A Preprocessor for
the Analysis of Space Frames
in the Microsoft Windows Environment

by

Zhijun Zheng

Report submitted to the Faculty of
Virginia Polytechnic Institute and State University
in partial fulfillment of the requirements for the degree of

Master of Engineering

in

Civil Engineering

APPROVED

Kamal B Rojiani

Kamal B. Rojiani, Chairman

W Samuel Easterling

W. Samuel Easterling

A. O. Abatan

March, 1993
Blacksburg, Virginia

A Preprocessor for

the Analysis of Space Frames

in the Microsoft Windows Environment

by

Zhijun Zheng

Kamal B. Rojiani, Chairman

Civil Engineering

(ABSTRACT)

A preprocessor for the analysis of space frame structures is developed. The preprocessor is designed to run in the Microsoft Windows graphical environment which is widely available on personal computers. The program is written in the Visual Basic programming language.

Visual Basic is an object-oriented programming language which provides a powerful way to create applications for the Windows environment. It is an ideal programming language for developing small or medium size Windows applications.

The basic features of the preprocessor include the ability to read and write data files, entry and review of structural data, and a context sensitive on-line help system. In addition, the program provides a user friendly interface and adopts standard Windows user interface guidelines. A sophisticated automatic data checking mechanism reduces data entry errors to a minimum. The preprocessor also features three dimensional graphics which can display the structural geometry and applied loads.

Acknowledgment

I would like to thank my advisor Dr. Kamal B. Rojiani for his encouragement, guidance and support during my education in Virginia Tech, and his continuous help on this project.

I would like to thank the other members of my advisory committee Dr. W. S. Easterling and Dr. A. O. Abatan, for their help in my course work and for their support of this project.

Special thanks go to my sister Dr. Limin Zheng and her husband Dr. Keying Ye. It would not be possible for me to have the opportunity to undertake graduate study at Virginia Tech without their encouragement and financial support. I would also like to express my gratitude for my parents, for their love and support throughout my education.

TABLE OF CONTENTS

Table of Contents

LIST OF FIGURES

List of Tables

CHAPTER ONE
INTRODUCTION

1.1 COMPUTING ENVIRONMENT

Computers have been used in structural analysis since the mid-1950s. Their potential for carrying out the more laborious analysis tasks was quickly appreciated. There has been steady development of methods designed to utilize the speed and numerical processing capabilities of computers. During the past four decades, there have been tremendous enhancements in both hardware and software. In recent years, computers have become a part of daily life. The widespread availability of computers, particular personal computers, which appeared only ten years ago, now enables most structural analysis tasks to be carried out by machines. Personal computer based structural analysis programs are well suited for the analysis of small and medium size structural models. Also, PC based programs are ideal for introducing the basic concepts of computer structural analysis to beginners because they are easy to understand and easy to use compared to large commercial programs developed for mainframes and minicomputers. Another advantage of using PC based structural analysis software is the cost efficiency.

A typical structural analysis program consists of three components: a preprocessor, a processor and a post processor. Through the preprocessor, the user enters information that describes the geometry of the structure. This information includes joint coordinates, member incidences, geometric properties, material properties and applied loads. This information is typically saved in a data file. The processor reads this data file and performs the analysis to determine the response of the structure to the applied loads. The results of the analysis include support reactions, joint displacements and member end forces. The processor writes the results of the analysis to an output data file. This output data file is used by the post processor to display the results either graphically or in tabular form so that they are easy to understand.

When performing the analysis of a structure, considerable effort is required in entering the computer model of the structure. A large amount of information such as joint coordinates, member incidences, geometric properties, material properties and loading has to be entered. Most programs for analyzing structures employ a character-based user interface. Input to these programs is typically through an input file consisting of a series of instructions to the analysis program. This input file is a text file. It consists of a series of commands and normally is created by using a text editor. Considerable effort is required in creating this file. The user has to memorize the syntax for each command. Also, since there is no visual feed back, the potential for making errors is very high.

The preprocessor described in this report takes advantage of the Windows graphical user interface and provides an easy user-friendly environment for creating this input file. The preprocessor is entirely menu driven and has extensive on-line help so the user does not have to spend time reading through a user's manual for the correct syntax. The program is developed as an event-driven program which means that the user is in complete control of the order of tasks that are to be performed, instead of the program dictating the order of operations to user. The program also has error checking capabilities for reducing data entry errors and displays the structure graphically.

Because Windows provides a series of common user access procedures, the preprocessor is very easy to use. All Windows programs make use of a common user interface such as drop down menus, dialog boxes, radio buttons and command buttons which reduces the learning curve enormously.

Introduction

From a programmer's viewpoint, Windows programs are very difficult to develop. Until recently, programming for Windows was a very tedious time consuming task and required writing many lines of code using a procedural language such as C. Because Windows is a message driven system, tremendous effort was required to develop applications that contain many dialog boxes in that the application had to respond to numerous messages. Also, the learning curve for Windows program development was very steep. It would take months for the programmer to read and understand all the intricacies of Windows programming.

Microsoft Visual Basic provides a good solution to the enormity of Windows programming. With Visual Basic, the programmer can create Windows resources with just a click of a mouse. Visual Basic allows the programmer to focus on the function of the program without having to worry too much about the actual coding.

1.2 OBJECTIVE

The objective of the project was to develop a user-friendly preprocessor for the analysis of space frames in the Microsoft Windows Environment. The preprocessor creates a data file with a format accepted by a three-dimensional structural analysis program that is currently under development. (Slobodan and Rojiani, 1993) The preprocessor can also be used as a valuable teaching tool for introducing the basic concepts of computer analysis of structures to beginners.

1.3 PREPROCESSOR FEATURES

The preprocessor is a Windows based application that serves as a front end to a three-dimensional structural analysis program. The preprocessor has the following features:

- Standard Windows controls are adopted.

- All data entry forms can be accessed by logical path at run time.
- A data checking mechanism verifies the accuracy of the data automatically before it is saved to a data file.
- Three-dimensional graphics output for viewing the geometry of the structure.
- Capabilities for quickly reviewing input data.
- Provision for both English and SI units.
- Automatic generation of member incidences and joint coordinates.
- On-line help for all the controls and data entry forms used in the preprocessor
- The preprocessor can be used to create a new model or modify an existing model.

1.4 OVERVIEW

Chapter 2 contains an introduction to the Visual Basic programming environment. It also presents a brief history of the BASIC programming language. Chapter 3 provides a detailed description of the structure of the preprocessor. It also provides an explanation of all the menu items and dialog boxes (also called forms). Chapter 4 summarizes the capabilities and limitations of the preprocessor. It also contains suggestion for future modification to the preprocessor. The user's manual in Appendix A provides additional information on using the preprocessor. It presents an explanation of basic Windows controls and describes all of the data entry forms. It also provides a tutorial on the use of the preprocessor and the steps needed for entering data in the various dialog boxes. Appendix B contains a listing of the contents of the data files produced by the preprocessor for four types of structures, a plane truss, a space truss, a plane frame and a space frame. An explanation of the format of the output data file that is created by the preprocessor is also provided in Appendix B.

CHAPTER TWO

THE VISUAL BASIC PROGRAMMING LANGUAGE

2.1 A BRIEF HISTORY OF BASIC

The BASIC programming language was created in 1963 by Professor John Kemeny and Thomas Kurtz at Dartmouth College. It was designed for the purpose of teaching the concepts of programming. It was an excellent computer language for beginners because it was easy to understand and follow. However, because BASIC was an interpreted language, rather than a compiled language, it was slow and inefficient for application development.

In the mid-1970s, GW-BASIC was introduced by Microsoft. It was an excellent tool for performing simple tasks. However, it was slow because it was an interpreted language. Also, the source code had to be provided to run the program. No professional software developer would consider writing marketable software in BASIC.

In 1982, Microsoft developed the QuickBasic programming language. QuickBasic transformed BASIC into a serious development language for the MS-DOS environment. It combined the power and speed of a compiled language, with the interactive, productive nature of interpreted BASIC. Microsoft added a number of features to the language, such as local variables, true subroutine calls, multiline functions, support for modules, all of which are essential for developing modular well-structured programs.

With the introduction of Windows 3.0 in 1990, Microsoft provided a powerful, user friendly graphical environment where all programs have a similar interface. Visual Basic takes advantage of the features provided by both Windows and QuickBasic. Visual Basic is a giant revolution in the way one develops programs.

QuickBasic and Visual Basic programs can be run interactively in interpreted mode, or compiled into stand-alone executable programs.

2.2 WHAT IS VISUAL BASIC?

Visual Basic is an object-oriented programming environment specially designed for creating applications for the Microsoft Windows environment. Visual Basic provides an easy way for programmers to develop Windows applications using interface objects such as command buttons, radio buttons, check boxes, pull down menus, scroll bars and list boxes.

A Visual Basic application is developed by creating one or more forms and placing the various interface elements on these forms. The developer then writes subroutines to handle the various events that relate to the particular interface object. For example, if a command button is pressed, then the developer may want to write a subroutine to respond to this event. Visual Basic defines a series of default subroutines that correspond to each type of control.

Visual Basic offers a toolbox window for the programmer to select controls and place them in a form. Details of the controls in the tool kit are given in Appendix A. The tool kit includes the following items: (see Figure 2.1)

- Label
- Frame
- Check box
- Combo box
- Horizontal scroll bar
- Timer
- Directory list box

- Picture box
- Text box
- Command button
- Option button
- List box
- Vertical scroll bar
- Drive list box
- File list box

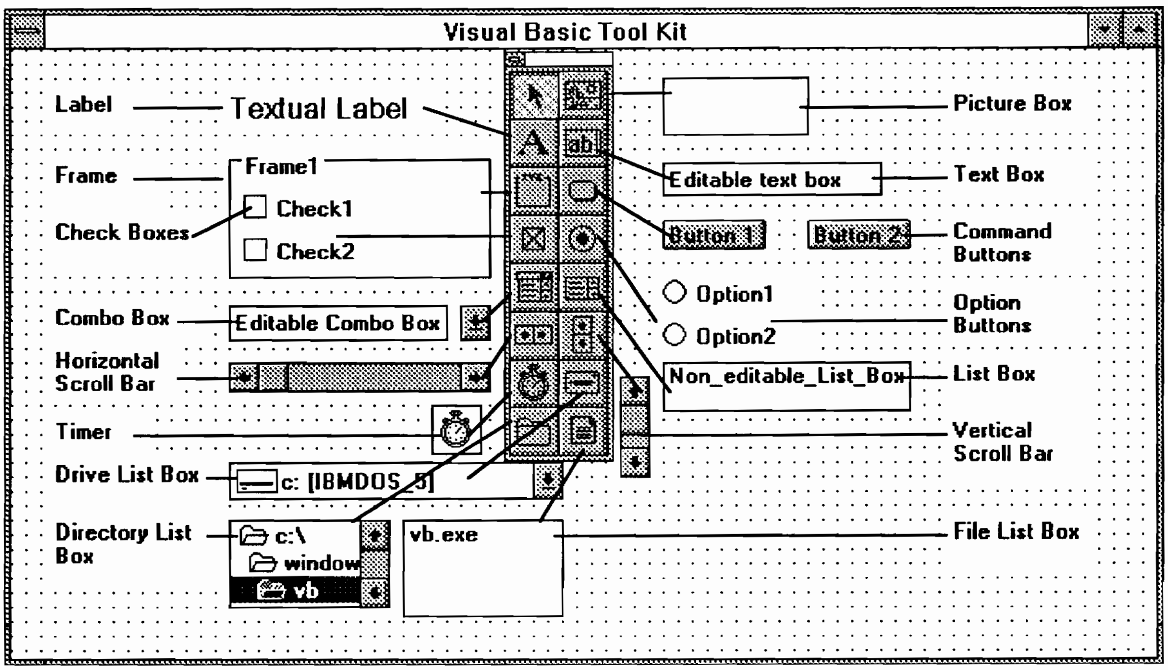


Fig. 2.1 The standard Visual Basic toolbox

The controls in the toolbox window can be simply selected and dragged to the desired position on the form. The controls can also be resized. Each control in a form has its own properties. For example, a property could be a control name, font name or font size, etc.

The toolbox saves considerable programming time and allows the programmer to create professional looking forms.

Visual Basic retains most of the advanced features of QuickBasic and has many enhancements for developing Windows applications. One enhancement over QuickBasic is the manner in which variables are scoped in Visual Basic. A Visual Basic program consists of a project which in turn consists of a global module and at least one form module. The project can also contain general modules containing subroutines that can be called from anywhere in the program.

The global module declares global variables and constants used in the project. The global variables and constants are accessible from any form or module in the project. There is no executable code in the global module. Variables and constants declared in the general declarations section of a form or general module are accessible from any subroutine or function in that form or module. Any variable or subroutine call in a form or module is local to that form or module. Figure 2.2 gives an example of the scope of variables (Craig, 1990).

In the example shown in Figure 2.2, the program starts to execute the For/Next statement when the form is loaded. Then it prints 0, 37.699116 6 when the mouse is clicked on the form. It is obvious that the global declared constant PI and variable Total exist anywhere in form1, the local declared constant TWOPI and local variable SubTotal exist only in the form, and the undeclared I and PiTotal exist only in Form_Load subroutine.

```
The Project

Global module
    Global Const PI = 3.141593
    Global Total As Integer

Form1 - General Declarations
    Const TWOPI = PI + PI
    Dim SubTotal as Double

Form1 - Runs when program starts
    Sub Form_Load ( )
        For I = 1 To TWOPI
            PiTotal = PiTotal + PI
            SubTotal = SubTotal + TWOPI
            Total = Total + 1
        Next I
    End Sub

Form1 - Runs when mouse is clicked
    Sub Form_Click
        Print PiTotal; SubTotal; Total
    End Sub
```

Fig. 2.2 Example of scope of variables in Visual Basic

Another enhancement is that a Visual Basic program is event driven. When viewed from the outside, a form contains a series of controls such as text boxes, labels, radio buttons or command buttons. These controls are designed for entering text, displaying information, making multiple choices or executing certain kinds of functions. When viewed from the inside, each control within a form has a unique control name and several subroutines that are associated with the control. Each subroutine within the form is specified by a control name and an event. For example, a button control called OKbutton would have a subroutine named OKbutton_Click( ) that is associated with the control OKbutton and that responds to the event that takes place when the button is pressed.

As a second example, consider the Open File form shown in Figure 2.3. The Open File form has a file list box which displays the file names to be opened. This control has a number of events associated with it. These include the following events:

- Click
- Double click
- Gotfocus
- Keydown
- Keypress

...

After accessing the File Open form, a file can be opened in two ways. The user can select the file name by clicking on it in the file list box and then clicking on the OK button, or the user can simply double click on the file name in the file list box. For this example, the programmer only needs to write three separate subroutines, called File1_Click( ), Command1_Click( ) and File1_DblClick( ) (File1 is the default control name for the file list box, Command1 is the default control name for the first command button in the form. Click and DblClick are the associated events.) When the event File1_Click occurs, the subroutine File1_Click( ) is called and the name of the file that has been selected is shown in the text box designed to display the file name. The subroutine Command1_Click( ) is called when the OK button is clicked. This subroutine then opens the file. When the user double clicks on the file name in the file list box, the subroutine File1_DblClick( ) is called. This subroutine places the name of the file in the text box and then opens the file.

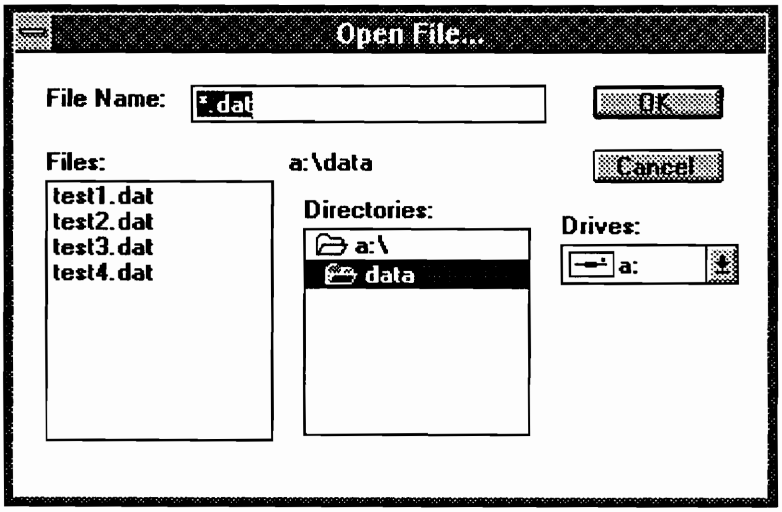


Fig. 2.3 Open File form

In summary, each user interface element has a series of events associated with it. The programmer writes a series of subroutines that respond to these events. Also, all executable code in Visual Basic exists within a subroutine or function block. This is ideal for the development of event-driven programs.

Visual Basic also supports the exchange of data with other applications and the establishment of links with other applications. It also provides support for custom routines written in another programming language.

CHAPTER THREE

PREPROCESSOR

3.1 INTRODUCTION

In this chapter a description of the preprocessor is presented. The important features of the preprocessor are described. A description of the input data required to generate a structural model and the various coordinate systems are also given. This chapter also describes the structure of the preprocessor and the forms (or dialog boxes) that the processor provides for entering structural data.

The main components of the preprocessor are:

a) File processing, b) Data entry forms, c) Data review with graphics and d) On-line help.

The main form of the preprocessor is shown in Figure 3.1

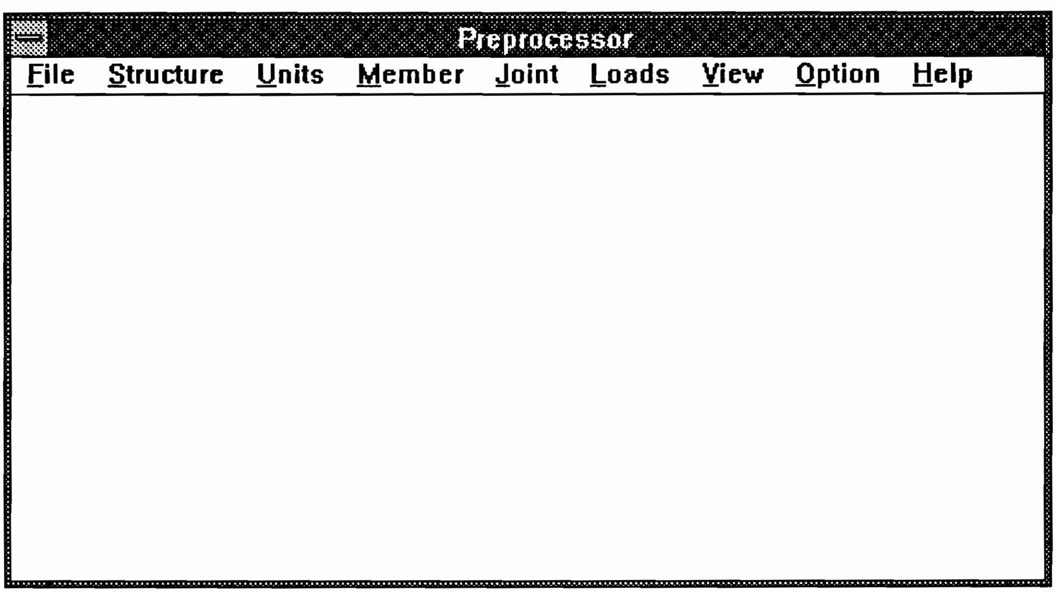


Fig 3.1 Main form of the preprocessor

Preprocessor

The file processing functions handle all options related to reading and writing files. The file menu provides options for creating new files, opening an existing file and saving and renaming files. All of these functions follow the standard Windows format for file operations.

Data entry forms are used to create the structural model. All of the forms related to data entry are provided with standard command buttons such as OK and Cancel buttons.

The data review forms provide an easy way to check the data. There is both graphical output and text output. The graphical output displays the structure geometry, joint number, member number and applied loads. The information displayed in the text output window is controlled by a pull down menu.

Help text gives on-line help for the user. The help text can be searched by index.

3.2 TYPES OF STRUCTURES

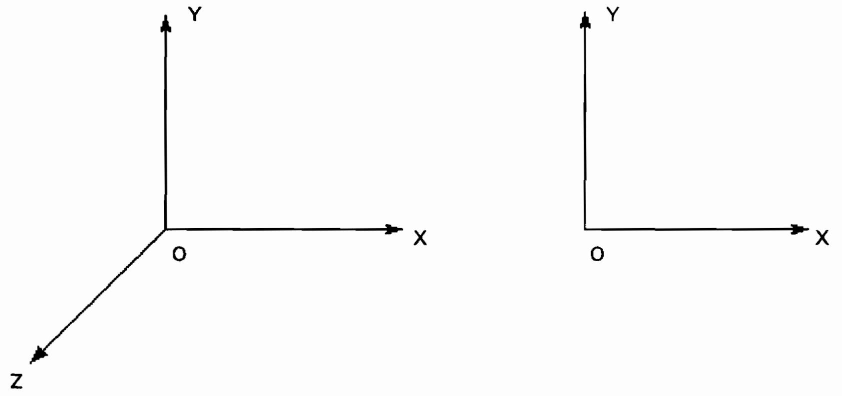
The preprocessor can be used to create four types of structure models. The most general case is the space frame structure, which is a three-dimensional frame with loads applied in any plane. A plane frame structure is bound by the global X-Y coordinate system with loads applied in the same plane. Space truss structure is a three-dimensional structure. It consists of members which are pinned at the ends. The members of the truss are subjected to axial forces only and there are no bending moments in the members. A plane truss is bound by the global X-Y coordinate system.

3.3 COORDINATE SYSTEMS

During the structural analysis, it is necessary to have a reference coordinate system for describing the structure, and the magnitude and direction of joint loads and member loads. Two types of coordinate systems are adopted in the preprocessor. The coordinate system used to reference information related to joints is known as the global coordinate system. The coordinate system used to reference individual members is known as the local coordinate system.

3.3.1 GLOBAL COORDINATE SYSTEM

All joint locations and all joint loads are described in the global coordinate system. The global coordinate system consists of a set of right-hand orthogonal axes, X, Y and Z, which are oriented as show in Figure 3.2.



a) Space frame and space truss b) Plane frame and plane truss

Fig. 3.2 Global coordinate system

The orientation of the global coordinate system is arbitrary as long as the relative orthogonality of the axes, as shown in Figure 3.2, is maintained. A common choice is to orient the X and Z axes in a horizontal plane with the Y axis extending upward.

For a plane structure, the X and Y axes will be in the plane of the structure. The location of the origin of the coordinate system is also arbitrary.

3.3.2 LOCAL COORDINATE SYSTEM

A local coordinate system is associated with each member. The local coordinate system also follows the right-hand rule. Figure 3.3 shows the local coordinate system of a beam member. The local coordinate system is defined as follows:

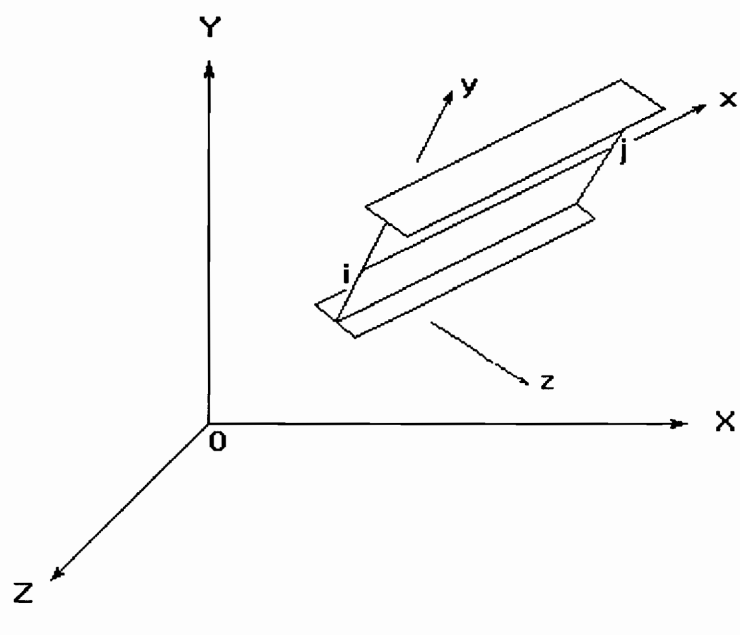


Fig. 3.3 Local coordinate system

The local x axis is always along the longitudinal axis of the member, positive from the smaller joint number to the larger joint number. The local y axis goes in the weak axis direction of the cross section. The local z axis is along the strong axis of the cross section. The positive direction of the local y and z axes is defined by the right-hand rule.

3.4 FILE PROCESSING MENU

The file processing function gives the user options for creating a new model, modifying an existing model, saving the current model on disk, loading a model from a file and exiting the program.

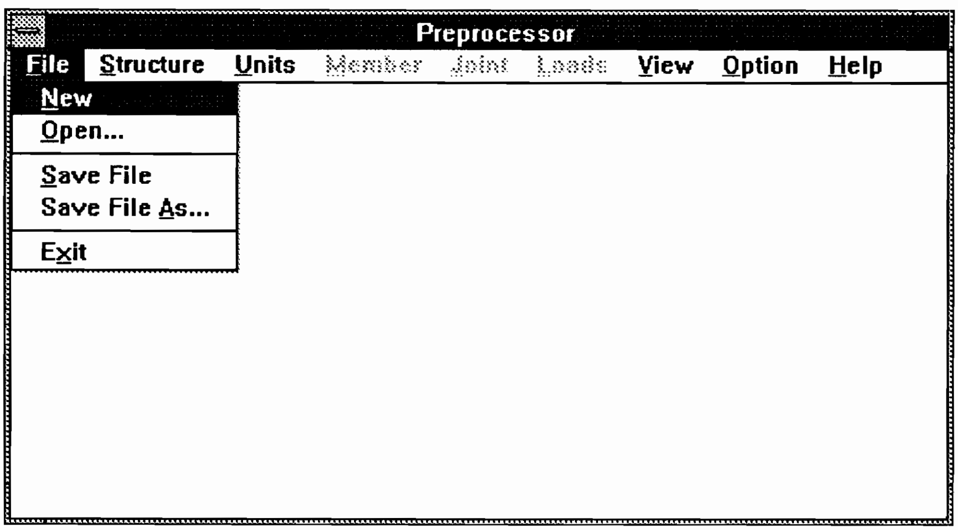


Fig.3.4 File menu
Fig.3.4 File menu

Selecting the New option from the file menu results in the creating of a new model. Also, all the existing data in memory is deleted. However, if an existing file has been changed and has not been saved or a new file has not been saved, a pop up message box will appear to remind the user to save the file. Selecting Yes will save the current data and then create a new model. Selecting No creates a new model without saving the current data. The Cancel option in the pop up message box transfers control to the previous screen.

The Open file option on the file menu allows the user to load an existing structural model saved on disk. The user can type in the drive name, directory name and the file name or simply click the drive list box, directory list box and file list box to open the file. The data will be loaded into the preprocessor after this operation. Again, if a previous model has not been saved, a pop up message box will appear to remind the user to save the current file before opening another file. The Open File form is closed automatically after the file has been loaded. This form can also be closed without loading in the data by pressing the Escape key (Esc) or by clicking the Cancel button. The Open File form is shown in Figure 3.5.

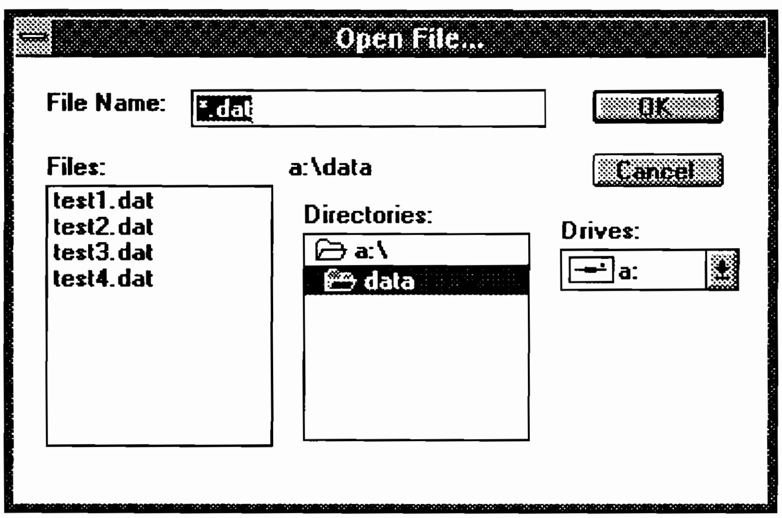


Fig 3.5 Open File form

The Save File option is used for saving the structural model to a disk file. If a name has not been assigned to the file, the Save File As form will appear when the Save File option is selected. The Save File As form allows the user to select the file name, drive name and directory name to which the file is to be saved. The Save File As form is closed automatically after the file has been saved. Pressing the Escape key (Esc) or clicking the Cancel button closes this form without saving the file. The Save File As form is shown in Figure 3.6.

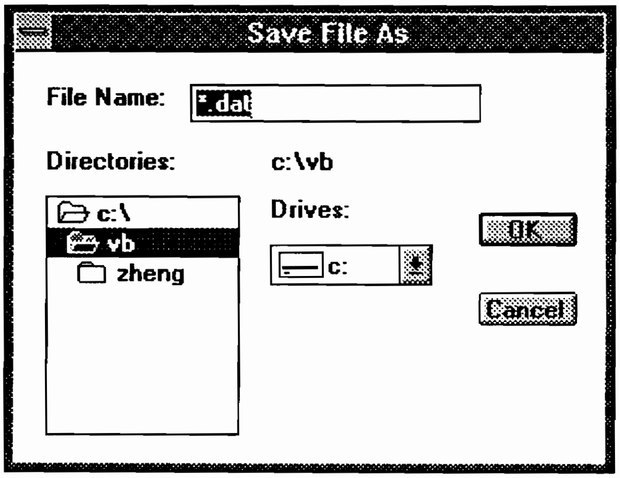


Fig. 3.6 Save File As form

The Save File As menu option is for saving the data file under a different name. Normally, this option is used to create a new file or to make a copy of an existing data file.

The Exit menu option allows the user to exit the application. If an existing file has been changed and has not been saved, or a new file has not been saved, a pop up message box appears to remind the user to save the file before exiting the preprocessor.

3.5 DATA ENTRY FORMS

The preprocessor utilizes a series of dialog boxes (or forms) for entering information regarding the structural model. This section describes various types of forms and the information that has to be entered in these forms.

3.5.1 FILE PATH FOR SUPPORTING FILES

The preprocessor requires the following supporting files:

1) AISC shape data base

2) On-line help text

These files should be installed on the hard drive or floppy diskette. It is necessary to specify the location of these files so that the program can access them. The input form for entering the file path of supporting files is shown in Figure 3.7.

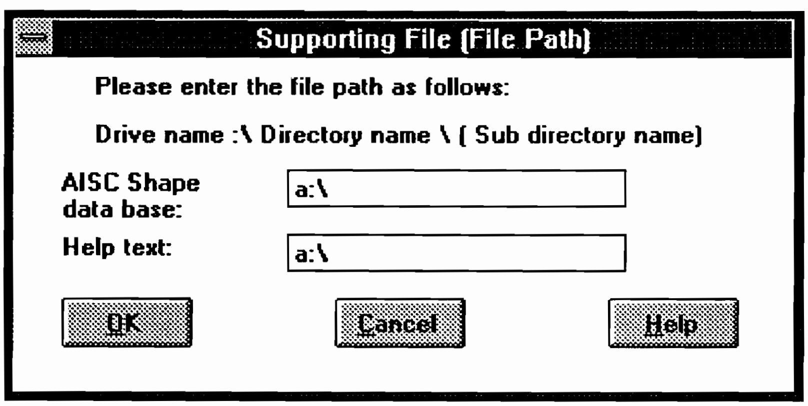


Fig. 3.7 File path for supporting files

The AISC Shape data base contains the section properties of the sections listed in the AISC Manual. (AISC, Load and Resistance Factor Design, First Edition, 1986) The available shapes include C sections (channel), W sections, WT sections (Tee), S sections, L sections (angle), HP sections, M sections and MC sections.

The on-line help file contains the help text. The preprocessor has a context sensitive help system which provides help on using the various input forms. It also provides detailed help on the required input parameters.

3.5.2 STRUCTURAL DATA

The type of structure to be analyzed is also specified in this form. The program can generate input files for four types of structures. The most general case is the space frame structure, which is a three-dimensional frame with loads applied in any plane. A plane frame structure is bound by the global X-Y coordinate system with loads in the same plane. A truss structure consists of members which are pinned at the ends. The members of the truss are subjected to axial forces only and there are no bending moments in the members. A truss can be either a space truss or a plane truss. The default structure is the space frame structure.

The structural data input form is shown in Figure 3.8. The information to be entered in this form consists of the project description, the type of structure, number of joints, number of members, number of load cases and number of load combinations.

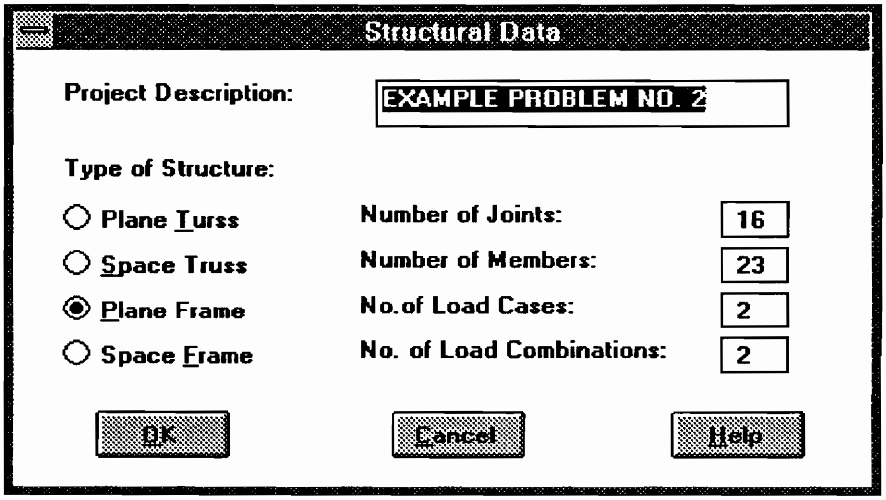


Fig. 3.8 Structural data input form

The automatic data checking mechanism for this form will perform the following checks:

1) If this form is opened a second time and the total number of joints or the total number of members has been modified, the preprocessor will provide the user with the option to save the current data. If the user does not select this option, then all the data in memory will be erased except for the data in the structural data input form. This feature is useful when changing the geometry of structure such as adding or deleting a story of a building since it allows the user to save existing information regarding the structure.

2) The maximum number of joints is limited to 100. The maximum number of members is also limited to 100. The maximum number of load cases and load combinations is 10. The preprocessor checks to make sure that these limits are not exceeded. The limits on the maximum number of members and maximum number of joints are based upon available computer memory, and the fact that the preprocessor is designed for the analysis small and medium size structures. However, it is possible to increase these limits by modifying the dimension statement in the program and recompiling the program.

3.5.3 SELECTION OF UNITS

The preprocessor provides a choice of two types of units, the US system and the SI system. The default system of units used by the preprocessor is the US system.

The units for the various quantities used in the US and SI systems are shown in Table 3.1.

Table 3.1 Units for various quantities

Quantity	US	SI
Length	ft	m
Force	kip	KN
Moment	kip-ft	KN-M
Temperature	°F	°C
Stress	ksi	GPa
Angle	degree	degree

3.5.4 MEMBER INCIDENCES

Member incidences are used to define how members are connected to the joints. Each member has a starting joint and an ending joint. Thus, to define a member, it is necessary to specify the starting joint number and the ending joint number. Figure 3.9 shows the member incidences input form.

The required input parameters for this form include the member number, starting joint number and ending joint number. The member number is generated automatically after the Update button is clicked. However, the user can also change the member number when necessary.

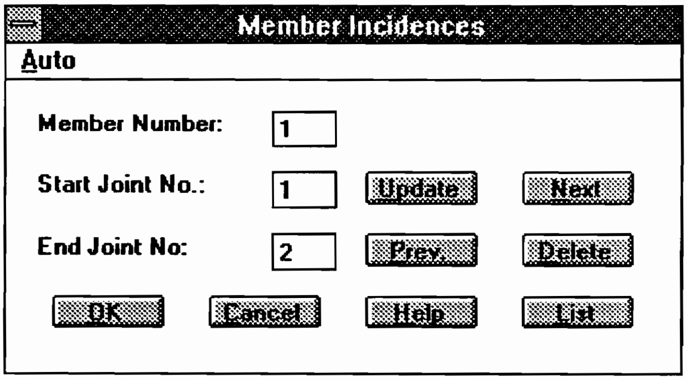


Fig. 3.9 Member incidences input form

This input form features an automatic member incidences generation function. The member incidences auto-generation form is activated by selecting the Auto menu option. The auto-generation form is shown in Figure 3.10. The user needs to input the member incidence of the first member of a set of members, the member number increment, the member number of the first and last member in the group and the joint number increment. The preprocessor generates the member incidences for the remaining members in this group.

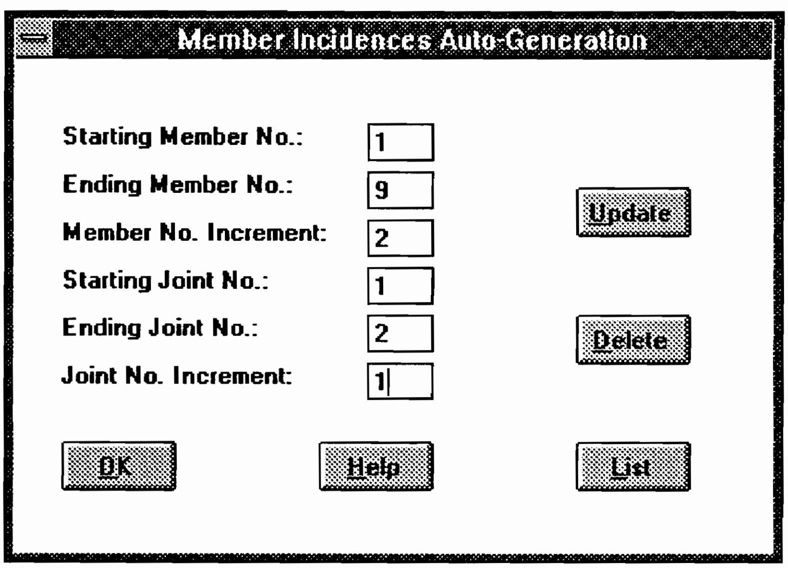


Fig. 3.10 Member incidences auto-generation form

The data checking mechanism performs the following checks:

1) The member number cannot be greater than the total number of members.

2) The member number cannot be less than or equal to zero.

3) Two or more members cannot have the same member incidences.

4) The joint number cannot be greater than the total number of joints.

5) The joint number cannot be less than or equal to zero.

6) The start joint number cannot be equal to the end joint number.

7) Any duplicate input is detected and the user is given the option to keep previous data or to overwrite it.

8) In the auto-generation mode, the preprocessor will also check to see whether the increment in joint number will result in a joint number that is larger than the total number of joints or the increment in the member number will result in a member number that is not included in the group to be generated.

To ensure the accuracy of the structural model, the preprocessor requires that member incidences be entered for all members before closing this form. However, the user can close this form without entering member incidences for all members by clicking the Cancel button if necessary.

3.5.5 MEMBER PROPERTIES

Member properties include material properties and geometric properties. Separate input forms are provided for entering material properties and geometric properties. These forms are described below.

3.5.5.1 MATERIAL PROPERTIES

This input form allows the user to enter material properties for each member. The user can select from several predefined materials such as steel, concrete or can define others as desired. Input parameters for material properties include the member number, the modulus of elasticity, the shear modulus (for space frame structures only), the coefficient of thermal expansion and concrete compressive strength. In US units, the preprocessor uses the following default values: 29,000 ksi for the modulus of elasticity and 11,200 ksi for the shear modulus for steel, $6.5 \times 10^{-6}/°F$ and $6.0 \times 10^{-6}/°F$ for the coefficient of thermal expansion for steel and concrete respectively. In SI units, the default values are 200,000 Mpa for the modulus of elasticity and 77,000 Mpa for the shear modulus for steel, $1.17 \times 10^{-5}/°C$ and $1.1 \times 10^{-5}/°C$ for the coefficient of thermal expansion for steel and concrete respectively. The modulus of elasticity and the shear modulus for concrete is computed from the concrete compressive strength. The type of material is not required by the output data file. The radio button for assigning the material type is for the purpose of obtaining default material properties only.

There are two options for the assignment of material properties to members. The series input mode assigns material properties to a series of members. The single input mode assigns material properties to individual members. The default input mode is the series input mode. Figure 3.11 shows the member material properties input form.

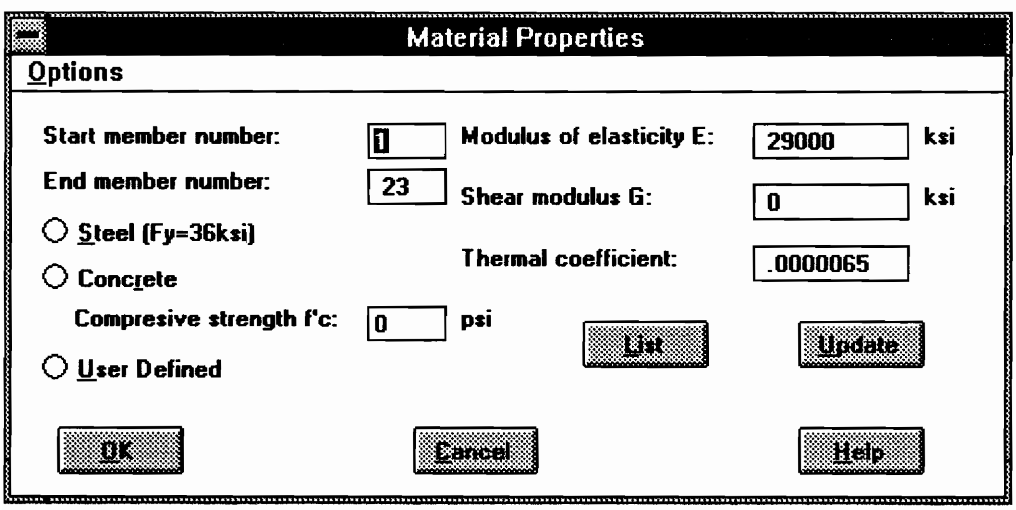


Fig. 3.11 Material properties input form

The data checking mechanism in this form checks for the following errors:

1) Member number out of range

2) Duplicate input

3) Modulus of elasticity less than or equal to zero

4) Shear modulus or coefficient of thermal expansion less than zero.

The shear modulus is required only for space frame structures. The coefficient of thermal expansion is not required if there are no temperature loads.

In the case of duplicate input, a message box appears on the screen. Selecting OK will overwrite existing data. Selecting Cancel will close the message box window and preserve the existing data.

3.5.5.2 GEOMETRIC PROPERTIES

The geometric properties input form allows the user to enter the geometric parameters for individual members or for a group of members. The input parameters for this form include the member number, cross sectional area and moment of inertia. The moment of inertia is not required for trusses since there is no bending in the truss model. For plane frame structures, the moment of inertia about the local z axis is required. It is necessary to input the moment of inertia about all three local axes for the members of a space frame structure since there is both bending and torsion in the members.

If the US system of units is selected then geometric properties can be assigned from a data base containing AISC shapes. The AISC data base includes all of the shapes listed in the AISC manual except for double angles. It contains the type of section, weight per foot, cross sectional area and moment of inertia about the local x, y and z axes. The following shapes are contained in the data base.

- C-Shape - Channels, 29 sections
- L-Shape - Angles, 131 sections
- HP-Shape - 15 sections
- M-Shape - 8 sections
- MC-Shape - 40 sections
- S-Shape - 31 sections
- W-Shape - 187 sections
- WT-Shape - Tees, 187 sections

The AISC shape data base is not available if the SI system of units is selected. In this case, the geometric properties have to be entered manually. Figure 3.12 shows the geometric properties input form.

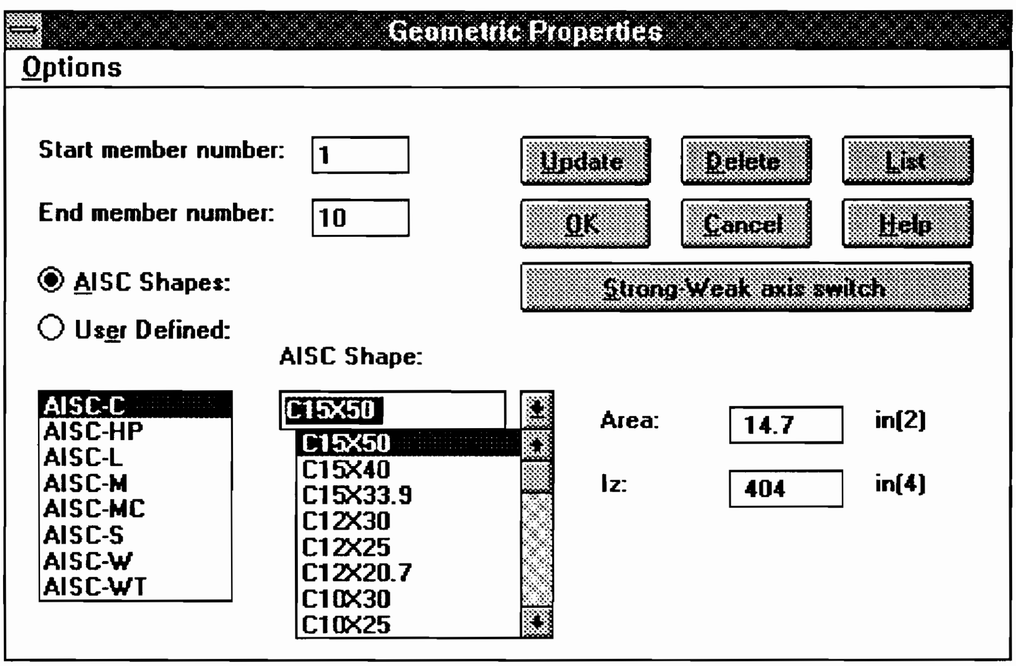


Fig. 3.12 Geometric properties input form

The data checking mechanism for this form checks for the following errors:

1) Member number out of range

2) Duplicate input

3) AISC shape not available in the data base

4) Compressive strength of concrete less than or equal to zero

5) Moment of inertia about the local z axis less than or equal to zero

6) Moment of inertia about the local y axis less than zero

7) Moment of inertia about the local x axis less than zero

In the case of duplicate input, a message box appears on the screen. Selecting OK will overwrite existing data. Selecting Cancel will close the message box window and preserve the existing data.

As was the case with the material properties input form, there are two options for the assignment of geometric properties. The geometric properties can be assigned to a series of members or to individual members. The default mode is to assign geometric properties to a series of members.

The switch option rotates the axis of the member by 90 degrees. This option is provided for plane structures only.

3.5.6 BETA ANGLE

In the space frame model, it is important to know the relationship between the member local axis and the global axis for the structure. This relationship is defined by an angle called the beta (ß) angle. The beta angle is defined as follows:

When the local x-axis is parallel to the global Y-axis, as in the case of a column, the beta angle is the angle through which the local z-axis has been rotated about the local x-axis from a position of being parallel and in the same positive direction of the global Z-axis.

When the local x-axis is not parallel to the global Y-axis, the beta angle is the angle through which the local coordinate system has been rotated about the local x-axis from a position in which the local z-axis was parallel to the global X-Z plane and the local y-axis was in the same positive direction as the global Y-axis. Figure 3.13 shows the orientation of the local and global axes for beta angles of 0 and 90 degrees. The input form for the beta angle is shown in Figure 3.14.

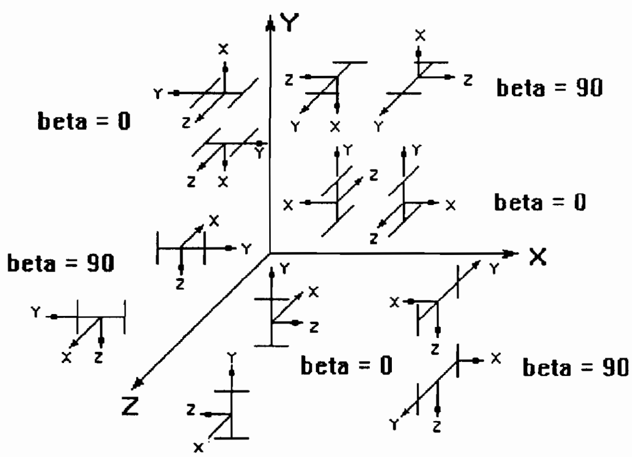


Fig. 3.13 Orientation of the local and global axes for beta angles equal to 0 or 90 degrees

The data checking mechanism for this form performs the following checks:

1) Member number out of range

2) Duplicate input

3) Beta angle less than zero or greater than 360 degrees

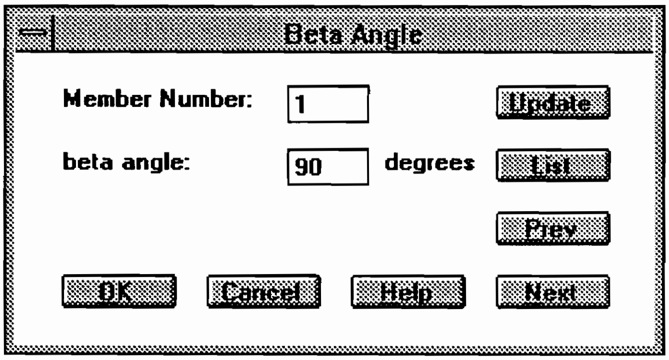


Fig. 3.14 Beta angle input form

In most cases, the beta angle will be 0 degrees. The default beta angle is 0 degrees so it is not necessary to input a zero beta angle.

3.5.7 MEMBER RELEASES

The preprocessor allows for the specification of internal releases in members at one or both ends of a member. One example is that of an internal hinge. An internal hinge can transfer shear force only. In this case, the moment at the end of the member must be released.

In space frame structures, member releases can be the force in local x, y and z directions and the moment about the local x, y and z directions at each end of the member. In plane frame structures, member releases are limited to the force in local x and y directions and the moment about the local z direction. There are no moment releases for members of a truss.

The member releases input form is shown in Figure 3.15. The necessary input data consists of the member number and the location and direction of the release.

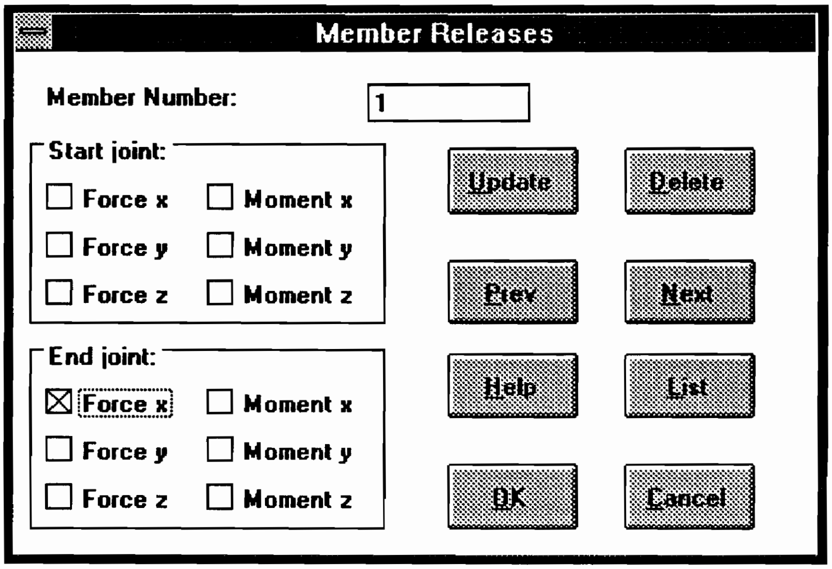


Fig. 3.15 Member releases input form

The following data checking is performed:

1) Member number out of range

2) Duplicate input

3.5.8 JOINT CONSTRAINTS

The joint constraints input form allows the user to specify constraints for given joints. A joint can be constrained against displacement in the global X, Y and Z directions or against rotation about the global X, Y and Z directions. The preprocessor provides several predefined joint constraints in order to simplify the amount of effort required in specifying joint constraints.

The predefined joint types for a space truss model are as follows:

- Pinned — Constrained against displacement in the global X, Y and Z directions.

- Roller (Horizontal) — Constrained against displacement in the global Y and Z directions.
- Roller (Vertical) — Constrained against the displacement in the global X and Z directions.
- Roller (In the plane) — Constrained against the displacement in the global X and Y directions.

The joint constraints input form for a space truss is shown in Figure 3.16.

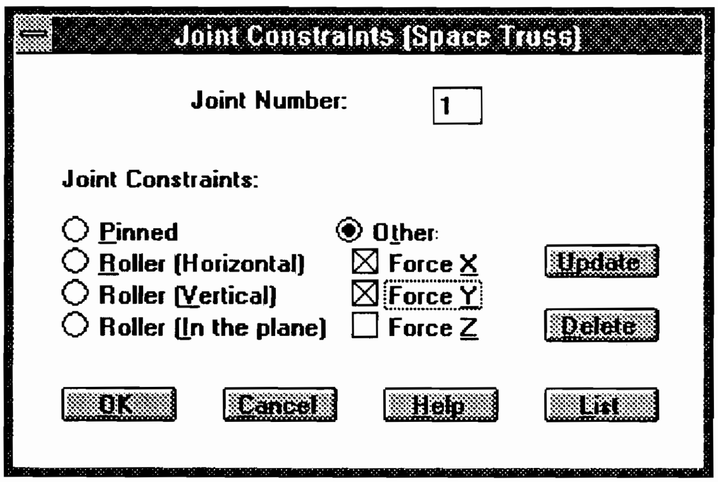


Fig. 3.16 Joint constraints input form for a space truss

For a plane frame model, the predefined joint types are as follows:

- Fixed — Constrained against displacement in the global X and Y directions and rotation about the global Z direction
- Pinned — Constrained against displacement in the global X and Y directions.
- Roller (Horizontal) — Constrained against displacement in the global Y direction.

- Roller (Vertical) — Constrained against displacement in the global X direction.
- Shear release (Horizontal) — Constrained against displacement in the global Y direction and rotation about the global Z direction.
- Shear release (Vertical) — Constrained against displacement in the global X direction and rotation about the global Z direction.

The joint constraints input form for a plane frame is shown in Figure 3.17

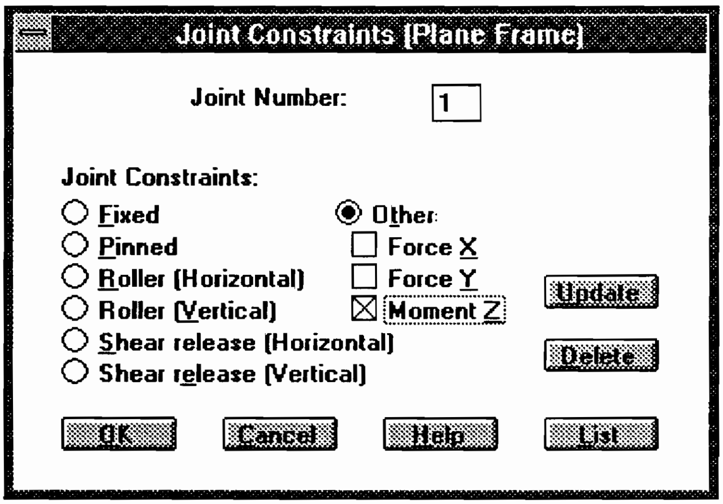


Fig. 3.17 Joint constraints input form for a plane frame

For a space frame model, the predefined joint types are as follows:

- Fixed — Constrained against displacement in the global X, Y and Z directions and rotations about the global X, Y and Z directions.
- Pinned — Constrained against displacement in the global X, Y and Z directions.

The joint constraints input form for a space frame is shown in Figure 3.18.

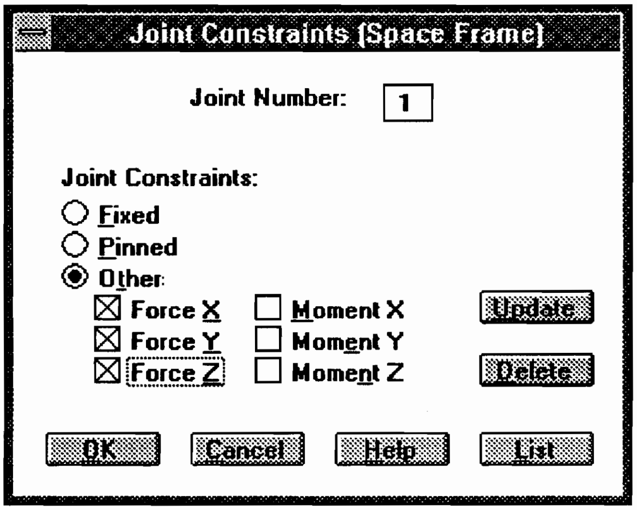


Fig. 3.18 Joint constraints input form for a space frame

For a plane truss model, the predefined joint types are as follows:

- Pinned — Constrained against displacement in the global X and Y directions.
- Roller (Horizontal) — Constrained against displacement in the global Y direction.
- Roller (Vertical) — Constrained against displacement in the global X direction.

The joint constraints input form for a plane truss is shown in Figure 3.19.

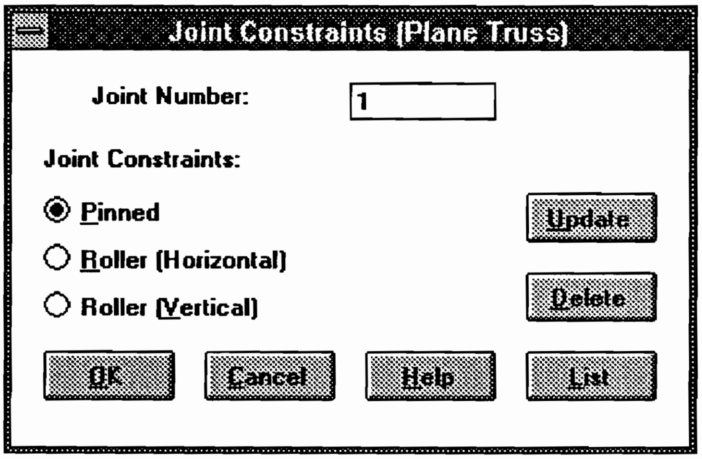


Fig. 3.19 Joint constraints input form for a plane truss

In addition to the predefined joint types, there is an option for entering a user-defined joint constraint.

The data checking mechanism checks for the following types of errors:

1) Joint number out of range

2) Duplicate input

In the case of duplicate input, a message box appears on the screen. Selecting OK will overwrite existing data. Selecting Cancel will close the message box window and preserve the existing data.

3.5.9 JOINT COORDINATES

The geometry of structural model is specified by joint coordinates. The joint coordinates input form allows the user to enter the coordinates of each joint in the structure. The input parameters for this form are the joint number and the coordinates of the joint in the global coordinate system. For space frames or space trusses, the global X, Y and Z coordinates are required for each joint. In the case of a plane truss or a plane frame, the global Z coordinate is not required. Figure 3.20 shows the joint coordinates input form.

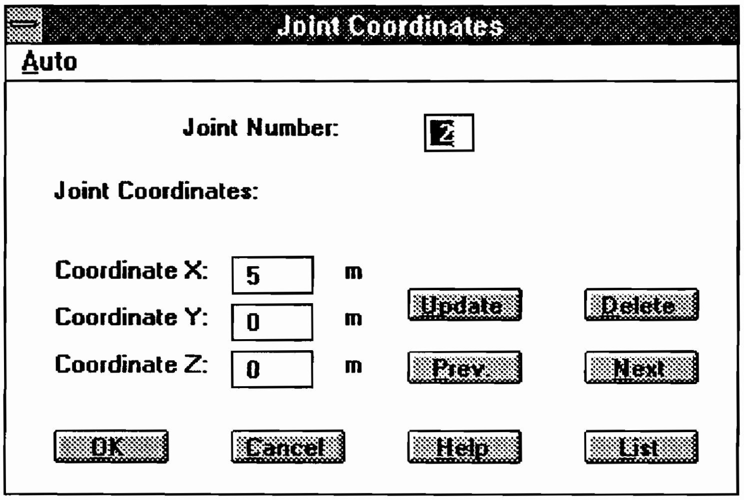


Fig. 3.20 Joint coordinates input form

The data checking mechanism checks for the following errors:

1) Joint number out of range

2) Duplicate input

3) Two or more joints have the same joint coordinates

In the case of duplicate input, a message box appears on the screen. Selecting OK will overwrite existing data. Selecting Cancel will close the message box window and preserve the existing data.

The joint coordinates input form has an automatic joint coordinates generation feature. In the auto-generation mode, joint coordinates can be generated by giving the joint number of the first and last joint in the series, the joint number increment, the joint coordinates of the first joint and the increment in these coordinates. The joint coordinates auto-generation input form is shown in Figure 3.21.

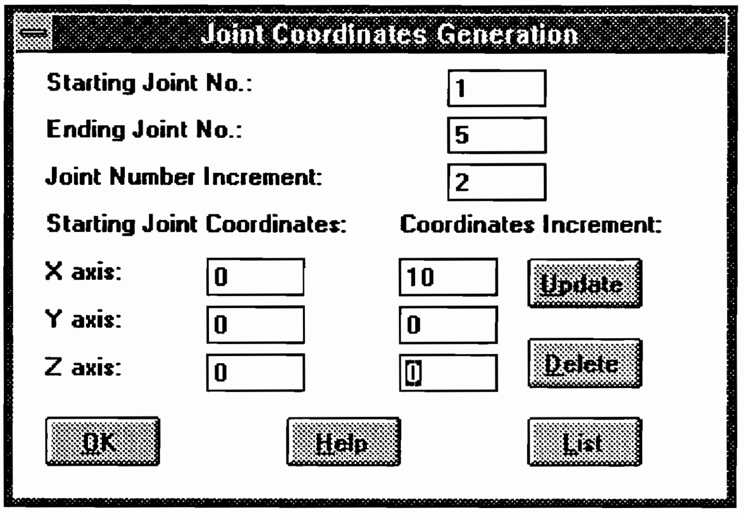


Fig. 3.21 Joint coordinates auto-generation form

The data checking mechanism in this form also checks whether the joint number increment results in a joint number that is not included in the group to be generated.

To ensure the accuracy of the structural model, the preprocessor requires the user to enter joint coordinates for all joints before closing this form. It is however, possible to exit without entering the data for all joints by simply clicking the Cancel button.

The preprocessor automatically computes the length for all members after the joint coordinates of all joints and member incidences of all members have been entered. Member lengths are not calculated if the joint coordinates input form is dismissed prior to entering all joint coordinates.

3.5.10 LOADS

The loads applied on a structure include joint loads, member loads and prescribed displacements. The preprocessor has separate input forms for entering the different types of loads. In addition, there are optional input forms that allow the user to enter load case descriptions and load combinations. These input forms are described below.

3.5.11 LOAD CASE DESCRIPTIONS

All loads are associated with a load case. The load case description input form allows the user to enter a brief description for each load case, such as Dead Load, Live Load, etc.

The required input parameters for the load case descriptions input form are the load case number and its description. The load case description can be entered directly by selecting from several predefined load case descriptions in the list box or by typing in the text for the load case description in the text box provided for this purpose. The load case descriptions input form is shown in Figure 3.22.

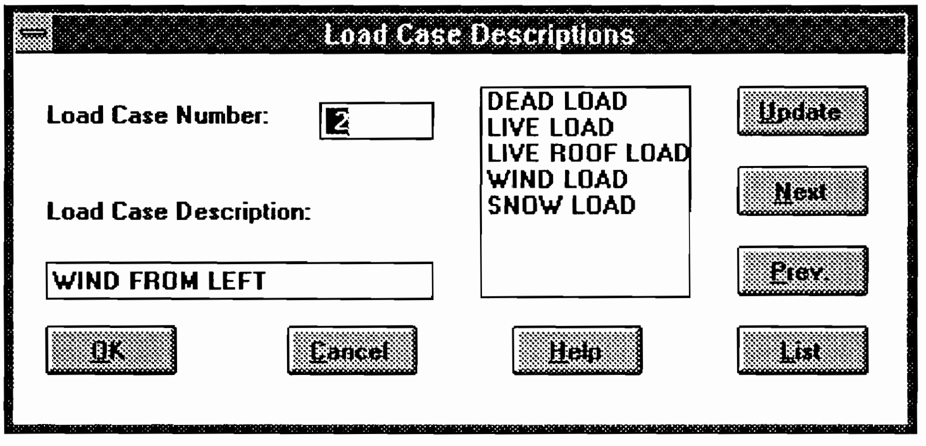


Fig. 3.22 Load case descriptions input form

The data checking mechanism for this form performs the following checks:

1) Load case number cannot be greater than the total number of load cases or less than 0.

2) Duplicate input

3) Same load case description cannot be entered a second time.

3.5.12 LOAD COMBINATIONS

With the adoption of Load and Resistance Factor Design (LRFD), the response of the structure under different load combinations has to be considered in the design process. The preprocessor allows the user to specify up to ten load combinations with each combination containing up to five load cases.

The required input parameters for the load combinations input form are the load case descriptions and the corresponding load factors. This form shall not be opened unless all load case descriptions have been entered. It is not necessary to type in the load case descriptions in this form since all load case descriptions are listed in a list box allowing for easy selection of the load case. The load combinations input form is shown in Figure 3.23.

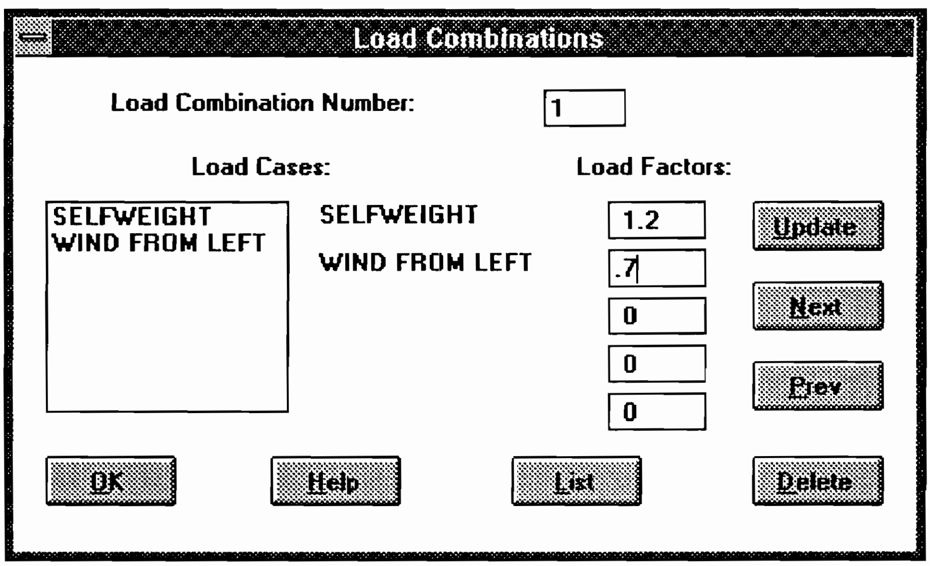


Fig. 3.23 Load combinations input form

The data checking mechanism performs the following checks:

1) Load combination number cannot be greater than the total number of load combinations or less than zero.

2) The same load case cannot be used more than once in a load combination.

3) Duplicate input

3.5.13 JOINT LOADS

Joint loads can be specified either as forces acting at the joint or as moments. In a space truss structure, a joint load can be a force acting in the global X, Y or Z directions. In a plane truss, joint loads have to be in the global X-Y plane. For a plane frame structure, a joint load can be a force acting in the global X and Y directions or a moment about the global Z direction. For a space frame, joint loads can be forces or moments acting in the global X, Y and Z directions. The input form for joint loads is shown in Figure 3.24.

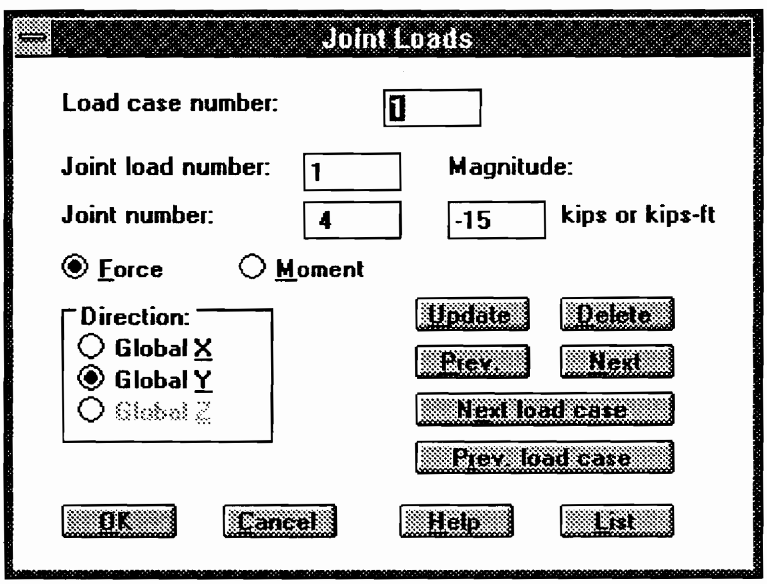


Fig. 3.24 Joint loads input form

Preprocessor

The input parameters for this form include the load case number, joint load number, joint number, and direction and magnitude of the load. Depending on the type of structural model, the preprocessor disables the input fields that are not applicable to that model.

The Next/Prev. Load Case buttons are provided for the selection of the load cases. The joint load number is the number of joint loads for a given load case. The joint load number is generated automatically. The joint number specifies the joint at which the load is applied. The direction of the joint load is defined by the global coordinate system.

The data checking mechanism for this form performs the following checks:

1) Load case number out of range
2) Joint load number out of range
3) Joint number out of range
4) Joint load type (force or moment) not selected
5) Joint load direction not selected
6) Joint load magnitude equals zero
7) Duplicate input

3.5.14 MEMBER LOADS

Member loads are specified by the type of member load, the plane in which it acts, the distance from the start of the member to the load, the magnitude of the load and the member number of the member on which the load acts.

The direction of the member load is defined in the local coordinate system. For a truss, the direction of member load is always along the length of the member since the load is axial. In a plane frame structure, transverse member loads act in the local x-y plane. If the structure is a space frame, a transverse member load is decomposed into forces acting in the local x-y plane and local x-z plane.

Member loads are classified as:

1) Concentrated load
2) Uniform load
3) Linear load
4) Uniform temperature change.

3.5.14.1 CONCENTRATED LOADS

A concentrated load is also called a point load. It is a force acting on the member at a point. Concentrated loads can be either axial or transverse. The input form for concentrated loads is shown in Figure 3.25. The required input parameters include the load case number, member number of the member subjected to the concentrated load, the magnitude of the load, distance from the starting joint to the point at which the load acts and the direction of the load.

3.5.14.2 UNIFORM LOADS

A uniform load acts along the member. It can be axial or transverse. The input form for uniform loads is shown in Figure 3.26. The required input parameters for the uniform load include the load case number, the member number of the member on which the load is acting, and the magnitude and the direction of the load.

3.5.14.3 LINEAR LOADS

A linear load acts on a portion of member and increases or decreases linearly in magnitude from the left to the right side. A linear load is the general case for the uniform load.

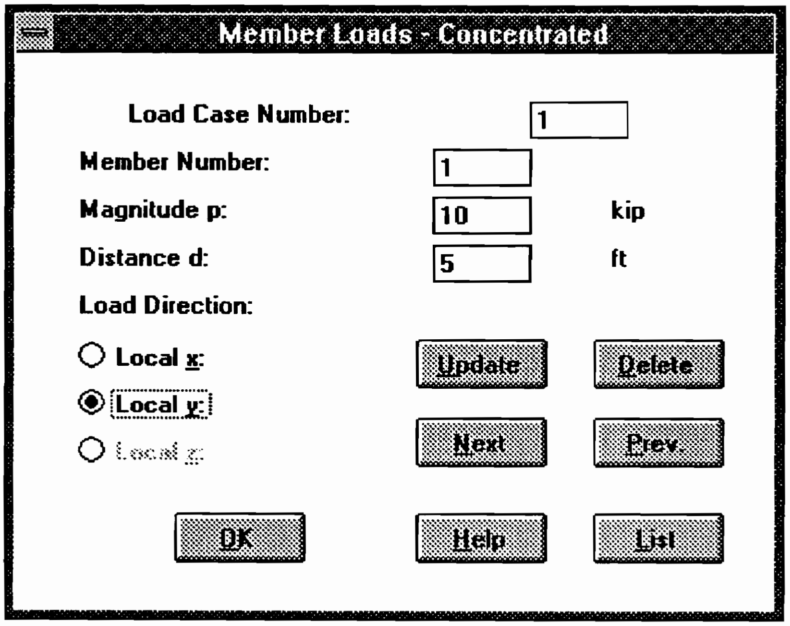


Fig. 3.25 Concentrated loads input form

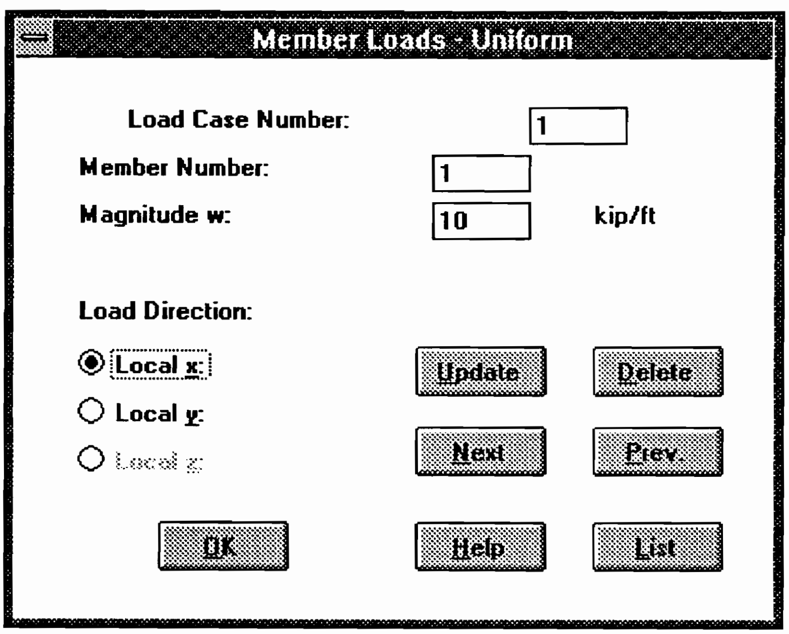


Fig. 3.26 Uniform loads input form

A uniform load can be expressed in the form of a linear load which has the same magnitude along the length of the member. The input form for linear loads is shown in Figure 3.27.

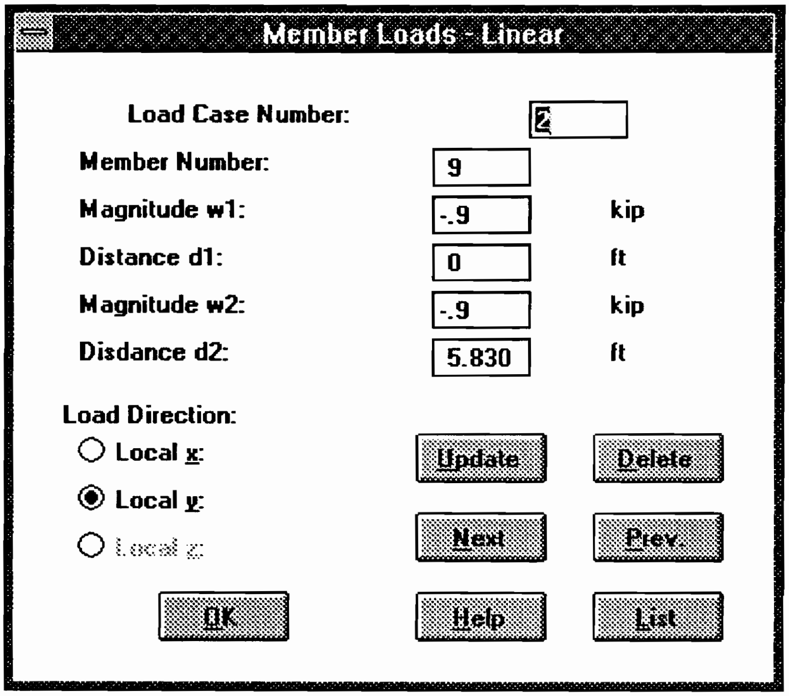


Fig. 3.27 Linear loads input form

The required input parameters for a linear load include the load case number, the member number of the member on which the load is acting, the magnitude of the load at both sides, the distances from the start joint to the start and end of the load, and the direction of the load.

3.5.14.4 UNIFORM TEMPERATURE CHANGE

A temperature load is different from the previously described member loads. To simplify the analysis, it is assumed that the temperature change is uniform. Therefore the required input parameters for temperature loads input form are the load case number, the member number of the member subjected to the temperature change and the amount of temperature change. The input form for a uniform temperature change is shown in Figure 3.28.

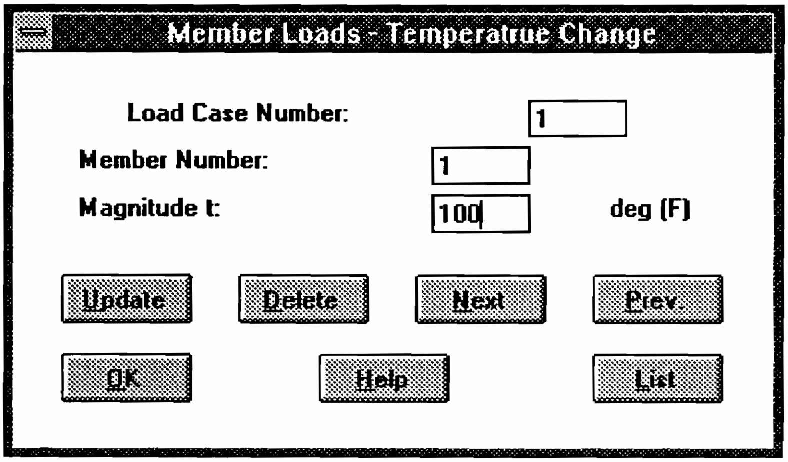


Fig. 3.28 Uniform temperature change input form

3.5.14.5 DATA CHECKING

All of the member load input forms have features for checking input errors. The type of checks made include:

1) Load case number out of range

2) Member number out of range

3) Member load direction not selected

4) Member load magnitude equals to zero

Note that in the linear loads input form, the load at one side can be zero.

5) Member load location out of range. Note that for the concentrated load, $0 < d < L$ where L is the length of the member, d is the distance from the start joint to the point at which the load acts. For the linear load, $0 \leq d1 < d2 \leq L$ where d1 and d2 are distances from the start joint to the start and end of the load.

6) Duplicate input.

3.5.15 PRESCRIBED DISPLACEMENTS

A prescribed displacement is specified by the direction of the displacement, the type of displacement, the magnitude of the displacement and the joint number of the joint subjected to the prescribed displacement. In a truss, the prescribed displacement can only be a translation. In a frame structure, the prescribed displacement can be either a translation or a rotation. The input form for the prescribed displacement is shown in Figure 3.29.

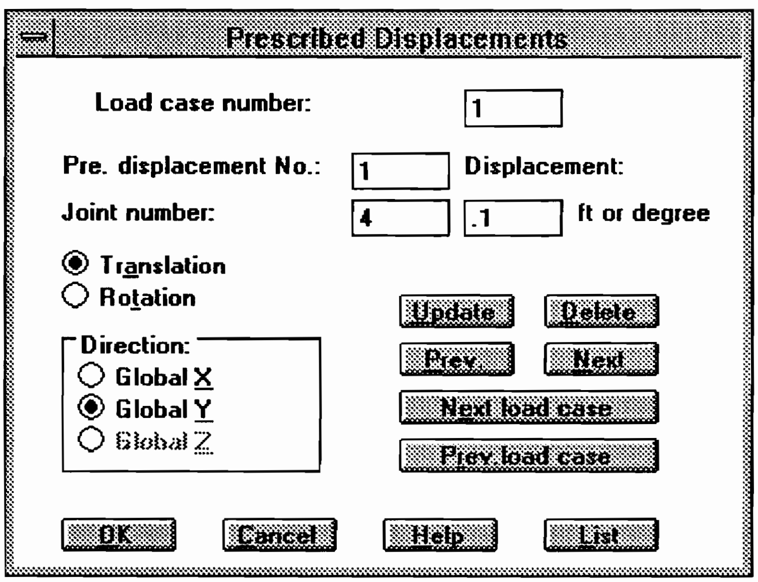


Fig. 3.29 Prescribed displacements input form

The following checks are performed:

1) Load case number out of range

2) Prescribed displacement number out of range

3) Joint number out of range

4) Prescribed displacement type (translation or rotation) not selected

5) Prescribed displacement direction not selected

6) The displacement equals to zero

7) Duplicate input

3.6 DATA REVIEW

The preprocessor has two facilities for reviewing the input data. The data review window displays the data entered in the form of tables. The graphical output function displays the data in graphical form. These are described below.

3.6.1 NUMERICAL DATA REVIEW

This form offers a very convenient way of checking the input data. Unlike the individual input forms which display only one set of data, the data review form can display up to 15 lines of information at a time. The data review window has several menus and sub menus similar to the menus in the main form. All of the input data can be reviewed by making a selection from this menu. The numerical data review form is shown in Figure 3.30.

The following input data can be reviewed through the data review window:

1) Structural data

2) Member incidences and member lengths

3) Geometric and material properties

4) Beta angles

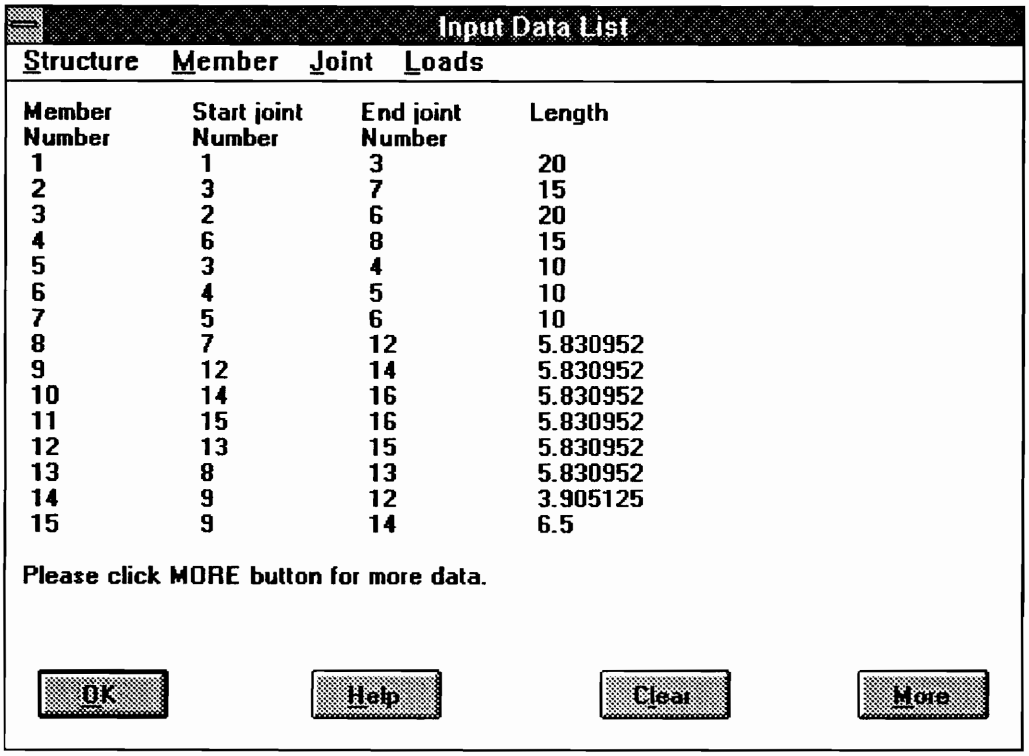
5) Joint releases and member releases

6) Joint coordinates

7) Load case descriptions and load combinations

8) Joint loads, member loads and prescribed displacements

The data review form can also be activated by clicking the List button in the other input forms. The current input form will be resumed automatically after the data review form is closed.



Input Data List

Structure Member Joint Loads

Member Number	Start joint Number	End joint Number	Length
1	1	3	20
2	3	7	15
3	2	6	20
4	6	8	15
5	3	4	10
6	4	5	10
7	5	6	10
8	7	12	5.830952
9	12	14	5.830952
10	14	16	5.830952
11	15	16	5.830952
12	13	15	5.830952
13	8	13	5.830952
14	9	12	3.905125
15	9	14	6.5

Please click MORE button for more data.

OK Help Clear More

Fig. 3.30 Numerical data review form

3.6.2 THREE-DIMENSIONAL GRAPHICS

The preprocessor takes advantage of the Visual Basic graphics functions to display the geometry of the structure. Figure 3.31 shows the graphics output form. The three-dimensional graphics display function has the following features:

3.6.2.1 STRUCTURE

The structure is scaled automatically according to the joint coordinates. This function makes is possible to display the structure so that it fits in the display window. However, some distortion of the displayed image may occur due to scaling.

Since it is necessary to display the structure prior to display other information such as joint numbers, member numbers and loads, all other display functions are disabled until the structure is displayed.

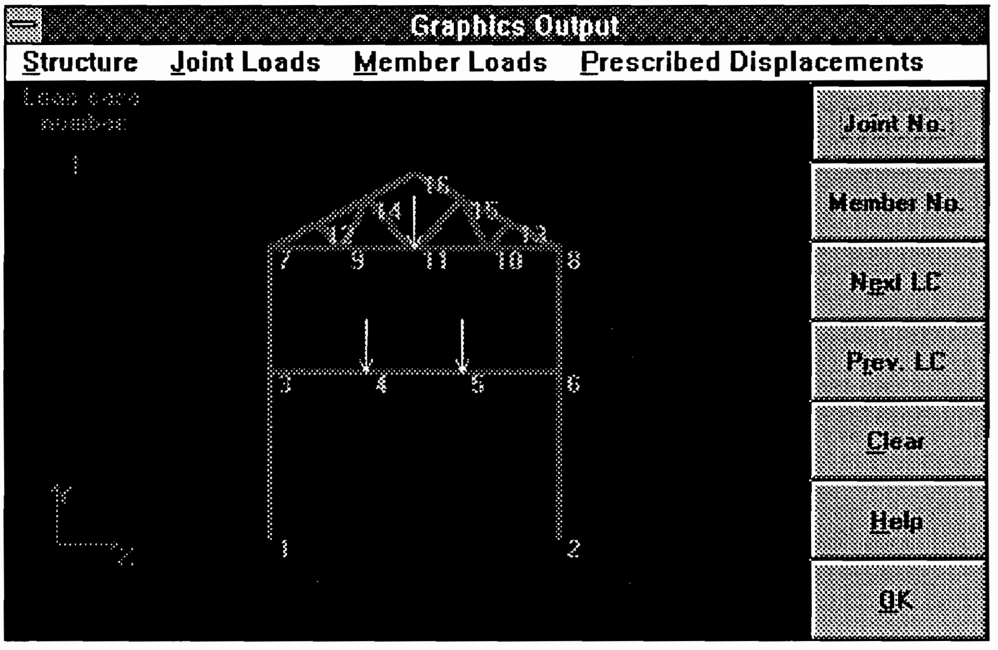


Fig. 3.31 Graphics output form

3.6.2.2 JOINT LOADS

Joint loads include forces and moments. A single arrow is used to display a force acting at a joint and a double arrow is used to display a joint moment. The direction of the joint moment is based on the right-hand rule. The menu items Forces and Moments serve as toggle switches for turning on/off the display of the joint forces and joint moments. These menu items are grayed if there are no joint loads for the given load case.

3.6.2.3 MEMBER LOADS

The different types of member loads are shown separately. The menu item corresponding to a particular type of member load is grayed if there is no member load of that type in the given load case. The display of loads can be erased from the screen by clicking the corresponding menu item a second time.

3.6.2.4 PRESCRIBED DISPLACEMENTS

Prescribed displacements include translations and rotations. A single arrow is used to show a translation and a double arrow is used to show a rotation. Again, the direction of the rotation is based on the right-hand rule. The menu items Translation and Rotations serve as toggle switches for turning on/off the display of the translation and rotations. These menu items are grayed if there are no prescribed displacements for the given load case.

3.6.2.5 JOINT NUMBERS AND MEMBER NUMBERS

Joint numbers and member numbers can be displayed on the structure by clicking the corresponding command buttons and can be erased by clicking the same button a second time.

3.6.2.6 NEXT/PREV. LOAD CASE BUTTONS

The Next/Prev. Load Case buttons allow for the selection of the next or previous load case. The load case description is also displayed on the screen.

3.7 THE HELP SYSTEM

The preprocessor has an on-line help system that is context sensitive. Each form has a help button which activates a help form that provides an explanation of the steps required to enter the data in the form. There is also a help index which provides access to help on a specific topic. The index help form is shown in Figure 3.32.

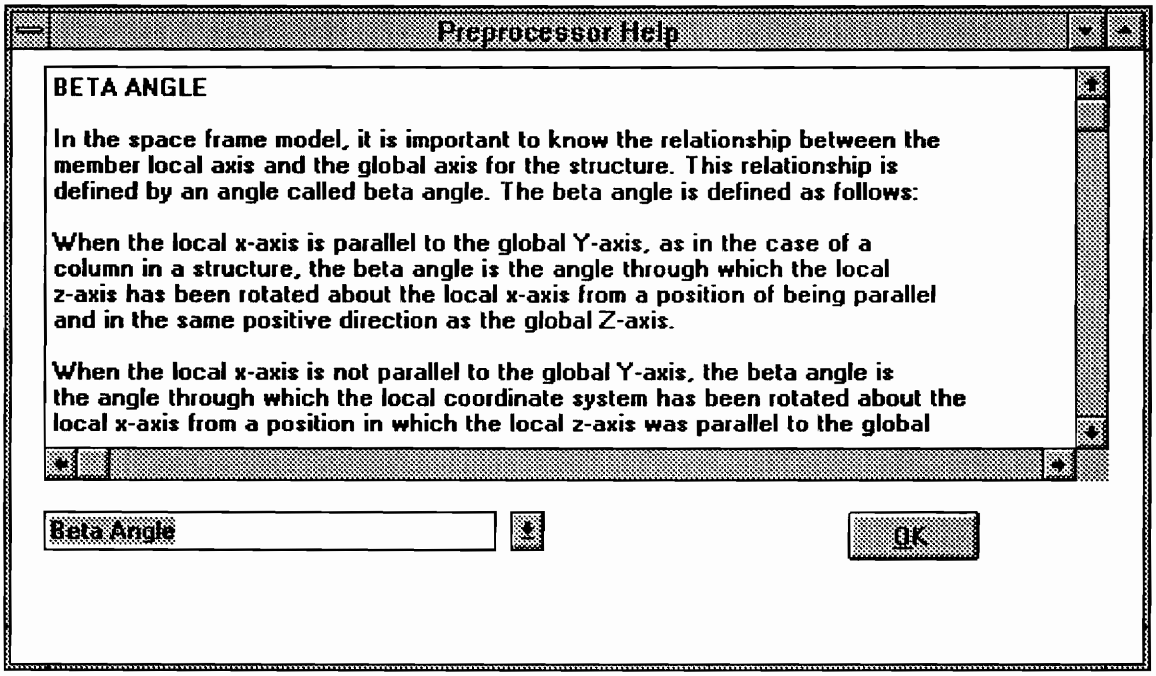


Fig. 3.32 Index help form

3.8 ABOUT

About form shows information about the preprocessor, the current version and the author. The about form is shown in Figure 3.33.

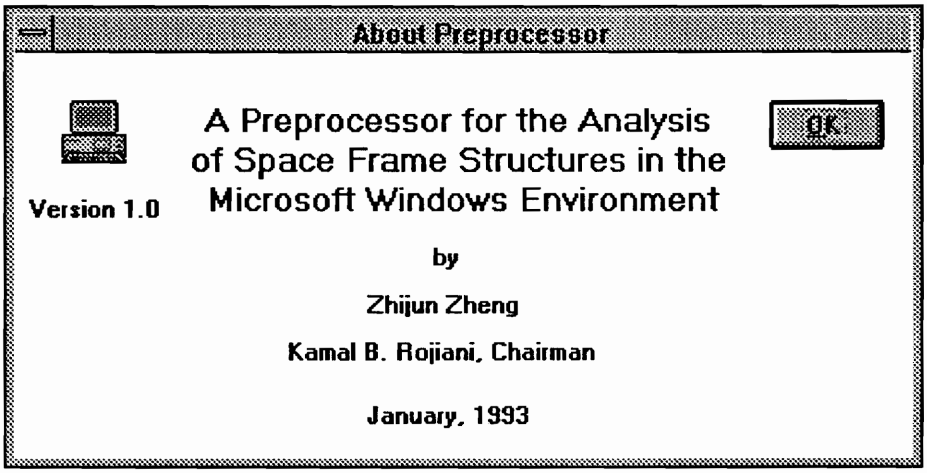


Fig. 3.33 About form

CHAPTER FOUR
SUMMARY AND CONCLUSIONS

4.1 SUMMARY

A preprocessor for the analysis of space frame structures was developed. The preprocessor is designed to run in the Microsoft Windows environment which is widely available on personal computers. The program is written in the Microsoft Visual Basic programming language.

The preprocessor allows the user to enter information regarding the geometry of the structure and applied loading. The basic functions of the preprocessor include the ability to read and write files containing structural data, and to review and modify information regarding the structure. The preprocessor can create and read data files for the plane truss, plane frame, space truss and space frame structures.

The program is fully event-driven and the user has the complete control over the order of tasks to be performed. The preprocessor provides a user-friendly interface and takes advantage of standard Windows graphical user interface elements such as pull down menus, dialog boxes, scroll bars, list boxes, command buttons and radio buttons. A sophisticated data checking mechanism reduces data-entry errors to a minimum. The preprocessor also features three-dimensional graphics which can display the structural geometry and applied loads.

The preprocessor features a file menu that is standard in Windows applications. It provides the option for creating a new structural model, modifying an existing structural model, and saving a structural model on disk as a text file in a format that is suitable for use by an analysis program.

A series of input forms are provided for entering information regarding the structural model. These input forms are accessed by selecting the corresponding menu item in the main form. The preprocessor also provides for two systems of units, English and SI. An AISC shape data base which contains all sections listed in the AISC Manual is used to assign geometric properties to the members. Automatic generation of member incidences and joint coordinates is also available.

To ensure the accuracy of the structural model, each data entry form features a sophisticated data checking mechanism which performs a series of checks on the input data. It provides warning messages for most of the common errors that can occur when entering the structural model such as missing member incidences, missing joint numbers and joint coordinates, duplicate input and invalid joint numbers or member numbers.

In addition to the data checking mechanism, the preprocessor also features a tabular data review window and a three-dimensional graphics display window. The tabular data review window can be used to display all the input data simply by selecting the corresponding menu item. The three dimensional graphics displays the structural geometry and applied loads.

The preprocessor also has a context-sensitive on-line help system. The help system provides information on input parameters, input procedures for each input form, and explains the function of the various controls used in the preprocessor.

4.2 CONCLUSIONS

The preprocessor performs well in creating small and medium size structural models. It is a valuable tool for introducing the basic concepts of computer analysis of structures to beginners. Data files generated by the preprocessor for several types of structural models have been tested by the processor program and have been proved to be acceptable. However, the preprocessor does require that the user enter detailed information regarding the structural model through various input forms. This makes it impractical for creating large structural models.

The program is a good illustration of the use of the Visual Basic programming language for developing Windows applications. The Visual Basic programming environment provides a good solution to the otherwise difficult and tedious task of developing Windows applications using a procedural language such as C. Suggestions for future enhancements to the preprocessor are discussed in the next section.

4.3 FUTURE MODIFICATIONS

A possible future modification to the preprocessor would be the enhancement of the current three-dimensional graphics output. Since the primary objective of the project was to create a user-friendly preprocessor in the Microsoft Windows environment, the current graphics output only shows essential information such as joint numbers, member numbers and applied loads. It would be useful to develop new graphics output subroutines that provide better display quality and allow for rotation of the structure.

Another possible modification would be to combine the preprocessing, structural analysis and post-processing features into one integrated program. The main form has the menus that contain all the functions for structural analysis so that the analysis can be performed in an integrated program, without having to exit the preprocessor, run the processor and then run the post-processor. The information regarding the analysis of structure such as input data file, structural geometry and the results of the analysis in graphics can then be printed by selecting the corresponding menu items.

The preprocessor is very useful for entering data for small and medium size structural models and is a significant improvement over the traditional character based approach to entering data. However, for large structures it would be time consuming to use any preprocessor that relies primarily on keyboard input for entering the data for the structural model. For large structures, it would be more efficient to have a graphics based preprocessor with features similar to those currently available in CAD packages. This would provide the capability of defining members, joints and loads by using a pointing device such as a mouse. The capabilities of the CAD engine could then be used to perform many functions, such as generating the mesh, copying members and joints, assigning member properties to individual members or groups of members, deleting members and applying loads. Since all of the work would be conducted in a graphical environment, there would be constant visual feedback which would significantly reduce the possibility of input errors.

References

Craig, John Clark (1991), Visual Basic Workshop, Microsoft Press, Redmond, Washington.

Weaver and Gere (1990), Matrix Analysis of Framed Structured, third edition. Van Nostrand Reinhold, New York, New York.

AISC (1986), Load and Resistance Factor Design, 1st edition, AISC, Chicago, Illinois.

Microsoft Visual Basic Language Reference (1991), Microsoft Coorpration, Redmond, Washington.

Zivkovic, Slobodan and Rojiani, K. B. (1993), An Interactive Program for the Analysis of Space Frame Structures (to appear), VPI&SU, Blacksburg, Virginia

Pugh, James Christopher (1989), User Friendly Preprocessor for Plane and Space Frames and Space Truss, VPI&SU, Blacksburg, Virginia.

Erkek, Mehmet Erkan (1991), Structural Analysis of Plane Frame in an Interactive X Window Environment, VPI&SU, Blacksburg, Virginia.

APPENDIX A

USER'S GUIDE

This section provides information regarding the use of the preprocessor. It presents a brief overview of the basic operations that are common to most Windows programs and then presents a description of the various input forms that are provided in the preprocessor.

A.1 HARDWARE AND SOFTWARE REQUIREMENTS

A.1.1 HARDWARE

The preprocessor will run on an IBM compatible personal computer with a 80286 or higher processor and a minimum of 1 MB RAM. However, a 80386 processor, 4 MB RAM and a mouse are strongly recommended.

A.1.2 SOFTWARE

The preprocessor runs under Microsoft Windows Version 3.0 or higher. The Microsoft Visual Basic dynamic link library VBRUN100.DLL is also required.

A.1.3 INSTALLING THE PREPROCESSOR

Copy the executable file named Prep.exe from the floppy disk to the hard disk. Use the New menu item in the File menu of the Windows Program Manager application to create a new Program Item. You will need to specify the path of the executable file and the description of the preprocessor. You can also use the Browse option to locate the file. Once the icon for the preprocessor appears in your selected program group, you can run the preprocessor by double clicking on the icon.

Copy the directories AISC, HELP which contain the AISC shape data base and help text to the hard drive preferably in the same sub directory as the executable file. The supporting files can also be installed in another sub directory. The preprocessor will allow you to specify the path for these supporting files.

A.2 BASIC SKILLS

The preprocessor uses the same controls as the standard Windows controls. These controls include drop down menus, command buttons, radio buttons, check boxes, scroll bars, list boxes, combo boxes, file list box, directory list box and drive list box. Text boxes are used to accept the data.

A.2.1 USING STANDARD CONTROLS

A.2.1.1 MENU

A menu in the preprocessor, which drops down from the menu bar, initiates tasks such as opening a file or entering data, depending on the menu item selected.

To select a menu item, simply click the menu name, or press the Alt key in combination with the underlined letter of the menu name. To cancel a menu item, click outside the menu or press the Escape (Esc) key.

To choose an item from a drop down menu, you need to first select the menu, then click on the menu item or type in the underlined letter of the menu item name. Figure A.1 shows the main form of the preprocessor with menus and submenus.

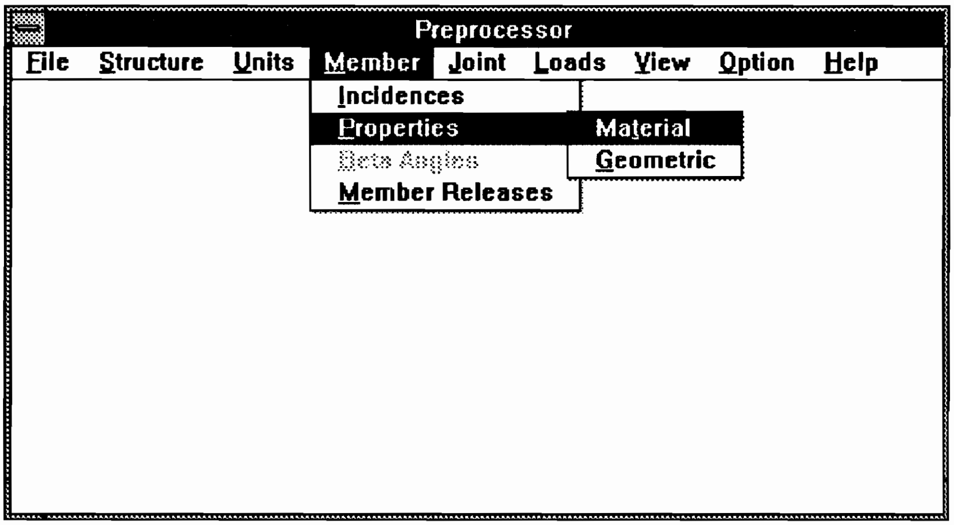


Fig. A.1 Menus and submenus

A.2.1.2 RADIO BUTTONS

Radio buttons display options that can be toggled on or off. Usually, radio buttons are used as part of a group to display multiple choices from which a selection can be made.

To select a radio button, move the mouse cursor over the radio button and click the left mouse button, or press the Alt key in combination with the underlined letter of the radio button name. A selected radio button is indicated by a solid circle in the center. Only one radio button can be selected from a group of radio buttons. When one button is selected, the other radio buttons in the group are automatically cancelled. Figure A.2 shows a group of radio buttons.

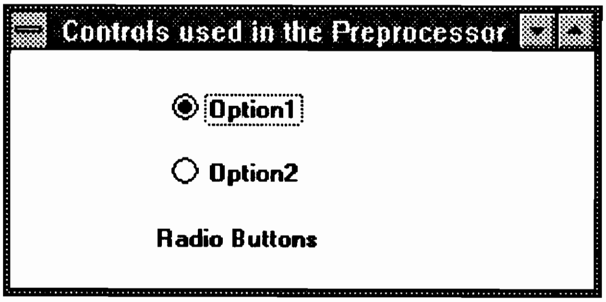


Figure A.2 Radio buttons

A.2.1.3 COMMAND BUTTONS

A command button performs a specific task as indicated by the name specified on the button. A command button is selected by moving the mouse cursor over the button and then clicking the left mouse button. A command button can also be selected by pressing the Alt key in combination with the underlined the letter of the command button name. Figure A.3 shows a command button.

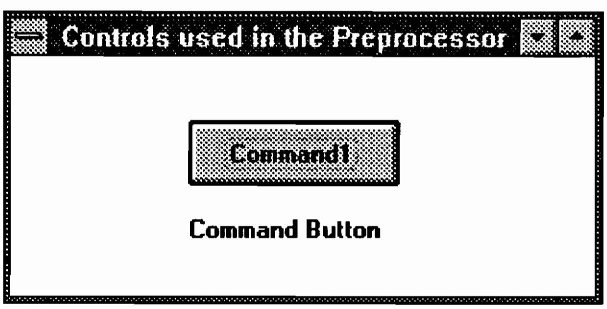


Fig A.3 Command button

A.2.1.4 CHECK BOXES

A check box displays an option that can be turned on or off. It has two states which are on and off (or yes and no). To select a check box, simply move the mouse cursor over the check box and click the left mouse button. A check box can also be selected by pressing the Alt key in combination with the underlined letter of the check box name. A selected check box is indicated by an "X". More than one check boxes can be selected in a group. The same procedure is used to clear a check box. Figure A.4 shows two check boxes (checked and unchecked).

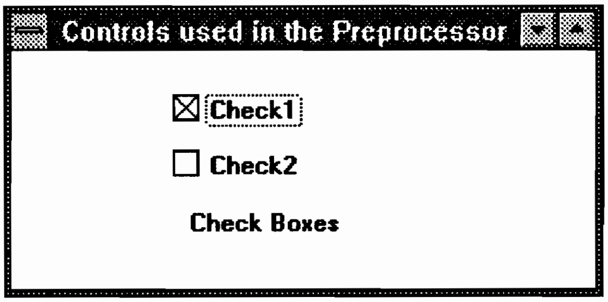


Fig A.4 Check boxes (Checked and unchecked)

A.2.1.5 SCROLL BARS

Scroll bars are graphical tools for quickly navigating through a long list of items, and for indicating the current position on a scale. A horizontal scroll bar allows movement in the horizontal direction while a vertical scroll bar allows scrolling up or down a list. To use the scroll bar, simply click the arrow which indicates the direction you want, or click the indicator on the scale and move it right/left or up/down. Figure A.5 shows both horizontal and vertical scroll bars.

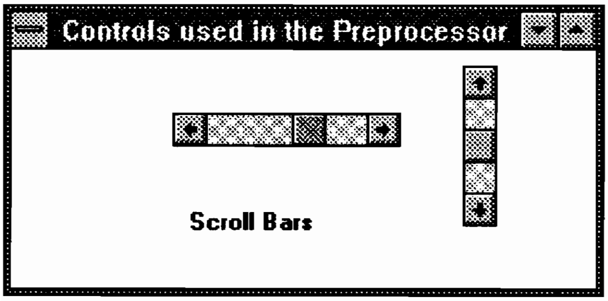


Fig A.5 Scroll bars (Horizontal and vertical)

A.2.1.6 LIST BOXES

A list box displays a list of items from which a selection can be made. A scroll bar is added to the list box if the number of items exceeds the number of items that can be displayed. To select an item in the list box, move the mouse cursor over the item and press the left mouse button. An item in the list box can also be selected by using the arrow keys to move up or down and then pressing the Enter key. Figure A.6 shows a list box.

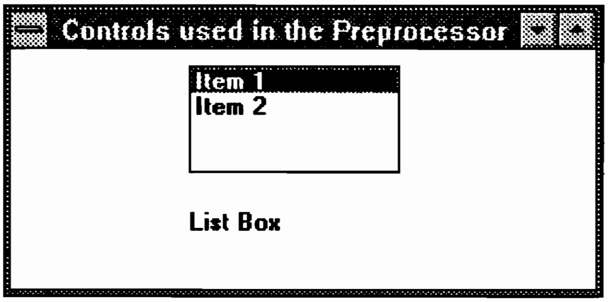


Fig A.6 List box

A.2.1.7 COMBO BOXES

A combo box combines the features of a text box and a list box. Use the combo box to make a selection by typing text into a text box or by moving the mouse cursor over the item and pressing the left mouse button. Figure A.7 shows a combo box.

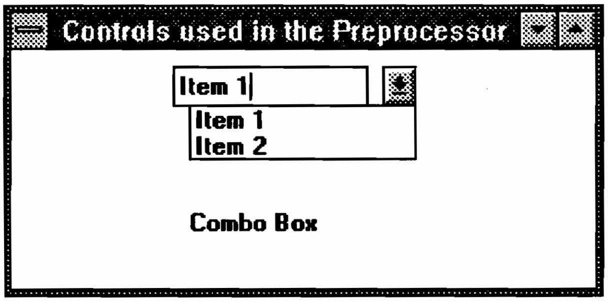


Fig. A.7 Combo box

A.2.1.8 FILE LIST BOX

A file list box is used for locating and selecting files to be opened or saved. A scroll bar is added to the file list box if the number of files exceeds the number of files that can be displayed. A file is selected by moving the mouse cursor over the file name in the file list box and then clicking the left mouse button. Figure A.8 shows a file list box.

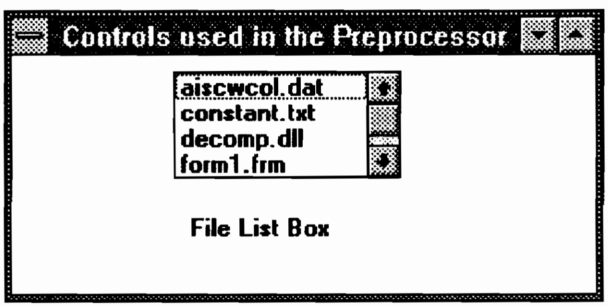


Fig A.8 File list box

A.2.1.9 DIRECTORY LIST BOX

A directory list box displays a list of sub directories. A scroll bar is added if necessary. A directory can be select by moving the mouse cursor over the directory name in the directory list box and then clicking the left mouse button. Figure A.9 shows a directory list box.

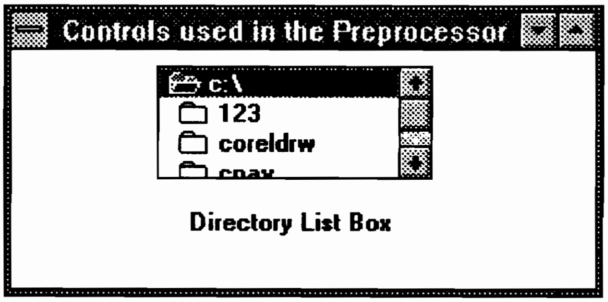


Fig A.9 Directory list box

A. 2.1.10 DRIVE LIST BOX

A drive list box displays a list of valid disk drives. A drive is selected by moving the mouse cursor over the drive name in the drive list box and then clicking the left mouse button. Figure A.10 shows a drive list box.

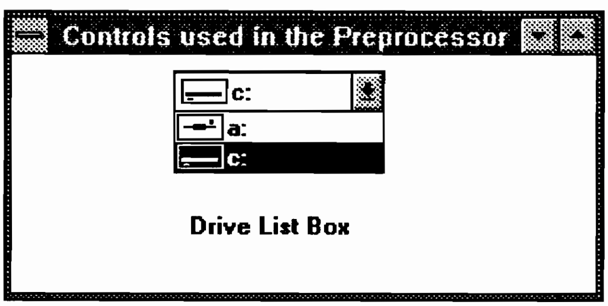


Fig A.10 Drive list box

A.2.2 ENTERING DATA IN TEXT BOXES

All edit fields in the preprocessor are followed by input prompts. Except for the option buttons and check boxes which are used to accept multiple choices, text boxes are used to enter all numeric and textual data.

Data can be entered in a text box when it has the input focus. A text box has the input focus if the text is highlighted. Pressing the Tab key moves the focus to the next control item in the form. Pressing the Enter key moves the focus among a group of text boxes only. However, moving the mouse cursor to the text box and clicking the left mouse button will also move the input focus to the text box. After the data has been entered, press the Enter key to move to the next item.

All text boxes in the preprocessor which require data entry have a single border. Text boxes without borders are used for displaying information.

A.2.3 COMMAND BUTTONS

All input forms are equipped with several standard command buttons. These command buttons perform a similar function in all of the input forms.

A.2.3.1 THE OK BUTTON

The OK button closes the input form after all data for this input form has been entered. In input forms such as the joint coordinates input form and the member incidences input form where data is critical for the accuracy of the model, the OK button also performs an automatic check to ensure that all data has been entered. In addition, if the input mode is "New", the OK button enables the menu for the next input form.

A.2.3.2 THE CANCEL BUTTON

The Cancel button allows the user to close the input form at any time. Pressing the Esc key will also close the input form.

A.2.3.3 THE HELP BUTTON

When the Help button is selected, a window containing instructions on basic input procedures for the input form is displayed.

A.2.3.4 THE LIST BUTTON

The List button opens a form which allows the user to review the input data without having to quit the current input form. When the data review form is closed, the current input form is activated again.

A.2.3.5 THE UPDATE BUTTON

The Update button is designed for accepting the data after the user has entered a set of data items. It also initiates automatic checking for input errors. If an error is detected, a message box appears on the screen to indicate the type of error and the user is provided with an opportunity to correct the error.

A.2.3.6 THE DELETE BUTTON

Delete button is provided for editing data. It will erase data currently displayed on the screen.

A.2.3.7 THE NEXT AND PREV. BUTTON

The Next and Prev. buttons are provided wherever necessary. These buttons allow the user to view previous or next data.

A.2.3.8 THE NEXT AND PREV. LOAD CASE BUTTON

The Next and Prev. Load Case buttons are used in the load input forms and in the graphics output form. These buttons allow you to select the particular load case of interest.

A.3 BASIC OPERATION

A.3.1 FILE PATH FOR SUPPORTING FILES

When you start the preprocessor for the first time, the first form that appears on the screen is the form for entering the file path for supporting files. Enter the file path for the AISC data base file in the first text box, for example as "C:\", then press the Enter key. Enter the file path for the help file in the second text box. Click on the OK button or press the Alt key in combination with the O key to close this form. The preprocessor will then create a file called "FILE.INI" in the C drive. The next time you start the preprocessor, the program will automatically read this file. The file path input form will not appear if "FILE.INI" exists in C drive.

A.3.2 MAIN FORM

The main form has the caption "Preprocessor". It has a menu bar which contains of all the menus for the various operations that can be performed.

A.3.3 CREATING A NEW MODEL OR MODIFYING AN EXISTING MODEL

A.3.3.1 CREATING A NEW MODEL

1. Select the File menu.

Move the mouse cursor over the menu and click the left mouse button; or press the Alt key in combination with the F key.

2. Select the New menu item in the File menu.

Drag the mouse cursor over the menu item and click the left mouse button or press the N key. A pop up window will appear to remind you to save the current data if the current file is a new file and has not been saved or if the current file has been changed and has not been saved.

A.3.3.2 TO MODIFY AN EXISTING MODEL

1. Select the File menu.

Move the mouse cursor over the menu and click the left mouse button; or press the Alt key in combination with the F key.

2. Select the Open menu item in the File menu.

Drag the mouse cursor over the menu item and click the left mouse button or press the O key. A pop up window will appear to remind you to save the current data if the current file is a new file and has not been saved or if the current file has been changed and has not been saved.

3. Click the drive list box in the Open File form to select the drive name.
4. Click the directory list box in the Open File form to select the directory name.
5. Click the file list box in the Open File form to select the file name.

You can also select the file name and directory path by entering it in the File Name text box, such as, for example "a:\test.dat".

6. Click on the OK button or press the Enter key to open the file.

You can also double click on the file name in the file list box to open the file.

A.3.4 STRUCTURE TYPE AND SIZE

1. Move the mouse cursor over the Structure menu item and press the left mouse button; or press the Alt key in combination with the S key to open the Structural Data input form.
2. Enter the Project Description in the first text box and press the Enter key. The description is limited to one line of text.
3. Enter the total number of joints in the second text box and press the Enter key.
4. Enter the total number of members in the third text box and press the Enter key.
5. Enter the total number of load cases in the fourth text box and press the Enter key.
6. Enter the total number of load combinations in the fifth text box and press the Enter key.

7. Click on one of the radio buttons to select the type of structure from the following options: space truss, plane frame, space frame or plane truss.

8. Click on the OK button to close this form.

A.3.5 MEMBER INCIDENCES INPUT

A.3.5.1 MEMBER INCIDENCES FOR INDIVIDUAL MEMBERS

1. Select the Member menu from the main form to display the items in this pull down menu.

2. Select the Incidences menu item to open the Member Incidences input form.

3. Enter the member number in the first text box and press the Enter key.

When this form is opened, the first text box has the input focus and contains the number "1". When creating a new model, it is recommended that you enter member incidences starting from the first member to the last member by pressing the Enter key to skip this text box. When modifying an existing model, the text in the member number edit box can be changed as needed.

4. Enter the start joint number in the second text box and press the Enter key.

5. Enter the end joint number in the third text box.

6. Click on the Update button or press the Alt key in combination with the U key to update data.

7. Repeat Steps 3 to 6 until all member incidences have been entered.

8. Click on the OK button or Press the Alt key in combination with the O key to close this form.

A.3.5.2 MEMBER INCIDENCES AUTO-GENERATION

1. Select the Member menu from the main form to display the items in this pull down menu.

2. Select the Incidences menu item to open the Member Incidences input form.

3. Select the Auto menu item to open the Member Incidences Auto-Generation form.

4. Enter the member number of the first member in the group of members in the first text box and press the Enter key.

5. Enter the member number of the last member in the group of members in the second text box and press the Enter key.

6. Enter the increment in the member number in the third text box and press the Enter key.

7. Enter the start joint number of the first member in the group in the fourth text box and press the Enter key.

8. Enter the end joint number of the first member in the group in the fifth text box and press the Enter key.

9. Enter the value for the increment in member incidences in the sixth text box.

10. Click on the Update button to update the data.

11. Repeat Steps 4 to 9 to generate member incidences for the members in another group.

12. Click on the OK button to return to the Member Incidences input form.

A.3.6 MEMBER PROPERTIES INPUT

A.3.6.1 MATERIAL PROPERTIES INPUT

1. Select the Member menu from the main form to display the items in this pull down menu.

2. Select the Properties menu item to display the Properties sub menu.

3. Select the Material menu item to open the Material Properties input form.

4. Enter the member number of the first member in the group of members in the first text box and press the Enter key.

5. Enter the member number of the last member in the group of members in the second text box and press the Enter key.

6. Select one of the radio buttons to choose the material type from the following choices: steel, concrete or user-defined.

If steel is selected:

The modulus of elasticity, shear modulus and coefficient of thermal expansion are displayed in the corresponding text boxes after this operation.

If concrete is selected:

6.1 Enter the compressive strength of the concrete in the text box and press the Enter key. The modulus of elasticity, shear modulus and coefficient of thermal expansion are computed and displayed in the corresponding text boxes after this operation.

If user-defined is selected:

6.1 Enter the modulus of elasticity in the corresponding text box and press the Enter key.

6.2 Enter the shear modulus in the corresponding text box and press the Enter key.

6.3 Enter the coefficient of thermal expansion in the corresponding text box and press the Enter key.

7. Click on the Update button or press the Alt key in combination with the U key to update the data.

8. Repeat Steps 4 to 7 to enter material properties for all members of the structure.

9. Click on the OK button or press the Alt key in combination with the O key to close this form.

Selecting Single mode in the Option menu allows you to enter the material properties for an individual member. In this case, simply enter the member number in Step 4 and skip Step 5.

A.3.6.2 GEOMETRIC PROPERTIES INPUT

1. Select the Member menu from the main form to display the items in this pull down menu.

2. Select the Properties menu item to display the Properties sub menu.

3. Select the Geometry menu item to open the Geometric Properties input form.

4. Enter the member number of the first member in the group of members in the first text box and press the Enter key.

5. Enter the member number of the last member in the group of members in the second text box and press the Enter key.

6. Select the radio button corresponding to the following choices: AISC shape assignment, or User Defined. Note that if the SI system of units is used, then the radio button for AISC shape assignment is grayed.

If AISC is selected:

7.1 Select the desired shape from the list of shapes that appear in the list box.

7.2 Select the desired section by moving the mouse cursor over the item in the combo box and clicking the left mouse button or simply typing in the desired section in the edit field of the combo box and pressing the Enter key. A message box appears on the screen to display an error message if the section entered is not included in the AISC shape data base.

The geometric properties of the section are displayed in the corresponding text boxes after this operation.

If User Defined is selected:

7.1 Enter the cross-sectional area in the corresponding text box and press the Enter key.

7.2 Enter the moment(s) of inertia according to the type of structure.

8. Click on the Update button or press the Alt key in combination with the U key to update the data.

9. Repeat Steps 4 to 8 to enter the geometric properties for the remaining members.

10. Click on the OK button or press the Alt key in combination with the O key to close this form.

If the Single mode option was selected, simply enter the member number in Step 4 and skip Step 5.

A.3.7 BETA ANGLE INPUT

1. Select the Member menu from the main form to display the items in this pull down menu.

2. Select the Beta Angle menu item from the pull down menu to open the Beta Angle input form.

3. Enter the member number in the first text box and press the Enter key.

4. Enter the beta angle in the second text box.

5. Click on the Update button or press the Alt key in combination with the U key to update the data.

6. Repeat Steps 3 to 5 to enter the beta angle for the rest of the members.

Note that the default value of the beta angle is zero, so it is only necessary to enter the beta angle for those members for which the beta angle is not zero.

7. Click on the OK button or press the Alt key in combination with the O key to close the Beta Angle input form and return to the main form.

A.3.8 MEMBER RELEASES INPUT

1. Select the Member menu from the main form to display the items in this pull down menu.

2. Select the Member Releases menu item from the pull down menu to open the Member Releases input form.

3. Enter the member number in the text box.

4. Select the appropriate check boxes to enter member releases.

5. Click on the Update button or press the Alt key in combination with the U key to update the data.

6. Repeat Steps 3 to 5 to enter all the member releases in the model.

7. Click on the OK button or press the Alt key in combination with the O key to close the Member Releases input form and return to the main form.

A.3.9 JOINT COORDINATES INPUT

1. Select the Joint menu from the main form to display the items in this pull down menu.

2. Select the Joint Coordinates menu item from the pull down menu to open the Joint Coordinates input form.

3. Enter the joint number in the first text box and press the Enter key.

4. Enter the global X coordinate in the second text box and press the Enter key.

5. Enter the global Y coordinate in the third text box and press the Enter key.

6. Enter the global Z coordinate in the fourth text box.

Note that the global Z coordinate is only required for the space truss and the space frame models.

7. Click on the Update button or press the Alt key in combination with the U key to update the data.

8. Repeat Steps 3 to 7 to enter the joint coordinates for the remaining joints.

9. Click on the OK button or press the Alt key in combination with the O key to close the Joint Coordinates input form and return to the main form.

A.3.10 AUTO-GENERATION OF JOINT COORDINATES

1. Select the Joint menu from the main form to display the items in this pull down menu.

2. Select the Joint Coordinates menu item from the pull down menu to open the Joint Coordinates input form.

3. Select the Auto menu item to open the joint coordinates auto-generation form.

4. Enter the joint number of the first joint in the series in the first text box and press the Enter key.

5. Enter the joint number of the last joint in the series in the second text box and press the Enter key.

6. Enter the value for the increment in the joint number in the third text box and press the Enter key.

7. Enter the coordinates for the first joint in the corresponding text box and press the Enter key.

8. Enter the value for the increments in the coordinates in the corresponding text boxes and press the Enter key.

9. Click on the Update button or press the Alt key in combination with the U key to update the data.

10. Repeat Steps 4 to 9 to generate the joint coordinates for the remaining joints.

11. Click on the OK button or press the Alt key in combination with the O key to close the Joint Coordinate Auto-Generation form and return to the Joint Coordinates input form.

A.3.11 JOINT CONSTRAINTS INPUT

1. Select the Joint menu from the main form to display the items in this pull down menu.

2. Select the Joint Constraints menu item from the pull down menu to open the Joint Constraints input form.

3. Enter the joint number of the constrained joint and press the Enter key.

4. Select the appropriate radio button to select joint constraints.

Note that four separate input forms are used to input joint constraints because of the different structure types. The check boxes under the Other (user defined) option are enabled automatically after the Other (user defined) option has been selected.

5. Click on the Update button or press the Alt key in combination with the U key to update the data.

6. Repeat Steps 3 to 5 to enter the remaining joint constraints.

7. Click on the OK button or press the Alt key in combination with the O key to close the Joint Constraints input form and return to the main form.

A.3.12 LOADS

A.3.12.1 LOAD CASE DESCRIPTIONS

1. Select the Loads menu from the main form to display the items in this pull down menu.
2. Select the Load Case Descriptions menu item from the pull down menu to open the Load Case Descriptions input form.
3. Enter the load case number in the first text box and press the Enter key.
4. Select from the items in the list box to enter a preset load case description by double clicking the left mouse button or enter a load case description in the second text box and press the Enter key.
5. Click on the Update button or press the Alt key in combination with the U key to update the data.
6. Repeat Steps 3 to 5 to enter all load case descriptions.
7. Click on the OK button or press the Alt key in combination with the O key to close the Load Case Descriptions input form and return to the main form.

A.3.12.2 LOAD COMBINATIONS

1. Select the Loads menu from the main form to display the items in this pull down menu.
2. Select the Load Combinations menu item from the pull down menu to open the Load Combinations input form.
3. Enter the load combination number in the first text box and press the Enter key.
4. Select the load case from the list box containing load case descriptions.
5. Enter the corresponding load factor for this load case in the text box to the right of the load description and press the Enter key.
6. Repeat Steps 4 to 5 to enter all load cases and the corresponding load factors.
7. Click on the Update button or press the Alt key in combination with the U key to update the data.

8. Click on the OK button or press the Alt key in combination with the O key to close the Load Combinations input form and return to the main form.

When editing an existing data file in which the load case descriptions have been changed, the previous load cases and the corresponding load factors appear in the load combinations when the Load Combinations input form is opened. In this case, click on the Delete button in the Load Combinations input form to delete the previous load cases and the corresponding load factors. To delete a load case from a load combination, simply select the load case from the list box and click on the Delete button.

A.3.12.3 JOINT LOADS

1. Select the Loads menu from the main form to display the items in this pull down menu.
2. Select the Joint Loads menu item from the pull down menu to open the Joint Loads input form.
3. Enter the load case number in the first text box and press the Enter key.

Load case numbers begin from 1 to the total number of load cases in the model. Use the Next/Prev. Load Case button to increase or decrease the load case number or select the text box to enter a load case number.

4. Enter the joint load number in the second text box and press the Enter key.

The joint load number is the number of joint loads in each load case. It is generated automatically. This text box can be skipped by pressing the Enter key.

5. Enter the joint number of the joint at which the load is applied in the third text box and press the Enter key.
6. Enter the magnitude of joint load (force or moment) in the fourth text box.
7. Select the radio button corresponding to the type of joint load (force or moment).
8. Select the radio button corresponding to the direction in which the load is applied (global X, Y or Z direction).

9. Click on the Update button or press the Alt key in combination with U key to update the data.

10. Repeat Steps 4 to 9 to enter all joint loads for this load case.

11. Click on the Next Load Case button or press the Alt key in combination with the E key to proceed to the next load case.

12. Repeat Steps 4 to 11 to enter joint loads for all load cases.

13. Click on the OK button or press the Alt key in combination with the O key to close the Joint Loads input form and return to the main form.

A.3.12.4 MEMBER LOADS

1. Select the Loads menu from the main form to display the items in this pull down menu.

2. Select the Member Loads menu item from the pull down menu to display the items in the Member Loads sub menu.

3. Selecting the menu item corresponding to the type of member load: Concentrated, Uniform, Linear or Temperature Change.

Since the input procedures for the input forms for all member loads are similar, only the input form for linear load is described below.

4. Enter the load case number in the first text box and press the Enter key.

5. Enter the member number in the second text box and press the Enter key.

6. Enter the magnitude w1 in the third text box and press the Enter key.

7. Enter the distance d1 in the fourth text box and press the Enter key.

8. Enter the magnitude w2 in the fifth text box and press the Enter key.

9. Enter the distance d2 in the sixth text box.

10. Select the radio button corresponding to the direction of load.

11. Click on the Update button or press the Alt key in combination with the U key to update the data.

12. Repeat Steps 4 to 12 to enter linear loads for all load cases.

13. Click on the OK button or press the Alt key in combination with the O key to close the Linear Load input form and return to the main form.

A.3.12.5 PRESCRIBED DISPLACEMENTS

1. Select the Load menu from the main form to display the items in this pull down menu.

2. Select the Prescribed Displacements menu item from the pull down menu to open the Prescribed Displacements input form.

3. Enter the load case number in the first text box and press the Enter key.

Load case numbers begin from 1 to the total number of load cases in the model. Use the Next/Prev. Load Case button to increase or decrease the load case number or select the text box to enter a load case number.

4. Enter the prescribed displacement number in the second text box and press the Enter key.

The prescribed displacement number is the number of prescribed displacements in each load case. It is generated automatically. This text box can be skipped by pressing the Enter key.

5. Enter the joint number of the joint subjected to the prescribed displacement in the third text box and press the Enter key.

6. Enter the magnitude of the prescribed displacement in the fourth text box.

7. Select the radio button corresponding to the type of prescribed displacement (translation or rotation).

8. Select the radio button corresponding to the direction of the prescribed displacement (global X, Y or Z direction).

9. Click on the Update button or press the Alt key in combination with U key to update the data.

10. Repeat Steps 4 to 9 to enter all joint load for this load case.

11. Click on the Next Load Case button or press the Alt key in combination with the E key to proceed to the next load case.

12. Repeat Steps 4 to 11 to enter prescribed displacements for all remaining load cases.

13. Click on the OK button or press the Alt key in combination with the O key to close the Prescribed Displacements input form and return to the main form.

A.3.13 DATA REVIEW

A.3.13.1 NUMERICAL DATA REVIEW

1. Select the View menu from the main form to display the items in this pull down menu.
2. Select the Data menu item from the pull down menu to open the Data Review window.
3. Select the type of information to be reviewed, for example, joint coordinates, joint loads, member incidences, etc.
4. Click on the More button or press the Alt key in combination with the M key to display more data.

If there is more data to be reviewed, a message is displayed on the screen to remind the user to click the More button.

5. Repeat Steps 3 to 4 to view other data.
6. Click on the OK button or press the Alt key in combination with the O key to close the Data Review window and return to the main form.

The Data Review form can also be opened by clicking the List button in any of the data entry forms.

A.3.13.2 GRAPHICS

A.3.13.2.1 DISPLAY

1. Select the View menu from the main form to display the items in this pull down menu.

2. Select the Graphics menu item from the pull down menu to open the Graphics Output window.
3. Select the Structure menu to display the structure mesh.

Note that the structure mesh must be displayed first.

When the structure mesh is displayed on the screen, the joint numbers and member numbers can be displayed by clicking the corresponding command buttons and the loads can be displayed by selecting the corresponding menu items. The menu items will be grayed if there are no loads for the given load case. Use the Next and Prev. Load Case buttons to select the load cases.

A.3.13.2.2 CLEARING THE CURRENT GRAPHICS DISPLAY

The currently displayed graphical information can be cleared in several ways. The menu items and command buttons which are used to display the information also serve as toggle switches for turning on/off the display. Therefore, each type of load can be cleared by clicking the corresponding menu item a second time. The member numbers and joint numbers can also be cleared by clicking the corresponding command buttons a second time. A check mark on the left side of the menu item for a load indicates that the load is currently displayed. The user can also click the Structure menu to clear all graphical information currently displayed on the screen and redraw the structure mesh. The Graphics Output form also provides a Clear button to clear the display.

The Graphics Output form can be closed by clicking on the OK button or pressing the Alt key in combination with the O key.

A.3.14 SAVING THE STRUCTURAL MODEL

To save the current structural data:

1. Select the File menu from the main form to display the items in this pull down menu.

2. Select the Save menu item to save the file.

The Save function saves an existing file only. If the file is a new file and the Save menu item is select, the Save As... form will be opened. See Section A.3.15 for more information.

A.3.15 SAVE AS...

To save the current data in a different file:

1. Select the File menu from the main form to display the items in this pull down menu.
2. Select the Save As... menu item to open the Save As... form.
3. Click on the item in the drive list box to select the drive name.
4. Click on the item in the directory list box to select the directory name.
5. Type the file name in the File Name text box and press the Enter key or click on the OK button.

You can also type the file path in the text box such as, for example, "a:\test\test1.dat" and then press the Enter key or click on the OK button.

The Save As function is used only if the file is a new file or the file is to be saved under a different name.

A.3.16 EXIT

1. Select the File menu from the main form to display the items in this pull down menu.
2. Select the Exit menu item in the pull down menu to exit the preprocessor.

Before exiting, the preprocessor will remind you to save the file if a file has not been saved or if an existing file has been changed.

A.3.17 HELP

Help on the preprocessor is available from the Help menu on the menu bar in the main form. To obtain help:

1. Select the Help menu from the main form to display the items in this pull down menu.
2. Select the Index menu item to open the Help form.
3. Click on the arrow on the right side of the combo box to open the list of help topics.
4. Click on the topic in the combo list or type in the name of item in the edit field of the combo box and press the Enter key.
5. Use the vertical scroll bar to review the help text.
6. Repeat Steps 3 to 5 to obtain help on another topic.
7. Click on the OK button or press the Alt in combination with the O key to close the Help window and return to the main form.

Help for individual forms can be obtained by clicking the Help button provided in the form. In this case, the combo box is not displayed and only the information regarding the corresponding form is provided. When the Help form is closed, the previous form is activated automatically.

APPENDIX B

EXAMPLES

In this appendix, examples of date files for plane truss, plane frame, space truss and space frame structures under various kinds of loads are presented. In addition, the format of the output data file is explained through the space frame example.

B.1 SPACE FRAME

The structure is shown in Figure B.1. The output data file text and explanation are described below:

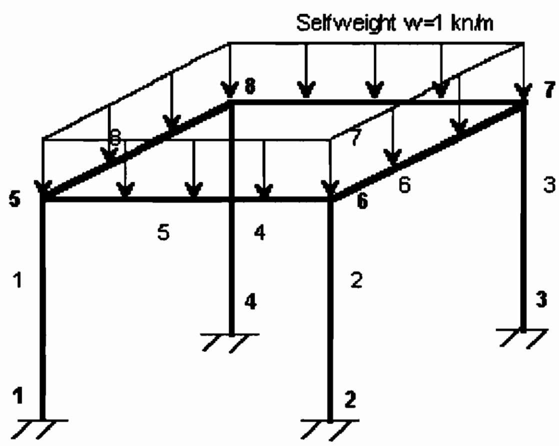


Fig B.1 Space frame structure (The member loads shown are for load case 2)

Data File Syntax	Explanations	Note
EXAMPLE PROBLEM NO.1	Project description	Any text, limit to one line
TYPE SPACE FRAME	Structure type	Other keywords: SPACE TRUSS, PLANE TRUSS, PLANE FRAME.
UNIT SYSTEM SI	keyword for units	Other keyword: US

Examples

Input	Description	Remarks
NUMBER OF JOINTS 8	Total number of joints	The total number of joints cannot be greater than 100 or less than 0.
NUMBER OF MEMBERS 8	Total number of members	The total number of members cannot be greater than 100 or less than 0.
NUMBER OF SUPPORTS 4	Total number of supporting joints	This number is determined by the joint constraints entered.
NUMBER OF LOADINGS 4	Total number of load cases	The total number of load cases cannot be greater than 10 or less than 0.
JOINT COORDINATES	Keyword for the processor program	The order of the keywords that describe the geometry of the model is random.
1 0 0 0 S 2 10 0 0 S *3 10 0 -10 S* 4 0 0 -10 S 5 0 10 0 6 10 10 0 7 10 10 -10 8 0 10 -10	Joint coordinates for all joints The syntax for the line with Italian font are explained in the Note.	"*3*" is the joint number, "*10*", "*0*" and "*-10*" are the joint coordinates in the global X, Y and Z directions. "*S*" means this joint is a support.
MEMBER INCIDENCES	Keyword for the processor program	
1 1 5 2 2 6 3 3 7 4 4 8 5 5 6 6 6 7 7 7 8 8 5 8	Member incidences for all members The syntax for the line with Italian font are explained in the Note.	"*1*" is the member number, "*1*" and "*5*" are the joint numbers of the start joint and end joint respectively.
JOINT RELEASE* (Note that there is no joint release in this example, the keywords here and below are for demonstration purpose only)	Keywords for joint release.	

Examples

1 FORCE X	Description of joint release.	"1" is the joint number, "FORCE X" is the direction of the joint release. Other keywords are "FORCE Y", "FORCE Z", "MOMENT X", "MOMENT Y" and "MOMENT Z".
MEMBER RELEASE* (Note that there is no member release in this example, the keywords here and below are for demonstration purpose only)	Keywords for member release.	
2 START MOMENT Z	Description of member release .	"2" is the member number, "START" specifies the location of the release. Other keywords are "END". "MOMENT Z" is the direction of member release. Other syntax are the same as joint release.
MEMBER PROPERTIES PRISMATIC	Keyword for the processor program	
1 AX IX IY IZ BETA 1.4 0 .228 .116 0 2 AX IX IY IZ BETA 1.4 0 .228 .116 0 3 AX IX IY IZ BETA 1.4 0 .228 .116 0 4 AX IX IY IZ BETA 1.4 0 .228 .116 0 5 AX IX IY IZ BETA 5.2 0 2.92 1.73 0 6 AX IX IY IZ BETA 5.2 0 2.92 1.73 0 7 AX IX IY IZ BETA 5.2 0 2.92 1.73 0 8 AX IX IY IZ BETA 5.2 0 2.92 1.73 0	Geometric properties of members. The keywords in Italian font are explained in the Note.	"*1*" is the member number, "*1.4*", "*0*", "*.228*", "*.116*" and "*0*" are the cross-sectional area, moment of inertia about the local x, y and z axes and beta angle respectively.
CONSTANTS E 200 ALL	Modulus of elasticity This statement assigns 200 Gpa ksi as the modulus of elasticity for all members.	The keywords for assigning the modulus of elasticity to individual members are "CONSTANTS E 200 N1 TO N2", where N1 and N2 are the start and end member numbers of the group.

Examples

CONSTANTS G 77 ALL	Shear modulus This statement assigns 77 Gpa as the shear modulus for all members.	The keywords for assigning the shear modulus to individual members are "CONSTANTS G 77 N1 TO N2".
CONSTANTS ALPHA 0 ALL	Coefficient of thermal expansion This statement assigns 0 as the coefficient of thermal expansion for all members.	The keywords for assigning the coefficient to individual members are "CONSTANTS ALPHA 0.0000065 N1 TO N2".
LOAD COMBINATION 1	Keywords for the processor program. .	
0.7 2 1.2 3 LOAD COMBINATION 2 0.8 2 1.3 3	Load combinations The keywords in Italian font are explained in the Note.	*0.7*" is the load factor for load case "*1*", "*1.2*" is the load factor for load case "*2*".
LOADING 1 WIND LOAD	Load case description	
JOINT LOADS	Keyword for joint loads	
5 FORCE X 10 8 FORCE X 10	Description of joint load. The keywords in Italian font are explained in the Note.	"*5*" is the joint number at which the load is acting. "*FORCE X*" is the direction of the joint load. Other directions can be "FORCE Y", "FORCE Z", "MOMENT X", "MOMENT Y" and "MOMENT Z". "*10*" is the magnitude of the joint load.
LOADING 2 DEAD LOAD		
MEMBER LOADS	Keyword for member load	

Examples

5 FORCE Y UNIFORM -1 6 FORCE Y UNIFORM 1 7 FORCE Y UNIFORM 1 8 FORCE Y UNIFORM 1	Description of member load. The syntax for the line with Italian font is explained in the Note.	"*5*" is the member number of which the member load on which the load is acting. "*FORCE Y*" is the direction of the member load. Other directions are "FORCE X" and "FORCE Y". "*UNIFORM*" is the type of member load. Other types are "CONCENTRATED", "LINEAR" and "TEMPERATURE". ".2" is the magnitude of the member load. The keywords for the linear load are "N FORCE Y W1 D1 W2 D2", where N is the member number, w1 and w2 are the magnitude of the load at the left and right side, d1 and d2 are the distance from the start jointto the start and end of the load. The keywords for temperature change are "N TEMPERATURE T", where T is the amount of temperature change.
LOADING 3 POINT LOAD	Load case description	
MEMBER LOADS 5 FORCE Y CONCENTRATED -5 5		
LOADING 4 SUPPORT DIS	Load case description	
SUPPORT DISPLACEMENTS	Keywords for the prescribed displacement	

1 FORCE X -.1	Description of prescribed displacement	keywords are the same as joint load. "FORCE X" means a translation in the global X direction and "MOMENT X" means a rotation about the global X direction.
SOLVE	Keyword indicates the end of the data file	

B.2 PLANE TRUSS

This example shows a plane truss model subjected to joint loads in two directions. The structure is shown in Figure B.2.

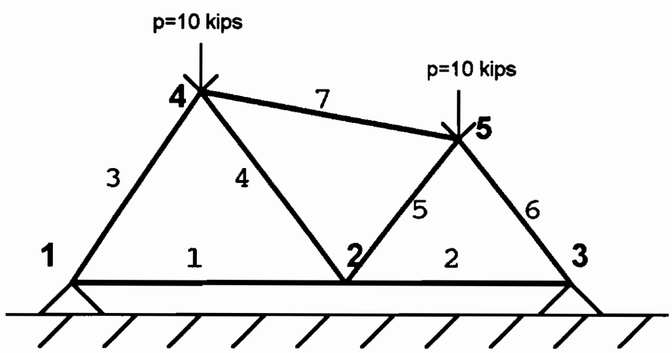


Fig B.2 Plane truss structure (The joint loads shown are for load case 1)

EXAMPLE PROBLEM NO. 2
TYPE PLANE TRUSS
UNIT SYSTEM SI
NUMBER OF JOINTS 5
NUMBER OF MEMBERS 7
NUMBER OF SUPPORTS 2
NUMBER OF LOADINGS 2
JOINT COORDINATES
1 0 0 S

```
2       5       0
3       10      0       S
4       2.5     5
5       7.5     4
MEMBER INCIDENCES
1       1       2
2       2       3
3       1       4
4       2       4
5       2       5
6       3       5
7       4       5
MEMBER PROPERTIES PRISMATIC
 1 AX .0086
 2 AX .0086
 3 AX .00432
 4 AX .00432
 5 AX .00432
 6 AX .00432
 7 AX .00246
CONSTANTS E 200 ALL
CONSTANTS ALPHA 0 ALL
LOADING 1
JOINT LOADS
 4 FORCE Y -10
 5 FORCE Y -10
LOADING 2
JOINT LOADS
 4 FORCE X  10
SOLVE
```

B.3 PLANE FRAME

This example shows a plane frame model. The structure is shown in Figure B.3.

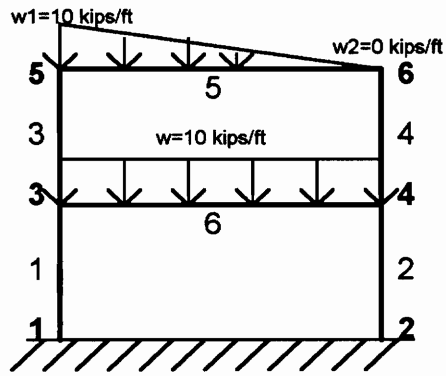


Fig B.3 Plane frame structure

EXAMPLE PROBLEM NO. 3

TYPE PLANE FRAME

UNIT SYSTEM US

NUMBER OF JOINTS 6

NUMBER OF MEMBERS 6

NUMBER OF SUPPORTS 2

NUMBER OF LOADINGS 1

JOINT COORDINATES

1	0	0	S
2	0	0	S
3	0	5	
4	10	5	
5	0	0	
6	10	10	

MEMBER INCIDENCES

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

MEMBER PROPERTIES PRISMATIC

Examples

```
 1 AX IZ 10  100
 2 AX IZ 10  100
 3 AX IZ 10  100
 4 AX IZ 10  100
 5 AX IZ 10  100
 6 AX IZ 10  100
CONSTANTS E 29000 ALL
CONSTANTS ALPHA .0000065 ALL
LOADING 1  DEAD LOAD
MEMBER LOADS
 6 FORCE Y UNIFORM -10
 5 FORCE Y LINEAR -10  0  0  10
SOLVE
```

B.4 SPACE TRUSS

This example shows a space truss model. The structure is shown in Figure B.4.

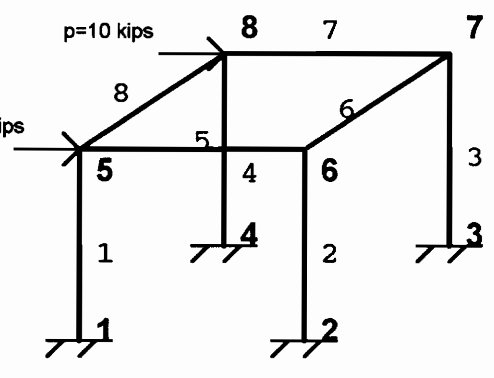


Fig B.4 Space truss structure

```
EXAMPLE PROBLEM NO. 4
TYPE          SPACE TRUSS
UNIT SYSTEM   US
NUMBER OF JOINTS      8
NUMBER OF MEMBERS     8
```

```
NUMBER OF SUPPORTS    4
NUMBER OF LOADINGS    1
JOINT COORDINATES
 1        0       0       0       S
 2        10      0       0       S
 3        10      0       -10     S
 4        0       0       -10     S
 5        0       10      0
 6        10      10      0
 7        10      10      -10
 8        0       10      -10
MEMBER INCIDENCES
 1        1       5
 2        2       6
 3        3       7
 4        4       8
 5        5       6
 6        6       7
 7        7       8
 8        5       8
MEMBER PROPERTIES PRISMATIC
 1 AX 4.16
 2 AX 4.16
 3 AX 4.16
 4 AX 4.16
 5 AX 4.16
 6 AX 4.16
 7 AX 4.16
 8 AX 4.16
CONSTANTS E 29000 ALL
CONSTANTS ALPHA .0000065 ALL
LOADING 1  WIND LOAD
JOINT LOADS
 5 FORCE X  10
 8 FORCE X  10
SOLVE
```

VITA

Zhijun Zheng was born on July 15, 1966 in Shanghai, China. He attended Shanghai Jiao Tong University in September, 1984 and received his Bachelor of Science degree in Engineering in July, 1988. In January 1991, he was enrolled in the graduate program in the Department of Civil Engineering at Virginia Polytechnic Institute and State University where he earned his Master of Engineering degree.

Zhijun Zheng