133.61	0.03%	0.30%	0.22%
66	45	1.20	-0.82	142.44	1.20	-0.83	142.42	0.00%	1.22%	0.01%
81	33	513.71	38.41	101.66	513.80	38.44	101.44	0.02%	0.08%	0.22%
121	49	465.50	37.82	81.01	465.59	37.89	80.88	0.02%	0.19%	0.16%
200	96	1.19	2.07	-74.07	1.19	2.05	-74.16	0.00%	0.97%	0.12%
321	145	-236.93	40.74	68.61	-237.01	40.80	68.52	0.03%	0.15%	0.13%
403	163	241.15	34.75	19.87	240.95	34.58	19.68	0.08%	0.49%	0.96%
415	175	241.15	-34.75	19.87	240.95	-34.58	19.68	0.08%	0.49%	0.96%
513	217	2.91	-0.19	-66.15	2.93	-0.19	-66.17	0.69%	0.00%	0.03%
560	236	11.17	0.53	56.28	11.20	0.59	56.24	0.27%	-	0.07%

Table 16. Design results for fourteen story frame

a. Beam longitudinal reinforcement - top

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	in^2	in^2	in^2	in^2		
57	3.00	2.40	3.00	2.37	0.00%	-1.25%
60	3.60	3.95	3.60	3.95	0.00%	0.00%
75	3.60	3.60	3.60	3.60	0.00%	0.00%
76	3.00	3.00	3.00	3.00	0.00%	0.00%
217	3.00	2.20	3.00	1.86	0.00%	-15.45%
220	3.00	4.74	3.00	4.74	0.00%	0.00%
235	3.60	3.60	3.60	3.60	0.00%	0.00%
236	3.00	3.00	2.64	2.64	-12.00%	-12.00%
377	3.16	1.76	3.16	1.76	0.00%	0.00%
380	2.17	4.74	2.17	4.68	0.00%	-1.27%
395	3.60	3.60	3.60	3.60	0.00%	0.00%
396	2.40	2.40	2.40	2.40	0.00%	0.00%
537	3.00	2.20	2.64	1.86	-12.00%	-15.45%
540	1.86	3.95	3.95	3.95	112.37%	0.00%
555	3.95	3.95	3.95	3.95	0.00%	0.00%
556	2.40	2.40	2.37	2.37	-1.25%	-1.25%

b. Beam longitudinal reinforcement - bottom

Element Number	MicasPlus	STAAD III	% Differ.
	Midspan	Midspan	
	in^2	in^2	
57	1.20	1.32	10.00%
60	1.86	1.76	-5.38%
75	1.86	1.76	-5.38%
76	1.20	1.32	10.00%
217	1.24	1.24	0.00%
220	1.86	1.80	-3.23%
235	1.86	1.76	-5.38%
236	1.20	1.32	10.00%
377	1.55	1.32	-14.84%
380	2.17	2.37	9.22%
395	1.86	1.60	-13.98%
396	1.24	1.24	0.00%
537	1.55	1.55	0.00%
540	2.64	2.37	-10.23%
555	1.86	1.80	-3.23%
556	1.55	1.32	-14.84%

Table 16, continued.

c. Stirrup spacing

Element Number	MicasPlus		STAAD III		% Difference	
	Left end	Right end	Left end	Right end	Left end	Right end
	inches	inches	inches	inches		
57	10.70	10.70	10.80	10.80	0.93%	0.93%
60	14.70	14.70	14.80	14.80	0.68%	0.68%
75	14.70	14.70	14.80	14.80	0.68%	0.68%
76	10.70	10.70	10.80	10.80	0.93%	0.93%
217	10.70	10.80	10.80	10.80	0.93%	0.00%
220	14.70	14.70	14.80	14.80	0.68%	0.68%
235	14.70	14.70	14.80	14.80	0.68%	0.68%
236	10.70	10.70	10.80	10.80	0.93%	0.93%
377	10.70	10.80	10.80	10.80	0.93%	0.00%
380	14.80	14.50	14.80	14.50	0.00%	0.00%
395	14.70	14.70	14.80	14.80	0.68%	0.68%
396	10.70	10.70	10.80	10.80	0.93%	0.93%
537	10.70	10.70	10.80	10.80	0.93%	0.93%
540	14.80	14.70	14.80	14.80	0.00%	0.68%
555	14.70	14.70	14.80	14.80	0.68%	0.68%
556	10.70	10.70	10.80	10.80	0.93%	0.93%

d. Column reinforcing

Element Number	MicasPlus	Hand	% Differ.
	Area steel	Area steel	
	in^2	in^2	
1	12.64	12.64	0.00%
6	36.00	36.00	0.00%
8	27.00	27.00	0.00%
16	12.64	12.64	0.00%
161	7.04	7.04	0.00%
166	36.00	36.00	0.00%
168	27.00	27.00	0.00%
176	7.04	7.04	0.00%
321	4.96	4.96	0.00%
326	36.00	36.00	0.00%
328	12.64	12.64	0.00%
336	4.96	4.96	0.00%
481	4.96	4.96	0.00%
486	12.64	12.64	0.00%
488	4.96	4.96	0.00%
496	4.96	4.96	0.00%

and an interior beam located on floor 2, running in the Y direction. Elements 75 and 76 are an interior beam and an exterior beam on floor 2, running in the X direction. Elements 217, 220, 235, and 236 are located in similar positions on the sixth floor. Elements 377, 380, 395, and 396 are located on the tenth floor in the same relative positions. Elements 537, 540, 555, and 556 are located in similar positions on the fourteenth floor. Elements 1, 6, 8, and 16 are columns located on the first level. Element 1 is a corner column. Element 6 is an interior column. Element 8 is an exterior edge column. Element 16 is a corner column opposite to element 1. The remaining elements used for column design comparison are elements in similar locations on the fifth, ninth, and thirteenth floors.

Table 16a compares the top longitudinal reinforcing at each end of the beam chosen by MicasPlus and STAAD III. In most of the 32 cases presented in Table 16a, MicasPlus and STAAD III agree exactly. However, the differences between MicasPlus results and STAAD III are substantial in some cases. Generally, the results agree within 15%. However, there is one case where STAAD III chose about twice as much steel as MicasPlus.

Values for the bottom longitudinal reinforcing at midspan are shown in Table 16b. Disagreement between MicasPlus and STAAD III ranges from 0% to 15%. There is no apparent correlation between the location, size, or loading of the beam to the degree with which MicasPlus and STAAD III agree.

A comparison of beam shear reinforcing at each end of the beam is shown in Table 16c. MicasPlus and STAAD III agree within 1% for every case.

A comparison of MicasPlus column design results and hand calculation results is shown in Table 16d. As with the seven story frame, there are no disagreements.

5.6 Summary of Analysis Results

There is almost no difference between the analysis results produced by MicasPlus and those produced by STAAD III. Support reactions, nodal displacements, and member end forces are the same for all members and nodes. Any difference is most likely round-off error. Regardless, the differences are so small that they are negligible.

5.7 Summary of Design Results

The design results are also very close. The largest differences were seen in steel areas for top and bottom longitudinal reinforcing. The differences ranged from 0% for many cases to 20% for the worst case. Results for shear reinforcing were consistently close. The difference in stirrup spacing recommended by the two programs was never different by more than 0.2 inches or 2.33%. The column design results for MicasPlus agreed closely with those obtained by hand calculations. Results agreed exactly for all but two columns. In those cases, the difference was less than 6%.

Chapter 6

Functionality of MicasPlus and STAAD III

In this chapter the functionality aspects of MicasPlus and STAAD III such as user interface, documentation, and ease of learning are discussed. The term functionality is used to refer to any aspect of the program other than the correctness of the program. Functionality is an important consideration in evaluating a computer program for several reasons. First, it is important for the user to know exactly what parameters the computer is using to model the structure. For example, there must be no confusion as to which material properties, section properties, effective length factors, support conditions, and other parameters, are being used. Second, it is important to be able to quickly verify the results of an analysis or design run.

Graphical presentation of the program results can be useful in verifying support reactions, deflections, or a symmetric response when appropriate. This can be a valuable tool for verifying that the results make sense. Finally, a program should be a tool for designing structures. If the program has poor functionality, it no longer serves as a useful tool. A poorly written program can be a hinderance and can in some cases can even increase the potential for errors.

6.1 Functionality of MicasPlus

6.1.1 User Interface of MicasPlus

MicasPlus has an excellent graphical user interface. The MicasPlus user interface consists of a main graphical drawing area and a series of icons which line the top and side of the drawing area. The icons are used to perform most commands and options and are activated using the mouse or digitizer. Activating an icon will either perform the desired function or display a sub-menu of icons. The majority of the MicasPlus screen is used for the graphical presentation of the structure. The user has many options available for controlling how the structure is displayed. The screen may be used to display one view or it can be subdivided into windows that each display a different view of the structure. There are many pre-defined views including right, left, top, bottom, front, and isometric views. In addition, user defined views may be created. Since MicasPlus runs in conjunction with MicroStation, Intergraph's CAD program, all the functionality and features of MicroStation are also available. These including multiple graphic levels, view manipulation, labeling, dimensioning, and basic model manipulation features.

The most important aspect of the MicasPlus user interface is its use of the graphical interface for creating models. A model may be entered and defined using the graphical interface. For example, a beam may be placed by pointing to its starting and ending locations with the mouse. When this is done, a line representing the beam appears on the screen in the proper location. Information such as member orientation, material properties, and loading may displayed above the beam if desired. All aspects of the model may be defined in a similar manner including model geometry, boundary conditions, and loading.

MicasPlus also uses the graphical interface to display output from an analysis or design run. Deflected shapes of the model under loading may be generated. Elements and nodes can be labeled with nodal displacements, support reactions, member end forces, design results, and many other values.

MicasPlus also provides an alphanumeric interface. This is useful as a supplement to the graphical interface. The alphanumeric interface provides most of the functions that are available in the graphical interface. The alphanumeric interface is useful mainly for viewing output results. However, on some occasions it can be helpful for entering information.

6.1.2 Program Output from MicasPlus

Output from MicasPlus can be either graphical or alphanumeric. The graphical analysis output is useful for quickly scanning the results for obvious errors. For example, if a symmetric structure with only a body load applied does not have a symmetric response, then this can be detected easily with the graphical output. Any quantity can be displayed graphically, including displacements, reactions, and member end forces. The quantity will be displayed at the appropriate locations on the structure. All values are color coded according to magnitude and a key can be displayed if desired. The values can be saved to a graphic level and later printed.

The alphanumeric analysis output is very good. All important information is displayed clearly. The first section of the output clearly defines the model geometry, material properties, support conditions, and loading. The remainder of the output displays tables containing displacements, member end forces, support reactions, and much more. The output is so complete, however, that it can be difficult to locate desired information. The user has total control over what information is printed. For example, the user could print only the support reactions if desired.

The design output is also very good. The graphical design output is similar to the graphical analysis output. The desired values are displayed on the screen with the appropriate member or members. Longitudinal and shear reinforcing for both beams and columns can be viewed in this manner. The alphanumeric design output is excellent. There are three choices in the amount of information given in the design report: short form, medium form, and long form. The short form gives material properties, member cross section, length of the member, and reinforcing bars chosen by MicasPlus. The long form gives much more information such as governing design loads, maximum/minimum loads, Cm, K, and moment magnifier coefficients. This information is helpful in verifying the design results produced by MicasPlus.

6.1.3 Learning to Use MicasPlus

Considering the complexity and capabilities of MicasPlus, it is fairly easy to learn how to use. Included with the documentation are training guides for MicasPlus ModelDraft and MicasPlus Analysis. These training guides contain step by step tutorials which are useful and easy to follow. Although many of the functions available in MicasPlus are not covered in the training guides, the guides give an idea of the general process so that the user has a starting point to learn MicasPlus. One drawback is that there is no training guide available for the MicasPlus Design. MicasPlus Design is fairly simple to use, but a training guide would still be helpful to a new user.

Another way to learn how to use MicasPlus is to take the training course offered by Intergraph. The author did not take this course, but he thinks that it would have greatly decreased the time it took to learn how to use MicasPlus.

6.1.4 MicasPlus Documentation

The documentation provided with MicasPlus is well written, clear, complete, and easy to understand. Every command in MicasPlus is explained thoroughly, often with step by step instructions. The index is complete, which makes it easy to find any desired subject. In addition to the sections which describe how to use the individual commands in MicasPlus, there are sections which describe how to interpret program output as well as sections which describe the theory used by MicasPlus to analyze and design structures. As noted previously, the documentation also includes training guides. These guides are a valuable part of the documentation since without them it would be very difficult to learn how to use

MicasPlus. They are easy to follow.

6.1.5 MicasPlus Analysis

The analysis module of MicasPlus is versatile and easy to use. The structure is entered into MicasPlus using the graphical interface (or alphanumeric if desired) as described above. The next step to analyze a structure is to choose the "Analysis Setup" command. This brings up a tutorial which allows the user to set various analysis parameters. There are three main sections of the tutorial. These are processing, result options, and report selection. The processing section allows the user to control the solution mode (such as static, dynamic, non-linear static, etc.) and the parameters that apply to each solution mode. The result options section allows the user to choose which analysis results will be produced and loaded into the graphics postprocessing database. Among possible choices are: load case results, node displacements, line element end actions, and finite element node forces. The report selection section allows the user to choose which quantities will be printed when a report is created. MicasPlus Analysis gives the user considerable control over the selection of analysis options as well as output options.

MicasPlus analysis has a large finite element library. A library of over forty finite elements allows the user to model plates, shells, or solids. Automatic mesh generation utilities speed the input of finite element grids.

6.1.6 MicasPlus Concrete Design

MicasPlus Concrete Design is simple to use. The general process is as follows: First, a design type, material, and starting section size are assigned to each member. The user can control the maximum member dimensions as well as the increment used to change the section size when necessary. Next, the concrete parts selection tables must be defined for the reinforcing bars, stirrup shapes, column bar patterns, and member shape/dimensions. These tables define what parts, such as reinforcing bar sizes, MicasPlus may use to design the structures. Next, the design tables must be defined. These tables tell MicasPlus which parts may be used for which members and in what order to try the parts. After this, mark groups may be assigned. All members in a mark group share a common design in size and shape, bars used, or both. Finally, all design and code parameters must be set. Among the parameters are: k values, lateral bracing parameters, beam and column code parameters, and strength and deflection checks. For many of the parameters it is possible for the user to either enter the value directly or allow MicasPlus to compute it.

MicasPlus Concrete Design does have several problems. First, it does not allow the user to specify a height to width ratio for the members. Generally, it is economical to use a height to width ratio of approximately 1.5 to 2.0 for beams. MicasPlus will sometimes choose sections that have height to width ratios that are much different from this. Second, MicasPlus still uses ACI-83 instead of ACI-89. Third, it is not possible to determine what strength reduction factor was used by MicasPlus for column design. The documentation implies that is uses a fixed value rather that interpolating between 0.7 and 0.9.

6.2 Functionality of STAAD III

6.2.1 User Interface

The user interface of STAAD III is inferior to that of MicasPlus. There is a graphical interface called STAADPL that is useful, but it is awkward when working with a 3D model. Manipulation of the model in STAADPL is difficult which makes model creation difficult. The STAAD III interface is primarily an alphanumeric interface. Input to the program is through an input file which is created with a line editor provided with STAAD III, or any line editor. This can be a long and tedious process especially for a large structure with varying geometry and loading conditions.

The graphical postprocessing capabilities of STAAD III are good. These capabilities are provided by STAADPL. STAADPL will draw deflected shapes of the structure, draw different views, create DXF files for use with other graphics and CAD programs, draw shear and moment diagrams, and perform other basic functions. Also included with STAAD III is STAAD-DRAFT, which is a useful but somewhat limited drafting utility.

6.2.2 Output from STAAD III

The output generated by STAAD III is excellent. It is compact and easy to read and understand. Input parameter and results are printed in tables. The user has considerable control over which results are printed. Due to the layout of the tables, STAAD III output uses about half as much paper as MicasPlus does for the same amount of information.

6.2.3 Learning to Use STAAD III

It is simple to learn how to use STAAD III. The documentation contains many examples that are explained well and are easy to follow. A new user can analyze a simple structure within hours of first opening the package.

6.2.4 STAAD III Documentation

The documentation provided with STAAD III leaves a lot to be desired. There are many deficiencies in the documentation: Nowhere in the documentation is there any information as to the methods used to design concrete members. Many commands are explained in an ambiguous manner. For many of the commands, there is no specification as to whether the command is required or optional. There is no index included with the documentation, which makes locating a subject difficult. The documentation is often useless when attempting to troubleshoot an input file.

6.2.5 Analysis

The analysis module of STAAD III is good. Once the structure is entered, only a few lines are needed in the input file to perform the analysis and specify the desired output.

STAAD III has limited finite element capabilities compared to MicasPlus. Plate and shell structures may be modeled using STAAD III. However, only three and four noded elements are available, which makes it difficult to model structures with curved surfaces.

6.2.6 Design

The functionality of the STAAD III concrete design module is poor. It is severely limited by the fact that the program will not choose member sizes. It will only select reinforcement for members. For columns, STAAD III designs only longitudinal reinforcement, and does not design the ties. Also, only square bar patterns are available. Therefore, the number of bars used is always a multiple of four. This tends to yield conservative results. Also, if a column goes into tension for a given load combination, that load combination is ignored.

Two methods are available for accounting for secondary moments in columns. In the first method a user-specified moment magnification factor can be applied to the members. The second method is a P-delta analysis whereby the program iterates a user-specified number of times to recalculate forces due to secondary moments. This option has a drawback that renders this option practically useless: load combinations cannot be specified. Therefore, for each new load combination, the user must re-enter the loads as a primary load case. For a large structure, this can be a very time consuming task.

Another problem with the concrete design is that neither the program nor the documentation provides any information about what values of K, Cm, or most other design parameters, are being use. The only value provided by STAAD III is the strength reduction factor.

6.3 Conclusions and General Comments

MicasPlus is superior to STAAD III in several respects. First, the user interface of MicasPlus is much easier to use than that of STAAD III. Due to this, a structure can be modeled more quickly in MicasPlus than in STAAD III. Second, the documentation provided with MicasPlus is complete, clear, easy to use, and accurate. The documentation provided with STAAD III is incomplete and often ambiguous. Third, the concrete design capabilities of MicasPlus are much more extensive than those of STAAD III. MicasPlus allows the user to control many design parameters such as k values, Cm values, and moment magnifiers. STAAD III concrete design is somewhat limited. For example, columns are always designed with equal reinforcement on all sides. Although MicasPlus is superior to STAAD III, it must be noted that MicasPlus is much more expensive than STAAD III. Currently, the bare minimum hardware and software requirements for MicasPlus cost approximately $25,000. The STAAD III software cost approximately $3,400. Most companies already have the hardware required to run STAAD III.

Following is a list of general comments concerning MicasPlus and STAAD III. Table 17 is a summary of the results of the functionality comparison between MicasPlus and STAAD III.

General Comments

1. MicasPlus version 4.0.1 was used.
2. STAAD III version 15.0 was used.
3. MicasPlus runs on a workstation, while STAAD III will run on a PC.
4. MicasPlus is much more expensive than STAAD III.
5. STAAD III does not provide the user with K values, Cm values, and other important values needed for verification of design.
6. When designing columns, STAAD III ignores load cases and combinations for which the column is in tension.
7. STAAD III will not choose member sizes.
8. STAAD III column design is incorrect.
9. MicasPlus allows the use of compression steel or multiple layers of reinforcing for concrete beam design.
10. There is no "undo" option in MicasPlus for undoing an action just performed.
11. Concrete design capabilities of MicasPlus are more extensive than those of STAAD III.

Table 17. Summary of functionality comparison results

	MicasPlus	STAAD III
User Interface	excellent	fair
Output	good	excellent
Learning	fairly easy	very easy
Documentation	excellent	poor
Analysis	excellent	fair
Design	excellent	limited

Chapter 7

Summary and Conclusions

The objective of this study was to compare the concrete analysis and design capabilities of MicasPlus and STAAD III. This was done by modeling four structures in both MicasPlus and STAAD III and comparing the results. The four structures considered included a continuous beam, a two story frame, a seven story frame, and a fourteen story frame. Support reactions, nodal displacements and end forces were tabulated in order to compare the analysis results given by the two programs. Positive, negative, and shear reinforcing results were tabulated in order to compare the design capabilities of the programs.

It was determined that the analysis results produced by MicasPlus and STAAD III agree very well for all structures and loading conditions. The results for the concrete design did not agree as closely, but it was determined that this is due to a difference in the methods used for selecting bars. The results obtained from both programs for the design of beams were in close agreement. The area of steel recommended by the two programs for longitudinal reinforcement agreed within 15%. The stirrup spacing recommended by the two programs never differed by more than 0.2 inches. The results produced by MicasPlus design of columns were not compared with STAAD III since it was determined that STAAD III does not give correct results for the design of columns. However, the MicasPlus column design results were verified by hand calculations. There was a close agreement between the results obtained from MicasPlus and those obtained by hand calculations. Whenever the design results from MicasPlus differed with STAAD III or hand calculations, MicasPlus tended to be more conservative due to the process by which it chooses reinforcement.

The functionality of MicasPlus and STAAD III was compared. Some topics that were discussed include: user interface, documentation, output, learning to use the software, analysis, and design. It was concluded that MicasPlus has superior functionality to STAAD III. This is mainly due to the graphical user interface on MicasPlus. The graphical user interface on MicasPlus is simple to use, versatile, and convenient. A structure can be modeled, analyzed, and designed using the graphical interface. STAAD III has a graphical user interface with many of the same capabilities of MicasPlus, but it is awkward to use. The author found the alphanumeric interface of STAAD III easier to use than its graphical interface. Although the functionality of MicasPlus was determined to be superior to that of STAAD III, it should be noted that MicasPlus is much more expensive than STAAD III.

There are two aspects of MicasPlus and STAAD III that are important topics for future study. First, a study of the finite element analysis capabilities of each program should be performed. Plate, shell, and solid finite elements should be considered in this study. Second, a study of the slenderness effects on column design should be performed. MicasPlus and STAAD III each provide several methods for considering slenderness effects in columns.

Bibliography

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

Machover, H., "How Good is PC CAD?", *Computer-Aided Engineering*, Vol. 8, No. 11, Nov. 1989, pp. 26-28, 30-31, 34.5.

MicasPlus, Intergraph Corporation, One Madison Industrial Park, Huntsville, Alabama.

Nadella, Navin Kumar. "*Evaluation of ISDS Software.*" Virginia Polytechnic Institute and State University, 1991.

Priest, J., *Engineering Design for Producibility and Reliability*, New York, Marcel Decker, Inc., 1988.

STAAD III / ISDS, Research Engineers, 1570 N. Batavia Street, Orange, California.2.

UBC. *Uniform Building Code*. Whittier, CA: International Conference of Building Officials, 1988.

Wallace, D., and Fujii, R. U., "Software Verification and Validation: and Overview", *IEEE Software (USA)*, Vol. 6, No. 3, May, 1989, pp. 10-17.

White, Maurice W. "*Verification and Evaluation of Structural Analysis and Design Software.*" Virginia Polytechnic Institute and State University, 1991.

Appendix A

Location of Nodes and Members

for Continuous Beam

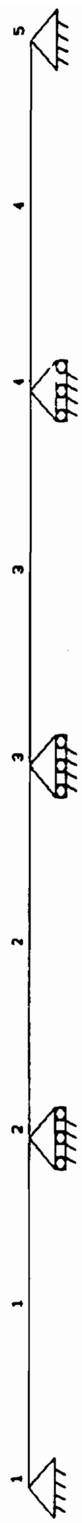


Figure A-1. Location of Nodes and Members for Continuous Beam.

Location of Nodes and Members

for Two Story Frame

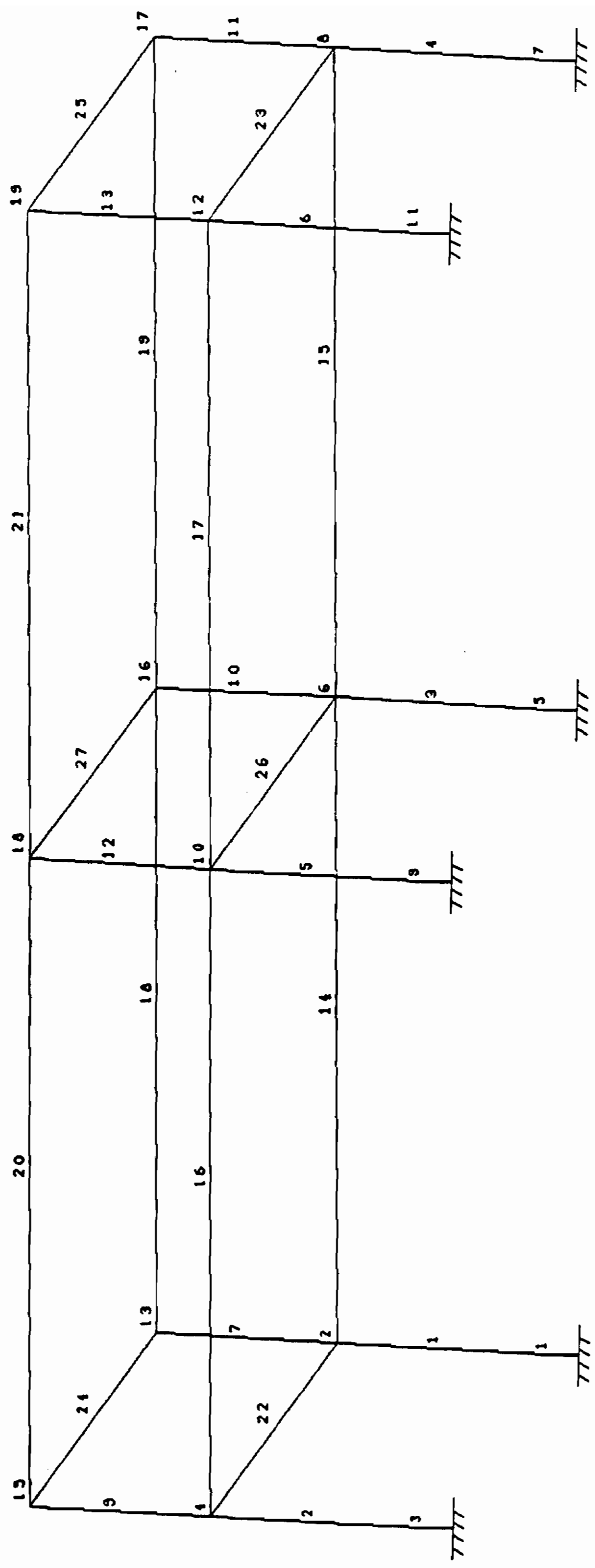


Figure A-2. Location of Nodes and Members for Two Story Frame.

Location of Nodes and Members

for Seven Story Frame

Figure A-3. Location of Nodes and Members for Seven Story Frame

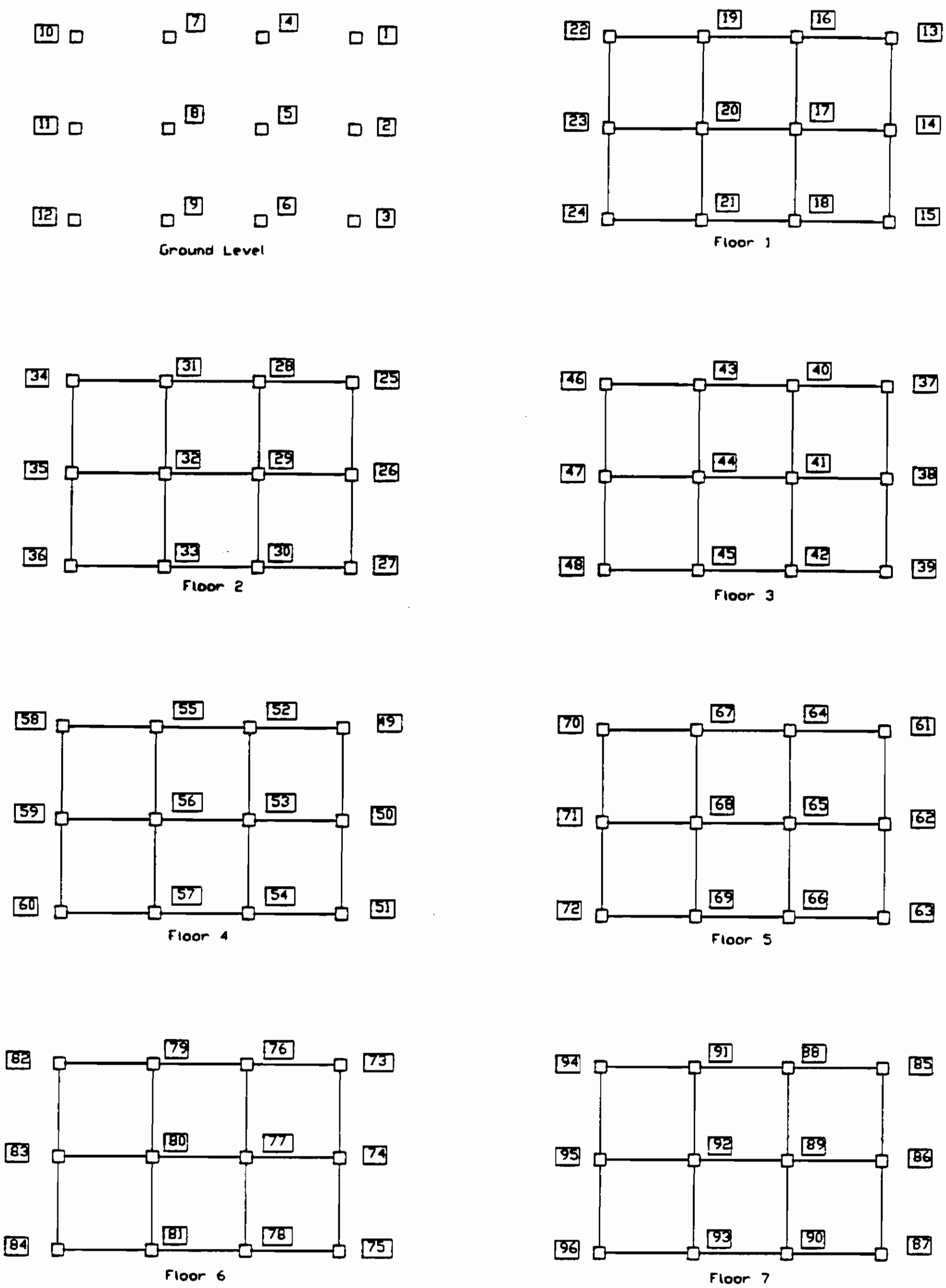
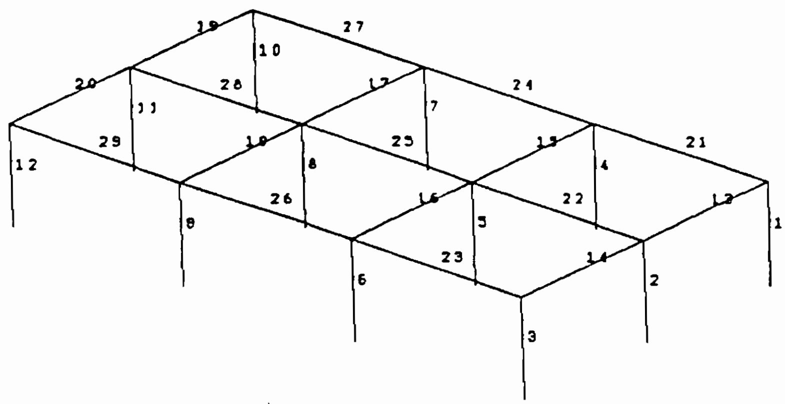
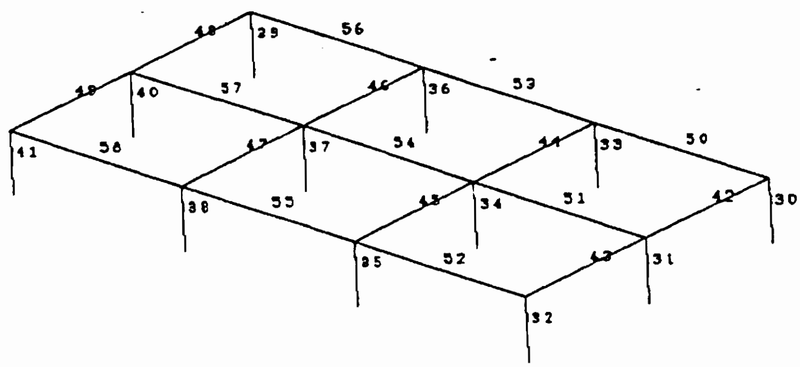


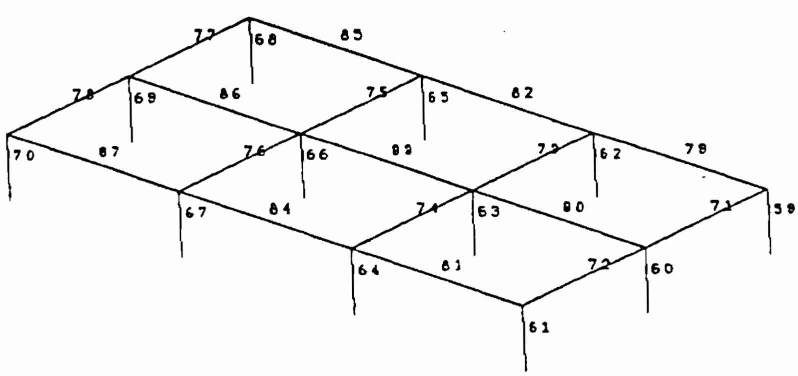
Figure A-3, continued.



Floor 1

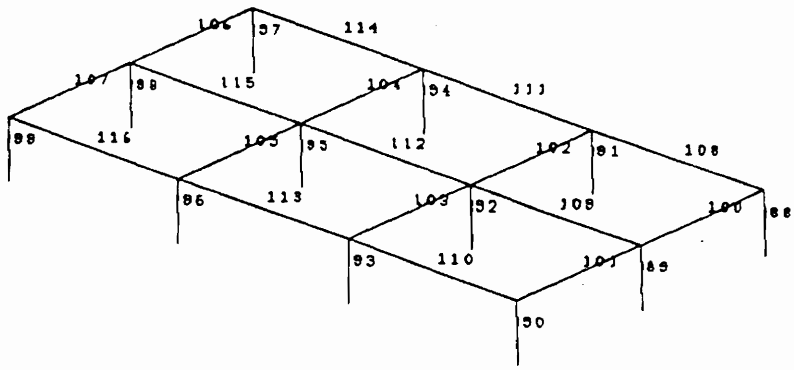


Floor 2

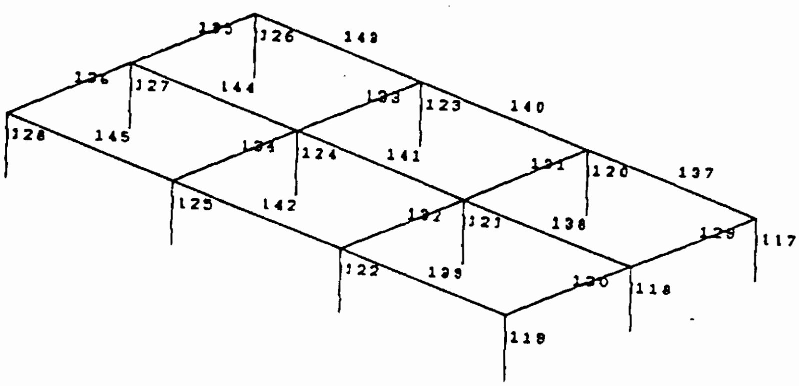


Floor 3

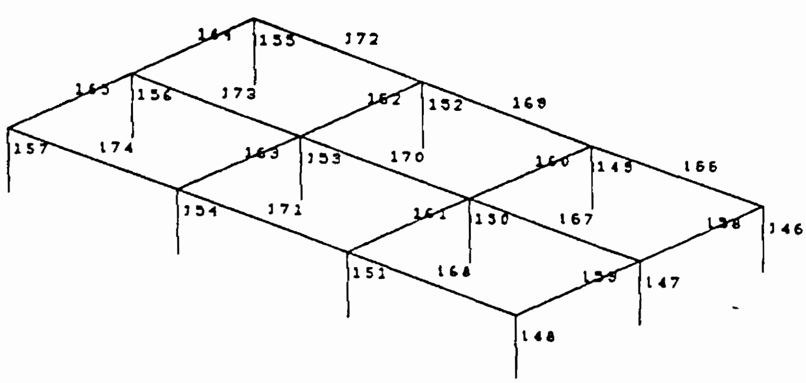
Figure A-3, continued.



Floor 4

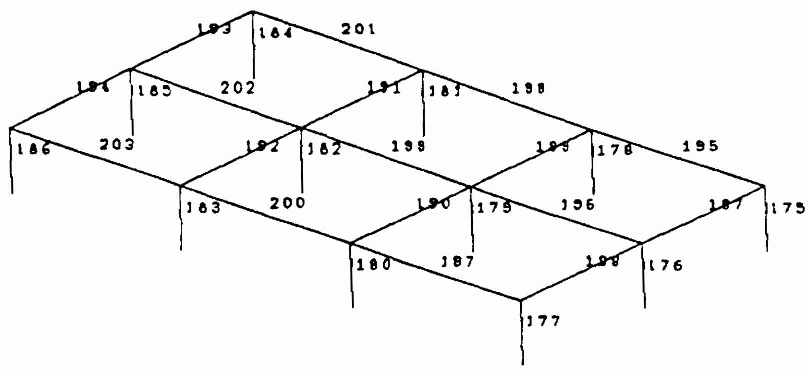


Floor 5



Floor 6

Figure A-3, continued.



Floor 7

Location of Nodes and Members

for Fourteen Story Frame

Figure A-4. Location of Nodes and Members for Fourteen Story Frame.

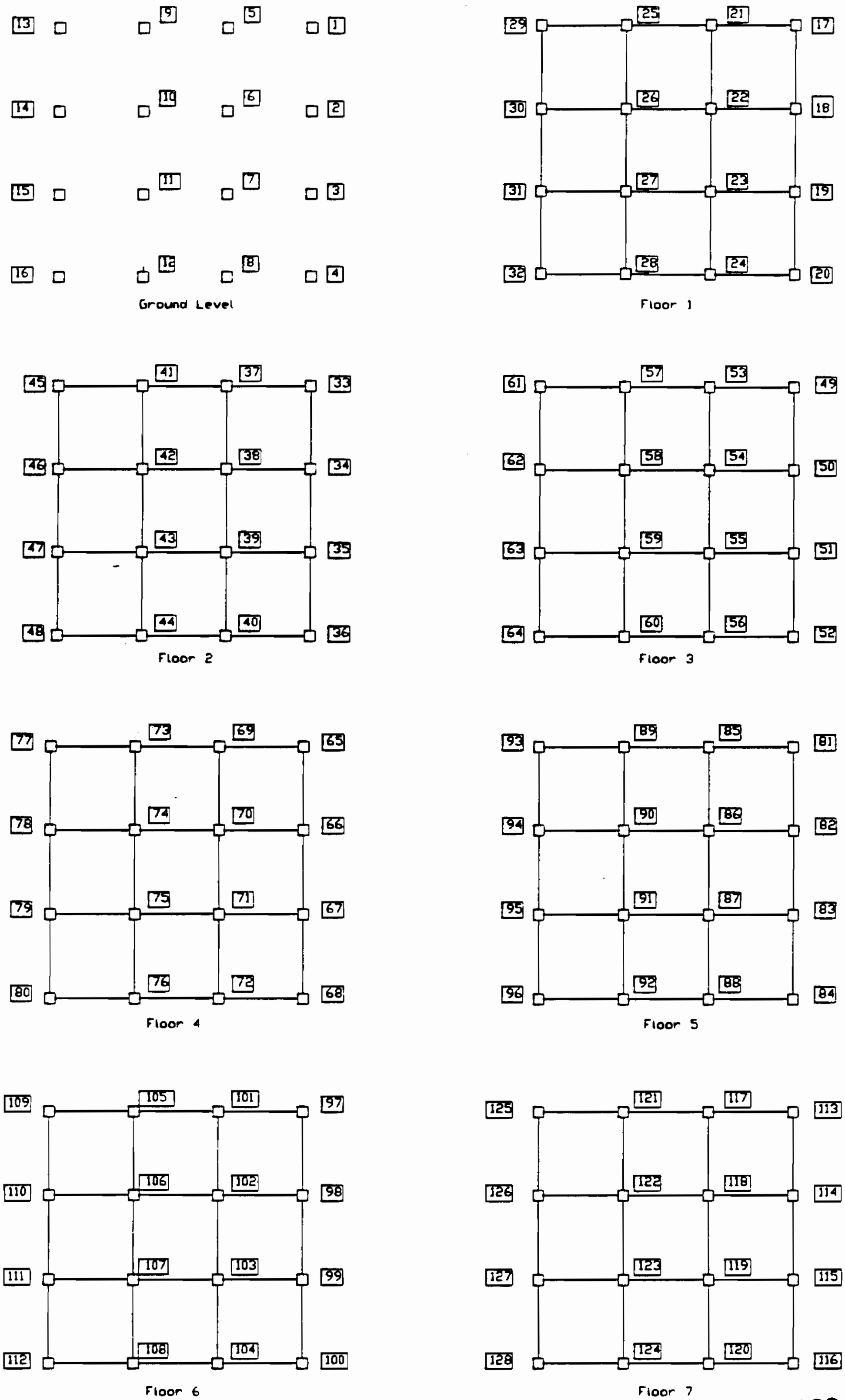


Figure A-4, continued.

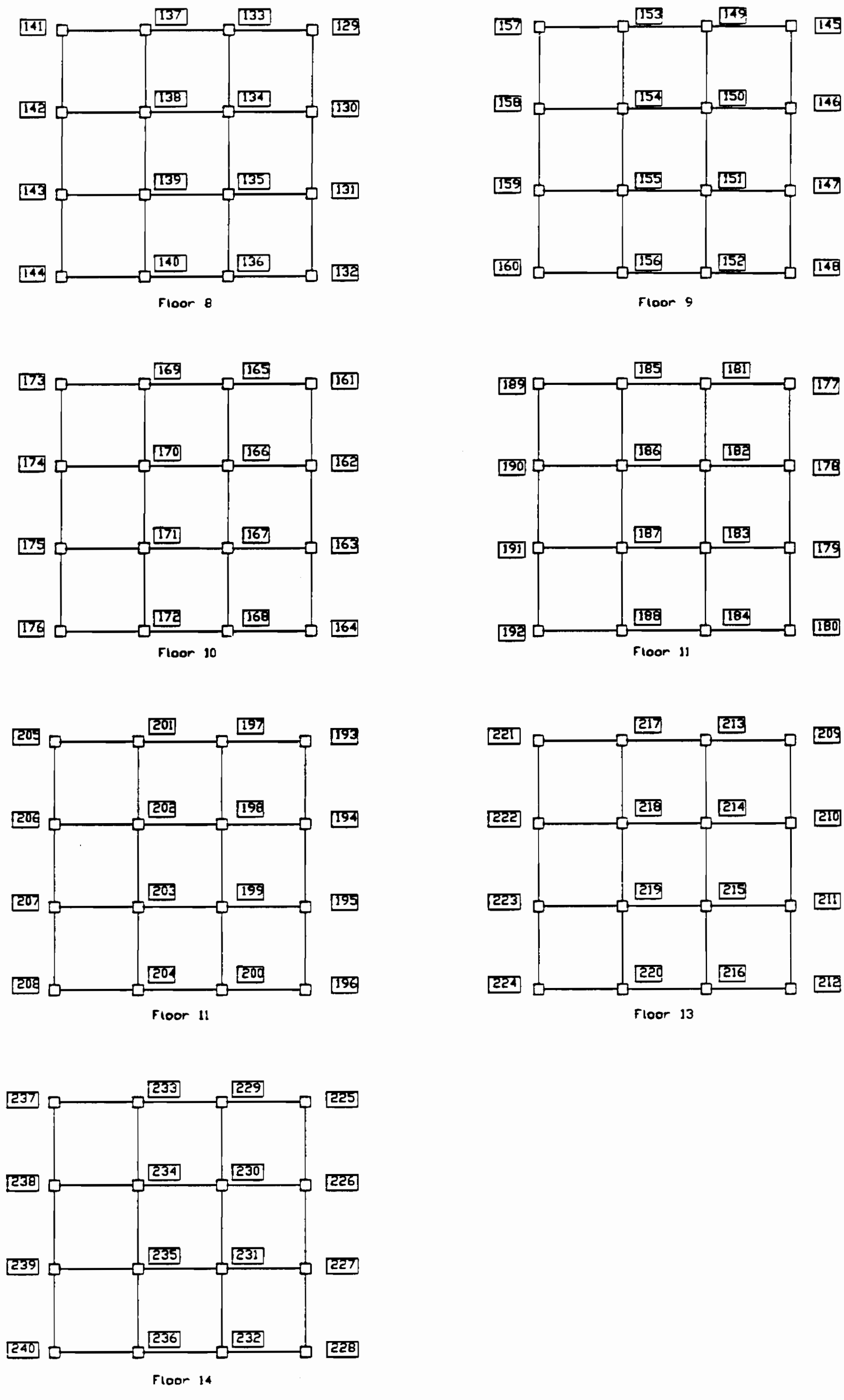
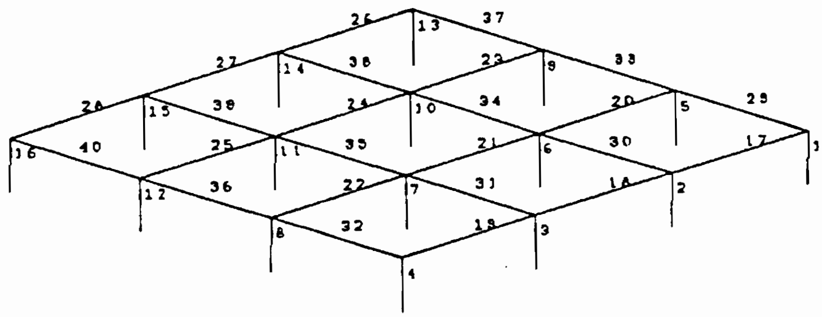
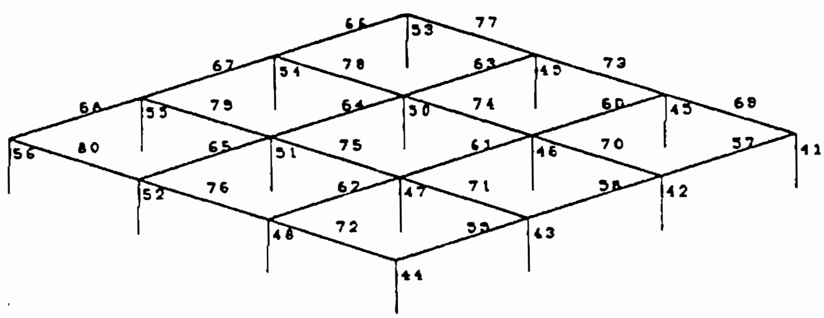


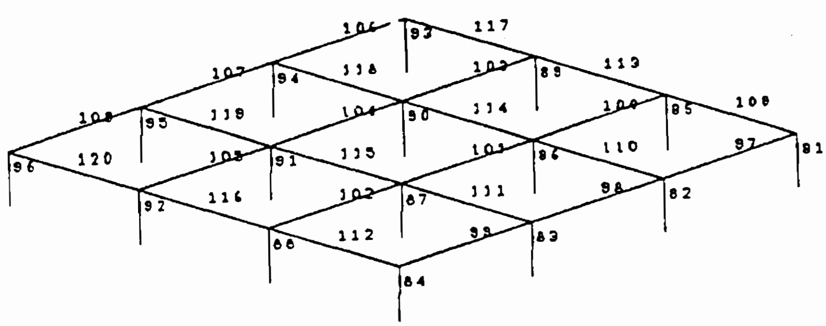
Figure A-4, continued.



Floor 1

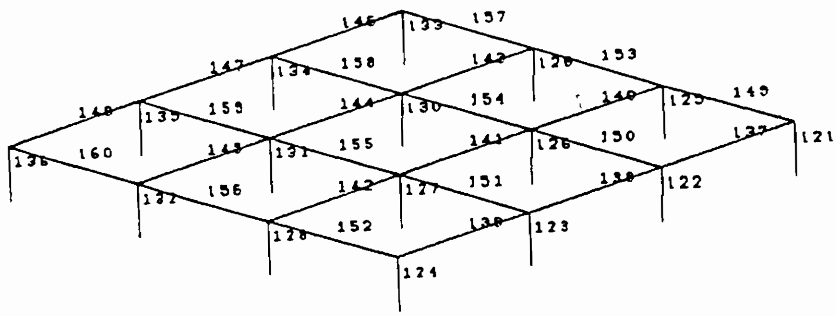


Floor 2

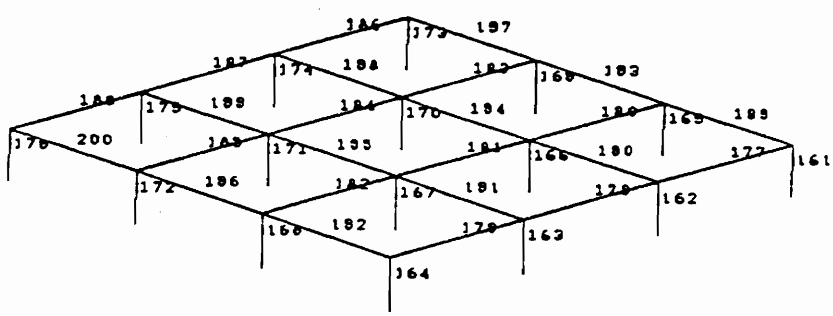


Floor 3

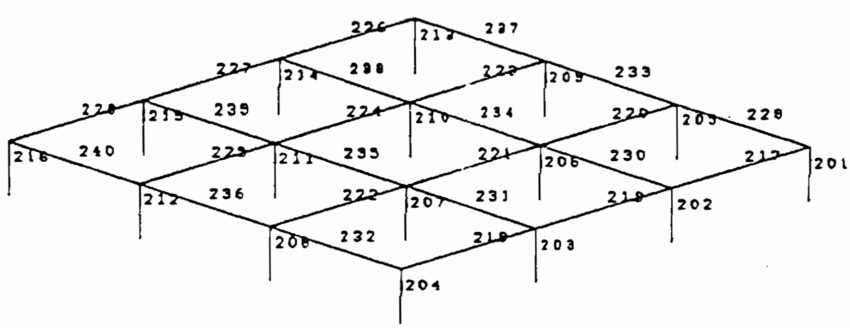
Figure A-4, continued.



Floor 4

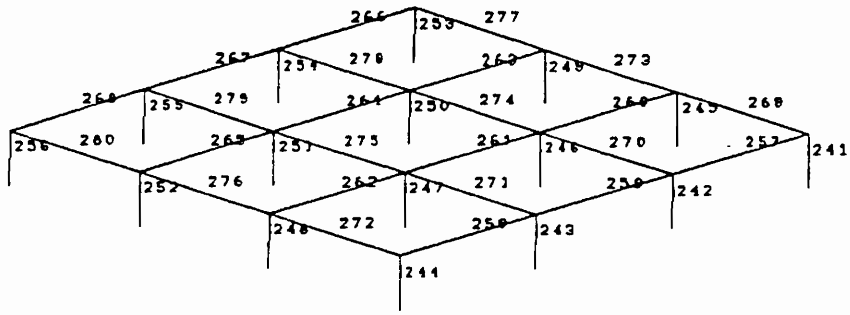


Floor 5

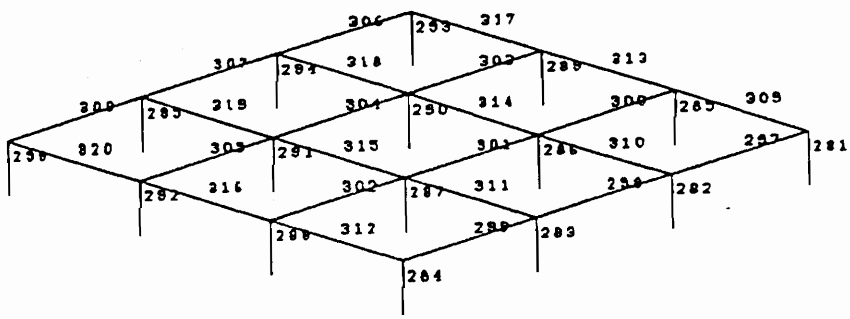


Floor 6

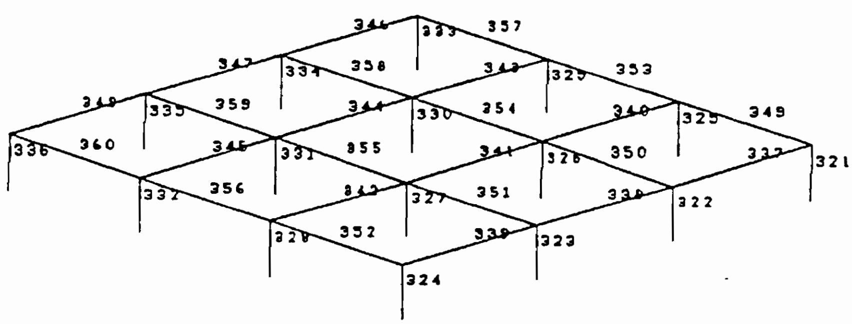
Figure A-4, continued.



Floor 7

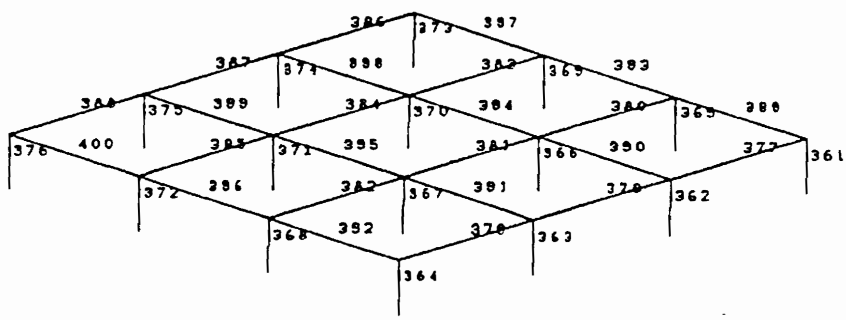


Floor 8

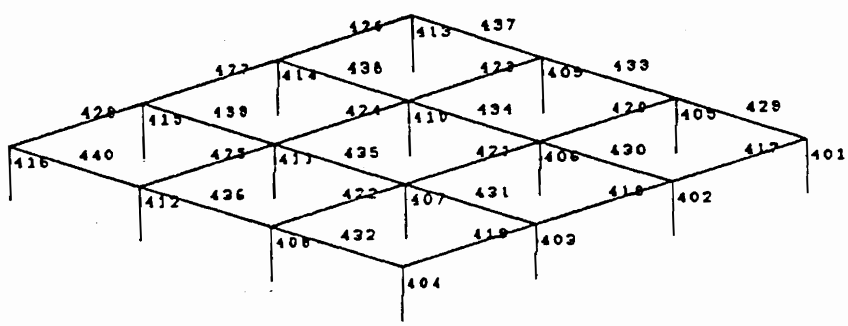


Floor 9

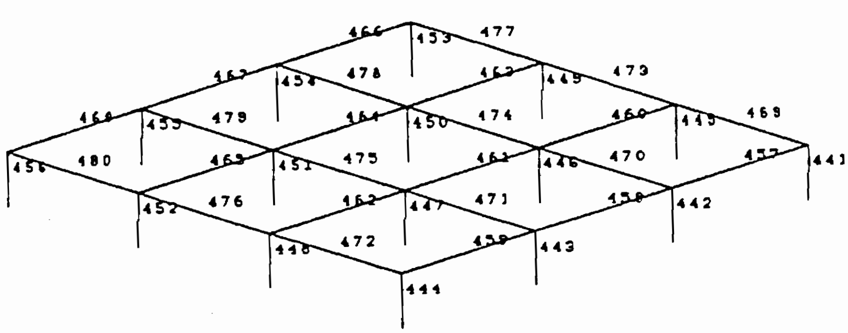
Figure A-4, continued.



Floor 10

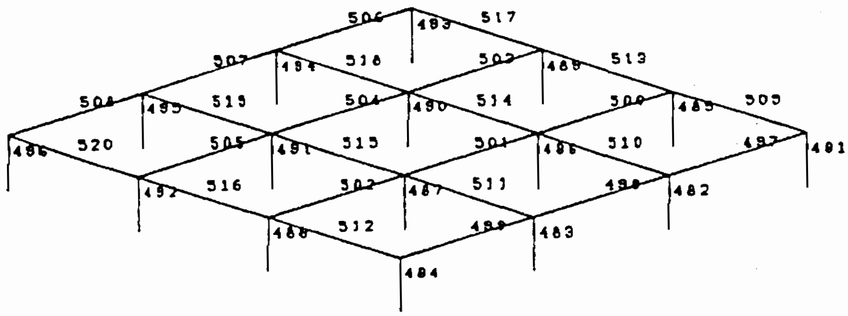


Floor 11

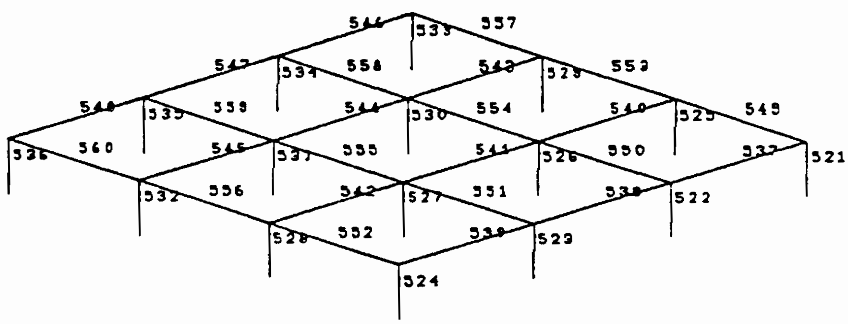


Floor 12

Figure A-4, continued.



Floor 13



Floor 14

Appendix B

Sample Calculations

Sample Calculation No. 1

Compression plus bending about one axis

Model: Seven story frame

Member: Column No. 4

Material Properties: fc'= 4 ksi
fy = 60 ksi

Critical Loading:

Case 1: Pu = 730 kips
Mu = 82 ft-kips

Case 2: Pu = 922 kips
Mu = 0 kips

Step 1: Case 1. Determine nominal loads and eccentricity. For tied columns, Φ=0.70

$$Pn=\frac{Pu}{\phi}=\frac{730}{0.7}=1043\ kips$$

$$Mn=\frac{Mu}{\phi}=\frac{82}{0.7}=117\ ft\text{-}kips$$

$$e=\frac{Mn}{Pn}=\frac{117\times 12}{1043}=1.35\ inches$$

Step 2: Assume a 16-inch square column.

Step 3: Calculate parameters for interaction diagram.

$$\gamma=\frac{16-2(2.5)}{16}=0.69$$

$$\frac{\phi Pn}{Ag}=\frac{Pu}{Ag}=\frac{730}{(16\times 16)}=2.85\ ksi$$

$$\frac{e}{h}=\frac{1.35}{16}=0.084$$

Step 4: Determine *Pg* from interaction diagram.

$$Pg = 0.030$$

Step 5: Determine required steel area for load case No. 1.

$$A_{st} = PgAg = 0.030\,(16)^2 = 7.68\ in^2$$

Step 6: Case 2. Determine required steel area for load case No. 2.

$$\phi Pn = 0.80\,(0.70)\,[0.85\,fc'\,(Ag - A_{st}) + fyA_{st}]$$

$$\phi Pn = 922\ kips$$

$$fc' = 4\ ksi$$

$$fy = 60\ ksi$$

$$Ag = 256\ in^2$$

Solving for Ast, Ast = 13.72 *in*^2

Step 7: Choose reinforcing.

$$Ast\,(required) = max \begin{cases} 7.68\ in^2 \\ 13.71\ in^2 \end{cases} = 13.72\ in^2$$

Using MicasPlus "Bar Precedence" option, select:

8 #14 *bars* (*Ast* = 18.0 *in*^2)

Use a 16-inch square column with 8 #14 bars. Provide equal number of bars each side.

Sample Calculation No. 2:

Compression plus biaxial bending

Model: Fourteen story frame

Member: Column No. 328

Material Properties: fc' = 4 ksi
fy = 60 ksi

Critical Loading:

Case 1: Pu = 1147 kips
Mux = 115 ft-kips
Muy = 26 ft-kips

Step 1: Case 1. Determine nominal loads. For tied columns, $\phi = 0.70$

$$Pn = \frac{Pu}{\phi} = \frac{1147}{0.7} = 1639 \, kips$$

$$Mnx = \frac{115}{0.7} = 164 \, ft\text{-}kips$$

$$Mny = \frac{26}{0.7} = 37.1 \, ft\text{-}kips$$

Step 2: Assume $\beta = 0.65$
Assume a 20-inch square column.

Step 3: Determine an equivalent uniaxial moment strength Mnox.

$$\frac{Mny}{Mnx} = \frac{37}{164} = 0.225 \text{ is less than } \frac{b}{h} = 1.0 \text{ (square column)}$$

Therefore,

$$Mnox \approx Mny\left(\frac{h}{b}\right)\left(\frac{1\text{-}\beta}{\beta}\right) + Mnx = 37.1\,(1)\left(\frac{1\text{-}0.65}{0.65}\right) + 164 = 184 \, ft\text{-}kips$$

Step 4: Determine required reinforcing based on:

ϕPn = 1147 kips

ϕMn = 0.7 (184) = 129 ft-kips

20-inch square column

$$\rho = \frac{20\text{-}2(2.5)}{20} = 0.75$$

$$\frac{\phi Pn}{Ag} = \frac{1147}{20 \times 20} = 2.9\ ksi$$

$$\frac{\phi Mn}{Agh} = \frac{129 \times 12}{20 \times 20 \times 20} = 0.19\ ksi$$

From interaction diagram, read

Ast = $\rho_g Ag$ = $0.031(20)^2$ = 12.40 in^2

Using MicasPlus "Pattern Precidence" option, select:

16 #8 bars (Ast = 12.64 in^2)

Step 5: Check selected section using PCA Load Contour Method.

a. Determine Po, Mnox, Mnoy.

$$Po = 0.85fc'\,(Ag\text{-}Ast) + (Ast)\,(fy)$$
$$= 0.85(4)\,(400\text{-}12.64) + 12.64\,(60)$$
$$= 2075\ kips$$

Due to symmetry, Mnox = Mnoy

$$\frac{\phi Pn}{Ag} = \frac{1147}{20 \times 20} = 2.87\ ksi$$

$$\rho g = \frac{Ast}{Ag} = \frac{12.64}{20 \times 20} = 0.032$$

From interaction diagram, with $\frac{\phi Pn}{Ag} = 2.87$ and

b. Determine actual β

$$\rho g = 0.032, \text{ read } \frac{\phi Mnox}{Agh} = 0.28\ ksi$$

$$\text{Therefore, } Mnox = \frac{0.28\,(20)^3}{12\,(0.7)} = 267\ ft\text{-}kips$$

$$\frac{Pn}{Po} = \frac{1639}{2075} = 0.79$$

$$w = \frac{\rho fy}{fc'} = \frac{0.032\,(60)}{4} = 0.48$$

From graph a biaxial bending design constants, read:

β = 0.56

c. Using above values, evaluate the following interaction equation.

$$\left(\frac{Mnx}{Mnox}\right)^{\left(\frac{\log 0.5}{\log \beta}\right)} + \left(\frac{Mny}{Mnoy}\right)^{\left(\frac{\log 0.5}{\log \beta}\right)} \leq 1.0$$

$$\log 0.5 = -0.3$$
$$\log \beta = \log 0.66 = -0.180$$
$$\frac{\log 0.5}{\log \beta} = 1.67$$

$$\left(\frac{164}{267}\right)^{1.67} + \left(\frac{37}{267}\right)^{1.67} = 0.48 < 1.0\ o.k.$$

Use a 20-inch square column with 16 8# bars. Provide equal number of bars each side.

Vita

Andrew Betaque graduated from Virginia Tech in 1991 with a Bachelor of Science degree in Civil Engineering. He then continued his studies at Virginia Tech and earned a Master of Science degree in Civil Engineering in December, 1992. He currently works as a structural design engineer in Little Rock, Arkansas.

Andrew D. Betaque