S A B R E

SIGN BRIDGE ANALYSIS AND EVALUATION SYSTEM

Bridge Engineering Software & Technology Center
Department of Civil Engineering
University of Maryland
College Park, MD

Revision 5C

April 2012

TABLE OF CONTENTS

Page

Tables

Figures

WIN-SABRE
Windows-Based Pre/Postprocessor for the Computer Analysis
of 3-D Sign Bridge

1 - INTRODUCTION

WIN-SABRE — Sign Bridge Analysis and Evaluation System runs on Windows platform personal computers and includes preprocessor, analysis, and postprocessor modules. The preprocessor includes data entry/editing, mesh generation, and on-screen graphing, among other functions. The analysis module uses the general stiffness method to perform static analysis of space frames and includes automatic load calculation (dead, wind, and ice), stress analysis, and code checking. The analysis is based upon the 2009 AASHTO "Standard Specifications for Structural Supports for Highway Signs, Luminaires and Traffic Signals" 5th Edition or the 2001 AASHTO Specs. 4th edition, for both steel and aluminum structures. (For comparison purpose, the AASHTO Specs. 3rd edition is also available, but for steel structure only.)

The postprocessor includes on-screen graphing, review of analysis results, base plate fatigue check, fatigue detail check in Excel and design of base plates and splice plates.

The American Association of State Highway and Transportation Officials (AASHTO) has developed the "Standard Specifications for Structural Supports for Highway Signs, Luminaires and Traffic Signals" to govern the design of sign structures. These specifications, Reference 1, standardize the requirements for load application, methods of analysis, allowable stresses and design details for sign supports and, as a result, have made easier the design and erection of sign supports.

All of the entry and editing of data is accomplished with the aid of windows input screens. This relieves the user of the responsibility of creating and maintaining the formatted text files required for analysis. Each screen is divided into a number of data cells which can be edited individually. The user can move freely from screen to screen and from cell to cell and can concentrate on the meaning of the data rather than whether it is entered into the proper location. Also, a number of utilities are provided to aid in the review and editing of the data, such as on-screen graphics and output file viewing.

The mesh generation capabilities cover thirty of the most common sign bridge configurations used in most of the states. This frees the user from the task of calculating and typing each structure joint and member into a data file. That data can now be generated automatically. Also, the structure solved by the stiffness method can be code-checked automatically.

2 - DESCRIPTION OF PROGRAM

2.1 PURPOSE

The primary purpose of the development of this program was to shorten and simplify the design/analysis process for sign support structures. To be used properly, though, the user must be aware of the capabilities and limitations of the program. These are described in this chapter.

2.2 SIGN BRIDGE COMPONENTS

1. Sign bridge configuration.
2. Tower types.
3. Beam types.
4. Sign Bridge types.
5. Member types.
6. Section types.
7. Sign types.
8 Walkway types.
9. Vertical attachment member (VAM) types.

These components are described in more detail in the remainder of this chapter.

Sign Bridge Configurations

The configuration of a sign bridge defines the basic layout of the structure, i.e., the overall number of towers and spans. Currently, five configurations are available in the program, as summarized in Table 2.1. The beams in a structure can be placed at different elevations and can have different depths and widths. The towers in a structure can have different top and bottom elevations but towers with two posts must all have the same width. The joint and member numbering schemes are explained in Chapter 7 - Methodology.

Tower Types

Two types of towers are allowed, single post towers or double posts. The tower posts can be either prismatic or linearly tapered. The double posts can be trussed or nontrussed. A restriction of the program is that all towers within a structure must have the same number of posts. Table 2.2 contains an illustration of the two tower types.

Beam Types

Eight basic beam types are allowed, as shown in Tables 2.3.1 and 2.3.2. The cantilever and span beams are similar except for their joint and member numbering schemes, which are explained in Chapter 7. The beam chords can be either prismatic or linearly tapered. A restriction is that all beams in a structure must have the same number of chords.

Sign Bridge Types

The four beam types and two tower types can be combined in six different ways. These define the sign bridge type, as summarized in Table 2.4.1. The six types can be used in any of the five sign bridge configurations, described previously, but only one bridge type can be used in a configuration at any one time.

Member Types

The program includes four different member types, which define the function of the member within the structure. The four member types, classified as either primary or secondary members, are listed in Table 2.5.

Section Types

Each member must be assigned a cross section type, which defines its physical properties. In all, eight section types are recognized, four tubular shapes (Table 2.6) and four general shapes (Tables 2.7.1 and 2.7.2). The tubular shapes are used as primary and secondary structural members. The general shapes are used only as vertical attachment members (VAMs).

Sign Types

Presently, only one sign type is considered - rectangular signs with constant thickness.

Walkway Types

Only one walkway type is considered - rectangular walkways mounted parallel to the bridge beams.

VAM Types

Only one vertical attachment member type is recognized - prismatic general shapes (see section types above).

A summary of the above components is presented in Table 2.8. To aid in understanding, the components in that table are grouped into three groups, I, II, and III. These groups represent different levels of detail, with increasing detail from level I to level III.

2.3 OVERVIEW

The program consists of an integrated environment for the analysis and design of sign bridge support structures. Included in the environment are a preprocessor module for data and file management, an analysis module, and a postprocessor module for screen graphics and output review. The organization of these modules and their relation to each other are described in more detail in Chapter 6 - Description of System.

2.4   SPECIAL FEATURES

WIN-SABRE contains several special features developed to make the program as easy as possible to use and to shorten the design/analysis cycle. These features are outlined below.

Windows Pull-Down Menu System

A Windows pull-down menu system is provided as a means of navigating about the program. SABRE operates in a manner whereby the user may enter, edit, or review data in any order, generally, or make use of the many tools available, at any time. See Chapter 6 for more information.

Data Entry and Editing

All data required by SIGN BRIDGE is entered by the user into input cells, generally via the computer keyboard. The user may move to any cell on a screen by using the mouse or enter key and may move to any screen by using the pull-down menu system. To minimize input errors, only data valid for that particular cell is accepted. For example, alpha characters cannot be entered into a cell expecting numerical data and a minus sign cannot be entered into a cell expecting a positive value.

Automatic Joint and Member Renumbering

If a joint or member is deleted or inserted, the remaining joints and members are automatically renumbered to maintain their positions relative to each other. This eliminates the need to recalculate and enter the other joints and/or members manually after deleting or inserting. Chapter 6 provides more information on this subject.

Section Lookup

Before analysis can occur, the section properties for each member must be defined. These properties are calculated from the member diameters, wall thicknesses, etc., which can be entered manually by the user or imported from a shape file. A shape file is a user created text file containing a table of dimensions for a particular structural shape. An example would be a file containing dimensions of the various AISC wide flange shapes. The user may call this file up onto the screen, review the data within it, and select a suitable section. The data for that section then is copied into the current data cell. See Chapter 6 for additional discussion.

On-Screen Help

The program provides extensive on-screen help during operation. This may be in the form of a brief description of the data expected, allowable values, or other pertinent information. Pressing "Help" displays a window on the screen with information. Help is available at all times except during analysis.

In addition to the special features just described, WIN-SABRE has other capabilities which make the program easier and faster to use. These are outlined in the following sections.

2.5   AUTOMATIC MESH GENERATION

Because the calculation and entry of the structural joints and members can be time consuming, automatic generation of these data is provided. When used, the mesh generator can create all data required to define the joints and members of a sign bridge. Currently, the five different basic structural configurations can be generated with each configuration built of one of the six sign bridge types. See Chapters 6 and 7 for additional information.

2.6   ON-SCREEN GRAPHICS

An image of a sign bridge structure can be viewed in Windows environment. This provides a convenient means of checking the data used to define a particular structure. The graphic image consists of a wire frame representation of the structure and can be of the current structure being edited or a structure previously analyzed. For a previously analyzed structure, the deflected shape can be viewed for any load combination (dead, dead and wind, etc.) and members found to be overstressed are highlighted. Zooming and image rotation capabilities are also provided.

2.7   STRUCTURAL ANALYSIS CAPABILITIES

SABRE can analyze support structures for a wide variety of configurations, boundary conditions, member types, and loading conditions. See Table 2.9 - Definition of Program Limits for more information. The analysis capabilities are further described below.

Configurations

The five basic support configurations are allowed with the six beam-tower types, for a total of thirty possible types. These thirty sign bridge types comprise the most common structures currently used in Maryland.

Coordinate System

Each joint is defined by X, Y and Z coordinates. This coordinate system is right-handed, as shown in Figure 2.1, and is termed the structural (or global) coordinate system.

Boundary Conditions

Boundary conditions consist of two distinct quantities, reactions and member releases. Reactions are those joints for which no movement is allowed. These may be specified such that no movement occurs for X, Y or Z translations and/or X-X, Y-Y, or Z-Z rotations. Member releases occur in members which are not continuous with respect to one or more components of shear, moment, axial load or torque. An example would be a

member which contains a hinge. The hinge is a moment release at either end of the member. It is necessitated by various construction details such as u-bolts, where no moment can be transmitted from a horizontal member to the vertical tower members. Table 2.13 gives the Definition of Member Types for Releases.

Prismatic or Tapered Members

Structural members can be either prismatic or tapered. To accommodate tapering, members can be grouped into a unit which can be assigned different cross sections at the two ends. For example, the members comprising a tapered tower can be grouped into a tower unit and the cross section properties defined at only the top and bottom of the tower. The dimensions and section properties of each member of that unit are then interpolated by the program.

Loadings

The program allows for both automatic load generation and for manual load input. The automatic load generation follows the AASHTO code for all dead, ice, and wind load calculations. The methods used in calculating these loads are given in detail in Chapter 7. Maximum flexibility with respect to loadings is allowed with the various options described as follows:

1. The dead load may be altered with a detail factor to account for connections, stiffeners, etc.
2. Ice loads may be adjusted by overriding the standard 3 psf that AASHTO specifies with another value. The ice load may be placed on either one or both sides of the sign.
3. Wind loads may be adjusted by altering the wind velocity or gust factor.

Finally, the program allows for completely general loadings via a manual entry. Here, the loads are input as joint loads.

Sign Bridge Details

The sign bridge details are composed of components such as signs, vertical attachment members (used to attach the sign to the structure), and walkway units. These can be placed at any location on the structure and the effects of their dead weights, wind loads and ice loads are then included automatically during analysis.

Stress Analysis

The stress analysis performed by the program follows the AASHTO Specifications, Reference 1. The stress investigation itself involves the determination of stresses and their interaction (i.e., the Combined Stress Ratio) for all members throughout the structure. A detailed description of the method can be found in Sect. 7.

Structural Analysis

The method used in the analysis of the sign support structure is the stiffness matrix method with a three-dimensional frame formulation. This method, subject to the assumptions listed in Table 2.10, yields joint displacements, member end moments, torques, shears, axial forces, and reactions. The limits of the number of members and joints allowed in the analysis are given in Table 2.9. A detailed description of the method can be found in Reference 2.

The results of the analysis, printed in an ASCII text file, form a complete report of the analyzed structure, including input verification, section properties, joint loads, deflections, stresses, etc. The user is then free to review the results graphically, import the reactions for design purposes, and perform other postprocessing operations. See Chapter 4 - Output of Results for more information on this subject.

Limitations

Currently, the program does not consider the following special conditions: Vibration effects, prestrained members, induced displacements, elastic connections or supports, thermal stresses, or P-Δ effects. Of special concern is the effect of vibration on the structure. According to the AASHTO code, the member L/r restrictions should prevent vibration failure and so only a static analysis is required. To be complete, however, a rigorous dynamic analysis would be desirable. Also of special concern, is the effect of elastic connections. As explained later, this program assumes all member connections to be either completely fixed or pinned, a simplification which does not model real conditions accurately. In an actual support structure such elements as u-bolts, although modeled as pins, have, in fact, some degree of fixity. This can be accounted for with the use of linearly elastic connections, where the degree of fixity can be set by the user. It is suggested that this too be added to the program.

The program also does not perform the following AASHTO checks: Allowable deflections, camber, and minimum material thicknesses. It is the responsibility of the user to check these for compliance with the requirements.

2.8 REVIEW OF ANALYSIS RESULTS

The output file created during analysis can be viewed on-screen or sent to a printer. With on-screen viewing the user can page through the output file or jump directly to selected tables. For printing purposes, either the entire file can be printed or selected tables.

2.9 POST PROCESSING CAPABILITIES

There are four special features of sign bridges within the program, base plate design, splice plate design and base plate and other details' fatigue check. A base plate is the plate connecting the bottom of a sign bridge tower to the concrete foundation. Given the proper data the program will calculate the shape and size of the base plate, number and dimensions of the

anchor bolts, and size of the tower-to-plate weld. A splice plate is a plate connecting two abutting members of a beam chord. Typically, a sign bridge is constructed using prefabricated tubular beam elements of constant length. When a beam exceeds that length then two elements must be spliced together. Given the proper data, the module will calculate the shape and size of the splice plate, number and dimensions of the connecting bolts, and size of the tube-to-plate weld.

The data required for either a base or splice plate (e.g. yield stresses, forces and moments, etc.) can be entered by hand or imported from an output file. The data required for base plate fatigue check (base plate moment range and column information) should be imported from a fatigue file. The results can be viewed on-screen or be sent to a printer. See Chapter 7 for more information on the program's post processing capabilities. The fatigue detail check in Excel is an independent calculation template for several most popular details.

Together, the features and options described above provide a comprehensive system for sign bridge analysis. Table 2.12 contains a summary of these features. With the development of SABRE, the capability now exists to design and analyze structures with a great many configurations, types, dimensions, etc., all in a reasonable amount of time.

TABLE 2.1 – SIGN BRIDGE CONFIGURATIONS		
CONFIG. NUMBER	DESCRIPTION	BASIC JOINT AND MEMBER NUMBERING SEQUENCE
1	Cantilever	
2	Single Span	
3	Butterfly	
4	Single Span with Cantilever	
5	Double Span	
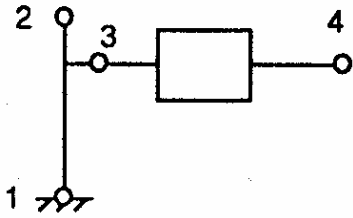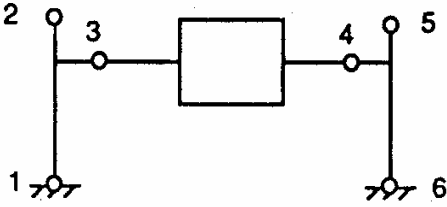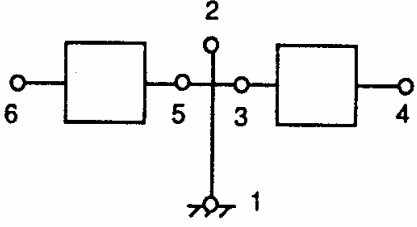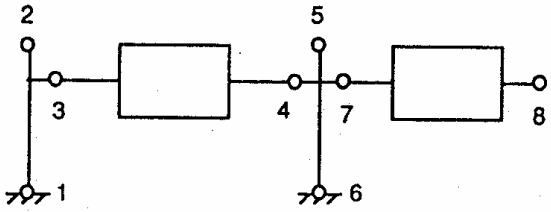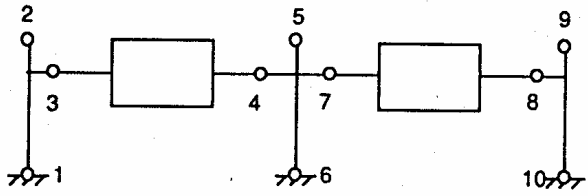
TABLE 2.2 – SIGN BRIDGE TOWER TYPES			
TYPE ID	DESCRIPTION	BASIC JOINT NUMBERING SEQUENCE	BASIC MAIN MEMBER NUMBERING SEQUENCE
1	Single point		
2	Double Post		
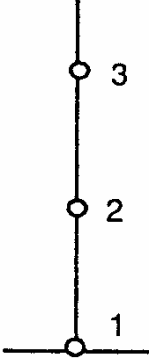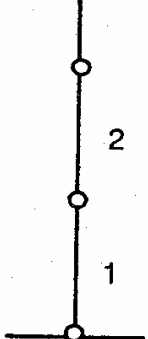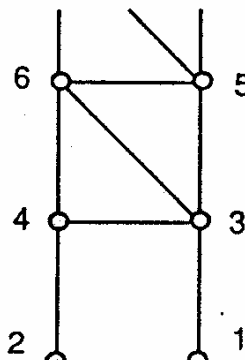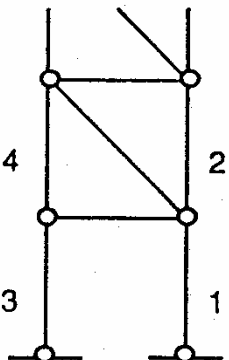
TABLE 2.3.1 – SIGN BRIDGE BEAM TYPES			
TYPE ID	DESCRIPTION	BASIC JOINT NUMBERING SEQUENCE	BASIC MAIN MEMBER NUMBERING SEQUENCE
1	Monotube, Cantilever	1 2 3	1 2
2	Plane Truss, Cantilever	2 4 6 / 1 3 5	3 4 / 1 2
3	Trichord Truss, Cantilever	2 5 8 / 3 6 9 / 1 4 7	3 4 / 5 6 / 1 2
4	Box Truss, Cantilever	3 7 11 / 2 6 10 / 4 8 12 / 1 5 9	5 6 / 3 4 / 7 8 / 1 2
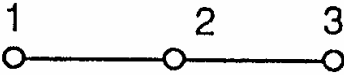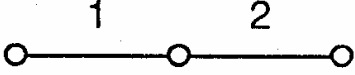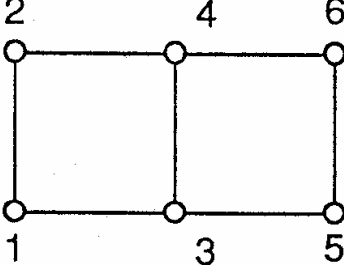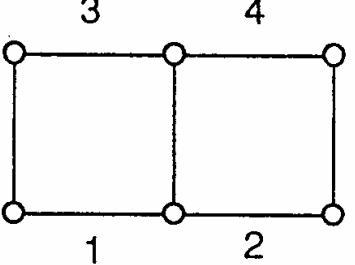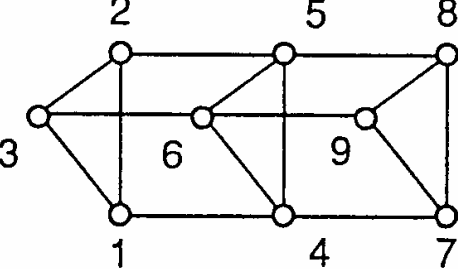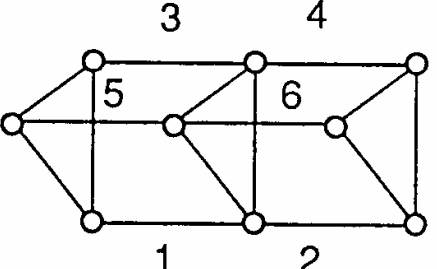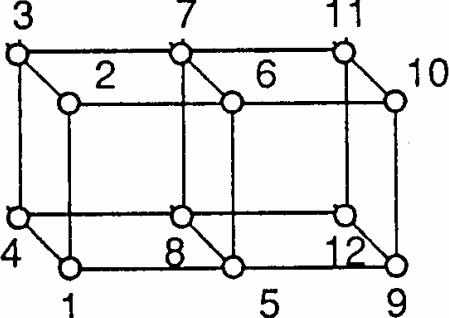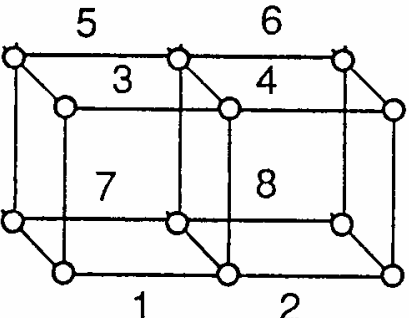
		TABLE 2.3.2 – SIGN BRIDGE BEAM TYPES	
TYPE ID	DESCRIPTION	BASIC JOINT NUMBERING SEQUENCE	BASIC MAIN MEMBER NUMBERING SEQUENCE
1	Monotube, Span		
2	Plane Truss, Span		
3	Trichord Truss, Span		
4	Box Truss, Span		
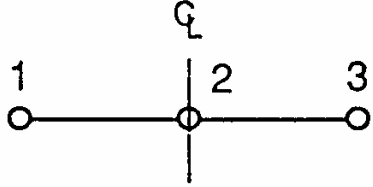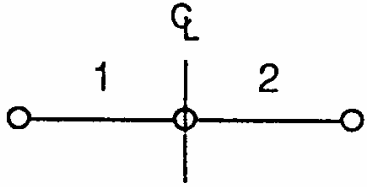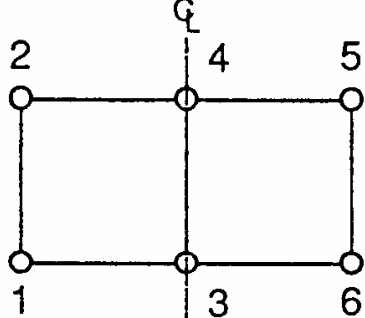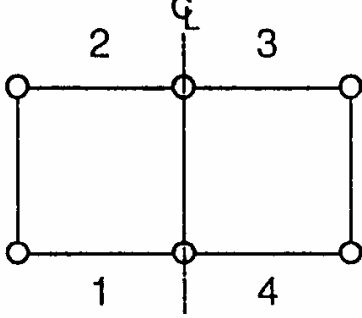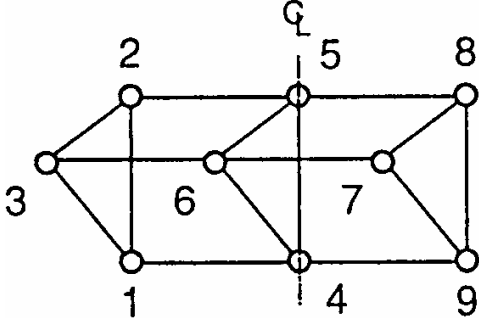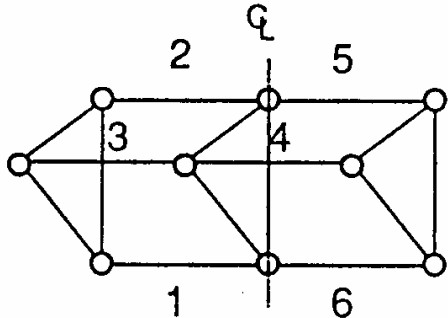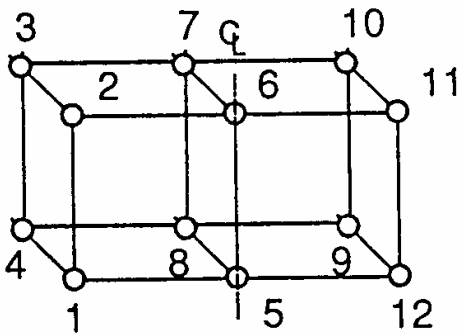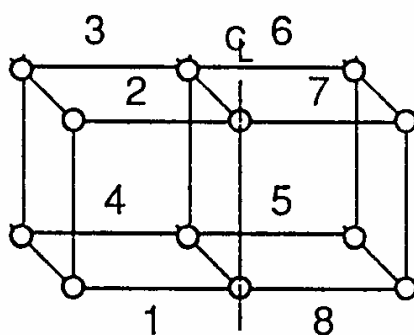
TABLE 2.4.1 – SIGN BRIDGE TYPES				
TYPE ID	DESCRIPTION	POSTS PER TOWER	CHORDS PER BEAM	REFERENCE
1	Monotube on Single Post	1	1	Table 2.4.2
2	Plane Truss on Single Post	1	2	Table 2.4.3
3	Trichord on Single Post	1	3	Table 2.4.4
4	Trichord on Double Posts	2	3	Table 2.4.5
5	Box Truss on Single Post	1	4	Table 2.4.6
6	Box Truss on Double Posts	2	4	Table 2.4.7

TABLE 2.4.2 – ACTUAL STRUCTURE TYPE VS. MODEL		
TYPE ID	SIGN BRIDGE TYPE	MODEL
1		Connection Member

TABLE 2.4.3 – ACTUAL STRUCTURE TYPE VS. MODEL		
TYPE ID	SIGN BRIDGE TYPE	MODEL
2		Connection Members

TABLE 2.4.4 – ACTUAL STRUCTURE TYPE VS. MODEL		
TYPE ID	SIGN BRIDGE TYPE	MODEL
3		Connection Members
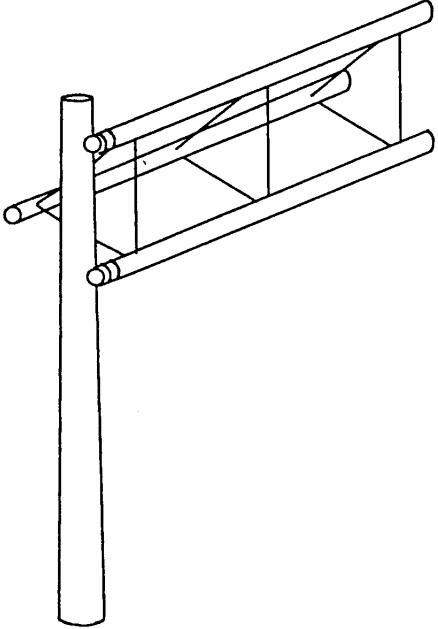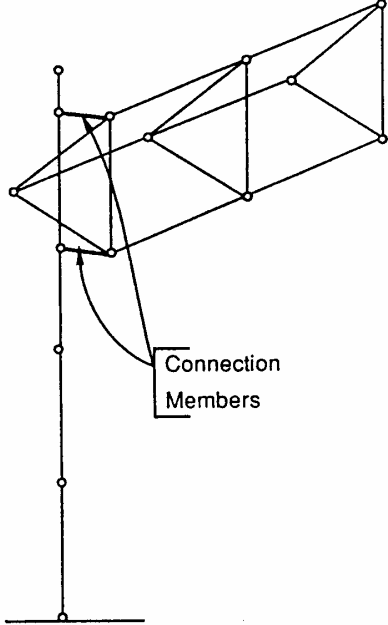
TABLE 2.4.5 – ACTUAL STRUCTURE TYPE VS. MODEL		
TYPE ID	SIGN BRIDGE TYPE	MODEL
4		Connection Members
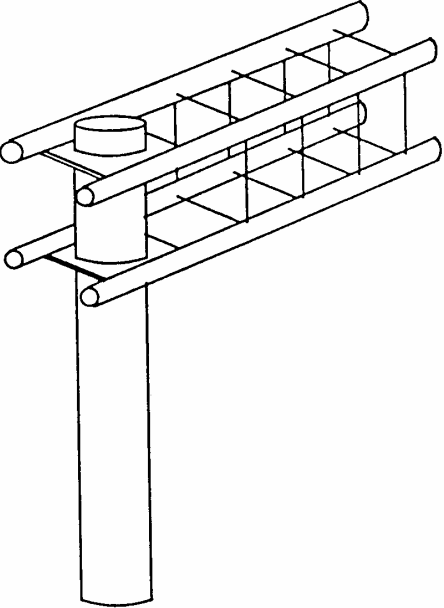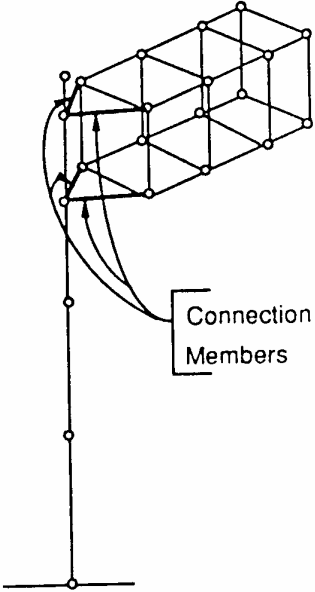
TABLE 2.4.6 – ACTUAL STRUCTURE TYPE VS. MODEL		
TYPE ID	SIGN BRIDGE TYPE	MODEL
5		
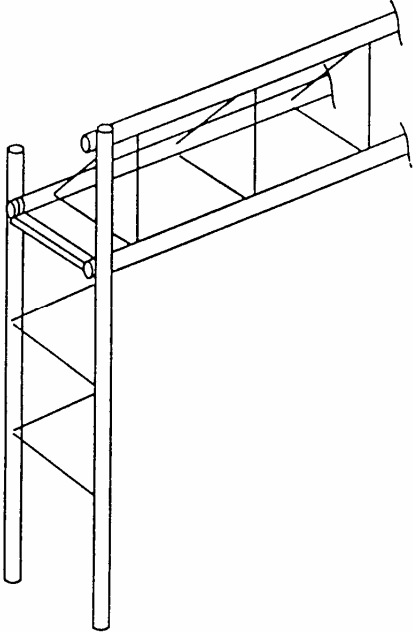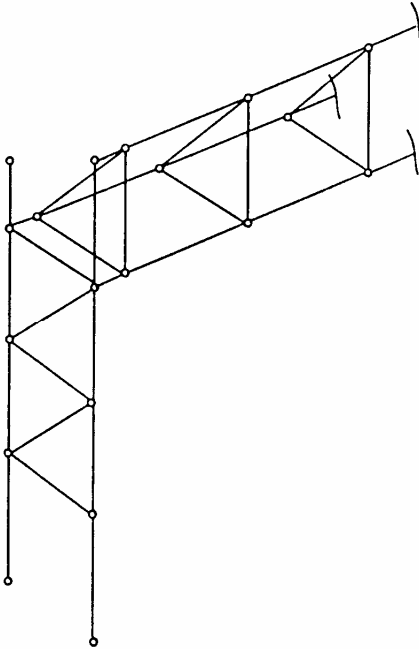
TABLE 2.4.7 – ACTUAL STRUCTURE TYPE VS. MODEL		
TYPE ID	SIGN BRIDGE TYPE	MODEL
6		
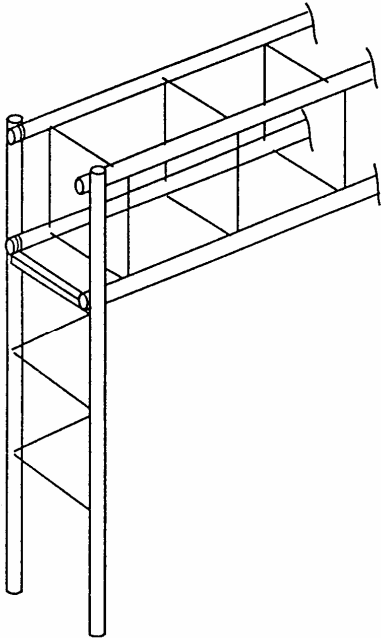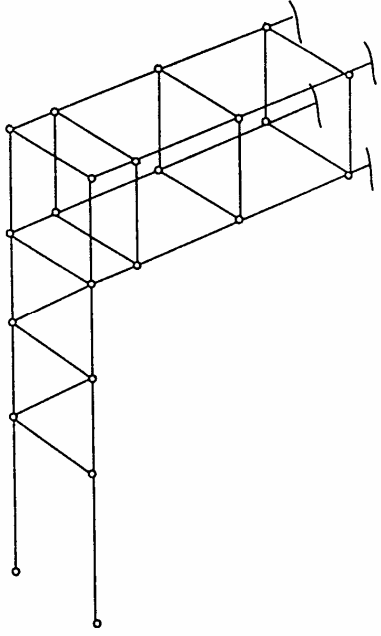
TABLE 2.5 – MEMBER TYPE AND CATEGORY WITHIN UNIT			
MEMBER		STRUCTURAL COMPONENT	
TYPE	CATEGORY	DESCRIPTION	FIGURE
1	Secondary	Interior truss members for plane, trichord and box trusses	
2	Primary	Exterior truss members for plane, trichord and box trusses	
3	Primary	Vertical (tower) members	
4	Secondary	Interior truss members for tower	
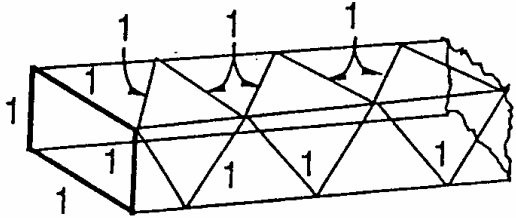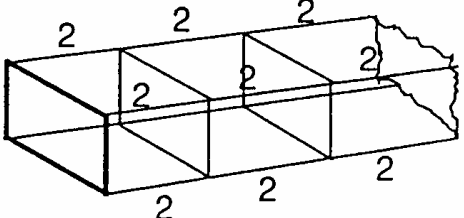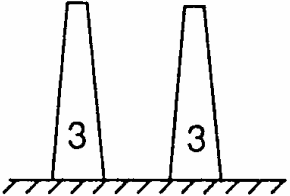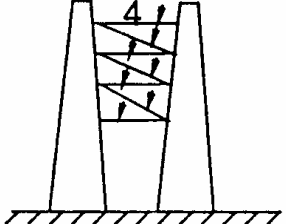
		TABLE 2.6 – TUBULAR SHAPES		
SHAPE ID	TYPE	FIGURE	STRESS CONCENTRATION FACTOR	COMMENTS
1		Not Used		
2	Round		Not Required	Note the definition of the outer diameter D.
3	Dodecagonal		Figure B-1 AASHTO Spec.	Note the definition of the outer diameter D.
4	Octagonal		Figure B-1 AASHTO Spec.	Note the definition of the outer diameter D.
5	Square		Figure B-1 AASHTO Spec.	Note the definition of the outer diameter D.
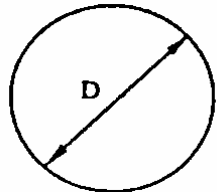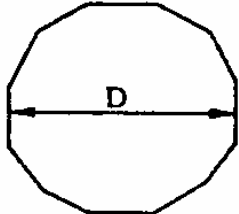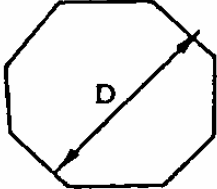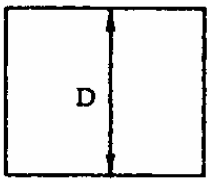
| TABLE 2.7.1 – GENERAL SECTIONS |

TABLE 2.7.1 – GENERAL SECTIONS
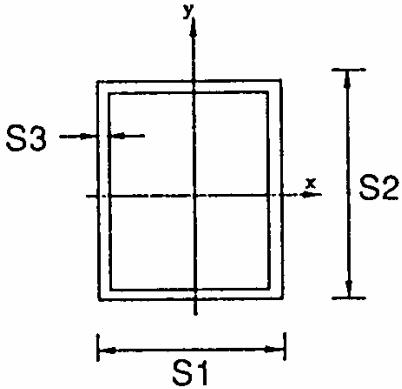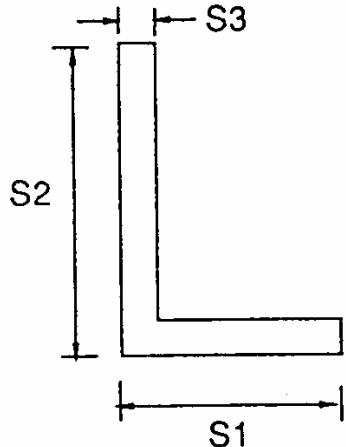
ID	TYPE	FIGURE	GENERAL PARAMETERS		
			PARAMETER	UNITS	DESCRIPTION
6	Rectangular Shapes		S_1	in (mm)	Width of section outside to outside measured parallel to x-x axis.
			S_2	in (mm)	Width of section outside to outside measured parallel to y-y axis.
			S_3	in (mm)	Thickness of section (assumed to be constant throughout section).
			S_4	none	Stress concentration factor. If left blank, program assumes value of 1.0.
			S_5		Not Used
			S_6		Not Used
7	Angle Shapes		S_1	in (mm)	Width of bottom leg
			S_2	in (mm)	Width of top leg
			S_3	in (mm)	Thickness of section (assumed constant)
			S_4	none	Stress concentration factor. If left blank program assumes value of 1.0.
			S_5		Not Used
			S_6		Not Used

2-21

TABLE 2.7.1 – GENERAL SECTIONS (cont.)					
ID	TYPE	FIGURE	GENERAL PARAMETERS		
			PARAMETER	UNITS	DESCRIPTION
6	Rectangular Shapes Alternate 1		S_1	in (mm)	Width of section outside to outside measured parallel to x-x axis
			S_2	in (mm)	Width of section outside to outside measured parallel to y-y axis
			S_3	in (mm)	Thickness of section (assumed to be constant throughout section)
			S_4	in (mm)	Cover plate with parallel to x-x axis
			S_5	in (mm)	Cover plate thickness
			S_6	in (mm)	To distinguish Alt. 1 from Alt. 2, the field for "thickness of round tube" is used to input cover plate thickness (T = S_5)
6	Rectangular Shapes Alternate 2		S_1	in (mm)	Width of section outside to outside measured parallel to x-x axis
			S_2	in (mm)	Width of section outside to outside measured parallel to y-y axis
			S_3	in (mm)	Thickness of section (assumed to be constant throughout section)
			S_4	in (mm)	Wing section width parallel to x-x axis
			S_5	in (mm)	Wing section height from outside of main section to outside of wing section
			S_6		Not Used

TABLE 2.7.2 – GENERAL SECTIONS					
ID	TYPE	FIGURE	GENERAL PARAMETERS		
			PARAMETER	UNITS	DESCRIPTION
8		Not Used			
9	Wide Flange Shapes		S_1	in (mm)	Flange width
			S_2	in (mm)	Depth of section
			S_3	in (mm)	Flange thickness
			S_4		Not Used
			S_5	in (mm)	Web thickness
			S_6		Not Used
10	"Z" Shapes		S_1	in (mm)	Width of both legs (assumed symmetric)
			S_2	in (mm)	Height of section
			S_3	in (mm)	Thickness of section (assumed constant)
			S_4	in (mm)	Stress concentration factor. If left blank, program assumes value of 1.0.
			S_5	none	Not Used
			S_6		Not Used

		TABLE 2.8 – SUMMARY OF SIGN BRIDGE COMPONENTS		
LEVEL	TERM	DESCRIPTION	NO.	REFS.
I	Sign Bridge Configurations	The basic number of beams and towers in the structure and their orientation	5	Table 2.1
II	Tower Types	The number of posts in each tower	2	Table 2.2
	Beam Type	The number of chords in each beam.	8	Table 2.3
	Sign Bridge	The combinations of beams and towers and their connections	6	Table 2.4.1
III	Member Types	The category of each member and its function within the structure	4	Table 2.5
	Section Types	The allowable member cross sections	8	Tables 2.6, 2.7
	Sign Types	The signs supported by the structure	1	—
	Walkway Types	The maintenance walkways (catwalks) supported by the structure	1	—
	VAM Types	The members connecting the signs to the structure	1	—

ITEM NO.	DESCRIPTION	MAXIMUM	
		NORMAL	LARGE
1	Number of Joints	420	4000
2	Number of Sections	40	40
3	Number of Members	999	4000
4	Number of VAMs	25	25
5	Number Supporting Members/VAM	3	3
6	Number of Walkways	5	5
7	Number of Load Numbers	20	60
8	Number of Signs	10	10
9	Number of VAMs per Sign	7	7
10	Number of Units per Structure	40	40
11	Number of Posts per Tower	2	2
12	Number of Chords per Beam	4	4
13	Number of Segments per Post	16	16
14	Number of Segments per Chord - Cantilever beam - Span beam	 16 36	 98 98

TABLE 2.9 – DEFINITION OF PROGRAM LIMITS

TABLE 2.10 – ANALYSIS ASSUMPTIONS	
ITEM NO.	DESCRIPTION
1	Linearly elastic material
2	Small deformations
3	Shear deformations neglected
4	6 kinematic degrees of freedom at each joint – 1 axial, 2 shears, 2 moments, and 1 torsion
5	Tapered members are modeled as a combination of stepped prismatic members
6	All steel, all aluminum, or steel post/aluminum chord construction (Window V3.2 based on 4th edition only)

TABLE 2.11 – SUMMARY OF SIGN BRIDGE FEATURES AND OPTIONS

SYSTEM FEATURES
- pull-down menus
- extensive on-screen help at all levels
- automatic generation of joints and member
- on-screen lookup table of AISC standard rolled sections
- on-screen lookup table of user defined tubular sections
- user selected out levels
- default value overriding capability
- extensive output generation in tabular form
- on-screen output file viewing
- indexed output tables in 8 ½" × 11" format
- design tools
- multi-level error checking and diagnostics
- in-core analysis
- complete documentation

SPECIFICATIONS
- Latest AASHTO Standard Specs. for Structural Supports for Highway Signs, Luminaires and Traffic Signals
- AASHTO load application
- AASHTO allow stress calculation
- AASHTO code checking

STRUCTURAL MODEL
- 3-D space frame model
- 6 degrees of freedom
- arbitrary boundary conditions
- arbitrary hinge placement
- general direct stiffness method w/ optimized memory
- 5 basic structural configurations
- 6 tower and beam combinations
- prismatic or tapered members
- All steel, all aluminum, or steel post/aluminum chord construction (Window V3.2 and up based on 4th edition only)
- tubular frame members

TABLE 2.12 – SUMMARY OF SIGN BRIDGE FEATURES AND OPTIONS

LOADINGS
- AASHTO or user defined wind application
- automatic ice load calculation
- wind shielding taken into account
- automatic dead load calculation
- optimal user defined joint loads
- maximum load combinations effects used

ANALYSIS
- analysis of all frame members
- automatic member property calculation
- calculation of joint displacements and reactions
- calculating of member end moments, torques, shears, and axial forces
- stress calculation of each load condition
- calculation of stress interactions

CODE CHECK
- AASHTO requirements for tubular members
- combined stress ratios
- equation number references

POST PROCESSING
- base plate design
- splice plate design
- optional import of design forces and moments
- base plate fatigue check
- fatigue detail check in Excel

GRAPHICS
- on-screen graphics
- input review
- output review
- real time image rotation
- zooming
- substructure isolation
- member detail review deflected shape viewing
- highlighting of overstressed members

TABLE 2.13 – DEFINITION OF MEMBER TYPES FOR RELEASE												
	NEAR (JJ) END RELEASES						FAR (JK) END RELEASES					
MEMBER	MOMENT			FORCE			MOMENT			FORCE		
TYPE	XX	YY	ZZ	X	Y	Z	XX	YY	ZZ	X	Y	Z
0 (or blank)												
1	X											
2		X										
3			X									
4	X	X					X					
5	X		X				X					
6		X	X									
7	X	X	X				X					
8							X					
9								X				
10									X			
11								X	X			
12		X						X				
13			X						X			
14	X	X	X				X	X	X			
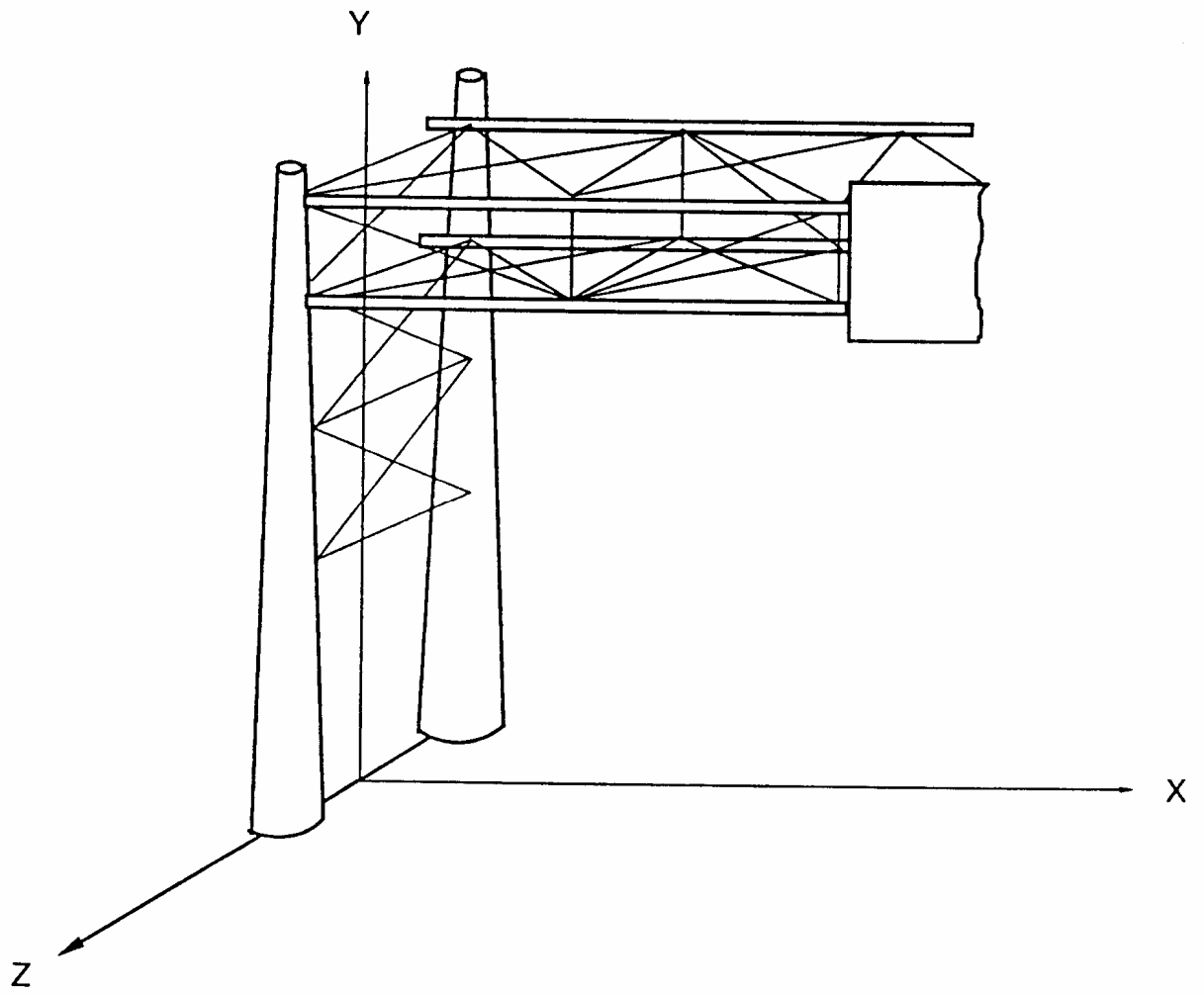
Figure 2.1 Structural Coordinate System

3 - INPUT OF DATA

3.1   GENERAL

In order for a sign bridge to be analyzed, certain data are required. These data define the joint locations, element connectivity, member properties, and other structure information required for analysis. It is important that the entry and editing of this data are made as easy as possible so that maximum time can be spent on the engineering aspect of the problem.

3.2   DATA INPUT

Data entry is accomplished with input screens. Instead of being entered into a formatted text file, data is entered into cells which appear on the screen. Each screen has a number of input cells which can be edited individually. Table 3.1 summarizes the program input screens and the card numbers used with each screen. Also, Appendix A contains copies of all SABRE input screens and Appendix B contains descriptions of each input cell.

TABLE 3.1 – LIST OF SIGN BRIDGE INPUT SCREENS	
SCREEN TITLE	DATA TYPE NUMBER USED WITH SCREEN
Project Data	01012, 01022
General Program Options	01032
Sign Bridge Configuration	21000
Element Definition	31000, 32000
Dimensions	34000, 35000, 36000, 37000, 41000
Cross Sections	33000
Bracing	33000, 38000
Yield Stresses	39000
Definition of Sections	04012
Definition of Joints	03012
Definition of Members	05012
Member Option	06012
Vertical Attachment Members	07012
Sign Data	09012
Gen. Program Design Parameters	01042
Joint Load Data	10012
Walkways	08012
Connections	01052
Height Coefficients (C_h)	09112
Hinges	40000
Base Plate Design Parameters	50000
Splice Plate Design Parameters	51000
Base Plate Fatigue Check Parameters	52000

4 - OUTPUT OF RESULTS

Results of a structural analysis by WIN-SABRE are printed in tabular form in an ASCII text file. The name of the output file and its location are defined by the user prior to analysis. Currently, the user has two possible levels of output from which to choose, level 1 (short), or level 2 (long). Level 1 generates the minimum amount of data, providing a quick review of the results. Level 2 generates additional data, providing more detailed results. See Table 4.1 for a description of the two levels.

The data within an output file are presented in various tables which are organized into three basic groups, input verification, analysis, and code check. The input verification tables provide a means for the user to check the validity of the data entered. The analysis tables summarize the structural analysis results, such as deflections, end actions, etc. The code check tables summarize the results of the AASHTO code checks such as allowable stresses and CSR values. The results are printed for each member, load combination, etc.

The data from any output file are available for base and splice plate design, graphing, and printing. Appendix C contains a list of each table generated during analysis.

OUTPUT LEVELS	DESCRIPTION	OUTPUT GIVEN
TABLE 4.1 – DEFINITION OF OUTPUT LEVELS		
1	Basic level for engineering design or analysis	All input data, maximum joint displacements, allowable stresses, and combined stress ratios
2	Basic level for engineering design or analysis with added detail above level 1	The same as Output Level 1 plus joint loads, member end actions, and joint deformations for all group loads

5 - DIAGNOSTICS

To minimize the occurrence of errors, certain checks are made within SABRE, at five levels of program operations.

The first level of error checking occurs during data input and has been described previously in Special Features. During data entry, information is screened immediately upon entry and any invalid data will either generate an error message or will not be accepted. As an example, the cell containing the number of desired beam chords will accept only four different values, the integers 1 through 4, corresponding to a one-chord beam (monotube), two-chord beam (plane truss), etc. If the user tries to enter any value outside of that range, it will not be accepted by the program; i.e., the number pressed will not be echoed to the screen.

A second level of error checking occurs as a prelude to mesh generation or analysis. Before mesh generation or analysis begins, all data is reviewed for its validity. If any invalid data is detected, then an error message is generated. For instance, in order for a valid mesh to be created, the elevations of the top and bottom of each tower must be entered. If, after checking, the top elevation of a tower is found to be lower than the bottom (a physical impossibility), then two actions will be taken. First, an error message will be printed in a screen window notifying the user of the specific problem and, second, control will be passed to the input screen where the tower elevations are entered. The improper data must be entered correctly before mesh generation can be reattempted. An error of this type cannot be ignored or overridden.

The third level consists of normally fatal errors found during analysis. These are errors undetected by the previous levels of checking but which result in unacceptable conditions. An example of such an error is the definition of a wide flange shape as a main tower member. Only tubular shapes are allowed for main members so this condition is checked and flagged. Errors detected at this level will result in termination of the analysis process.

A fourth level of error checking, warning messages, is less restrictive than the others. Warning messages notify the user of potential problems but also let the user ignore the implications. For example, a sign is connected to its support structure with one or more vertical attachment members (VAMs). If the user fails to define the VAMs properly for a sign, then that sign will not be connected to the structure and, as a result, its dead load, wind load, etc. will not be considered during analysis. To alert the user to this condition, a warning will appear in a screen window, giving him or her the opportunity to reconsider.

The final level of error checking occurs at the language level. These are unprogrammed error checks which are embedded in the programming languages and end with typically unpredictable results. An example of such an error is the calculation of the square root of a negative number. If this condition has not been trapped in the previous levels, then the result will depend on the present program level. SABRE has been designed to avoid all errors at this level.

A list of all SABRE error and warning messages is contained in Appendix D.

6 - DESCRIPTION OF SYSTEM

6.1   USING WIN-SABRE

WIN-SABRE currently is available for use on microcomputers using the Microsoft Windows environment. This manual describes the Windows version of **WIN-SABRE**, hereafter referred to as **SABRE**. This version utilizes a Windows-based pull-down menu structure to access **SABRE**'s input, analysis, output graphics, post processing, and printing utilities.

6.2   BEFORE YOU BEGIN

SABRE is designed to run on microcomputers that use the Microsoft Windows operating system. While this manual provides step-by-step instruction in the use of **SABRE**, it cannot address the specific operation of every personal computer (PC). Before you begin, please ask yourself the following questions:

1.   Are you familiar with the PC or microcomputer you are using?
2.   Are you familiar with Microsoft Windows?
3.   Do you have an understanding of the concepts and use of terms such as menus, help screens, cursor, mouse, files, etc.?
4.   Have you read/installed the **SABRE** software using the installation instructions you received with your system disks?
5.   Have you filed your installation instructions with your other **SABRE** reference material?

If you cannot answer "Yes" to all of these questions, please take the time to address them before continuing on in this manual.

If you are prepared to continue, take a moment to look over the Table of Contents provided at the beginning of this manual. You will find that the remainder of this document illustrates the detailed use of the four basic utility functions of **SABRE** in Section 1.4.

The remainder of this section describes how to enter **SABRE** and how to access the Main Menu.

6.3 ACCESSING THE MAIN MENU

The **SABRE** MAIN MENU is the main access screen to each of the utilities provided within the **SABRE** system. It is also the main return point when you have finished using one of the utilities.

If your PC is currently off, simply turn it on and run Microsoft Windows. After entering Windows, **SABRE** can be run by double-clicking the **SABRE** icon. The **SABRE** Introduction Screen will be displayed on your monitor in a few seconds.

University of Maryland
Bridge Engineering Software and Technology Center

SABRE

Sign Bridge Analysis and Evaluation System

Windows Version 3.2

Copyright 1987-2004 BEST Ctr., UM
All rights reserved
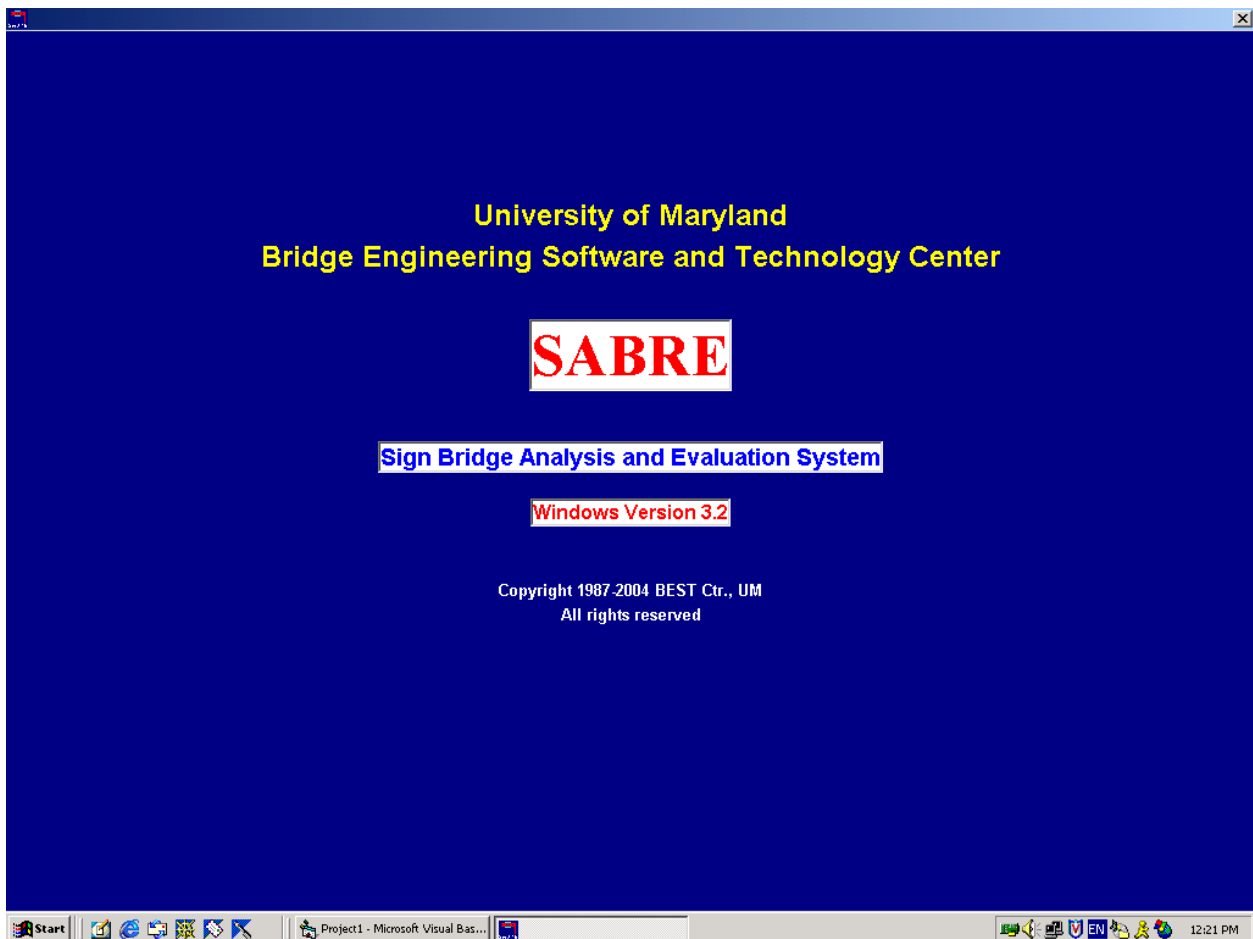
6.4   THE SABRE MAIN MENU

This screen allows you to access any of the six utilities in **SABRE** or to exit the program. These are the Input, Analysis, Output Graphics, Post Processing, Print, and Help utilities
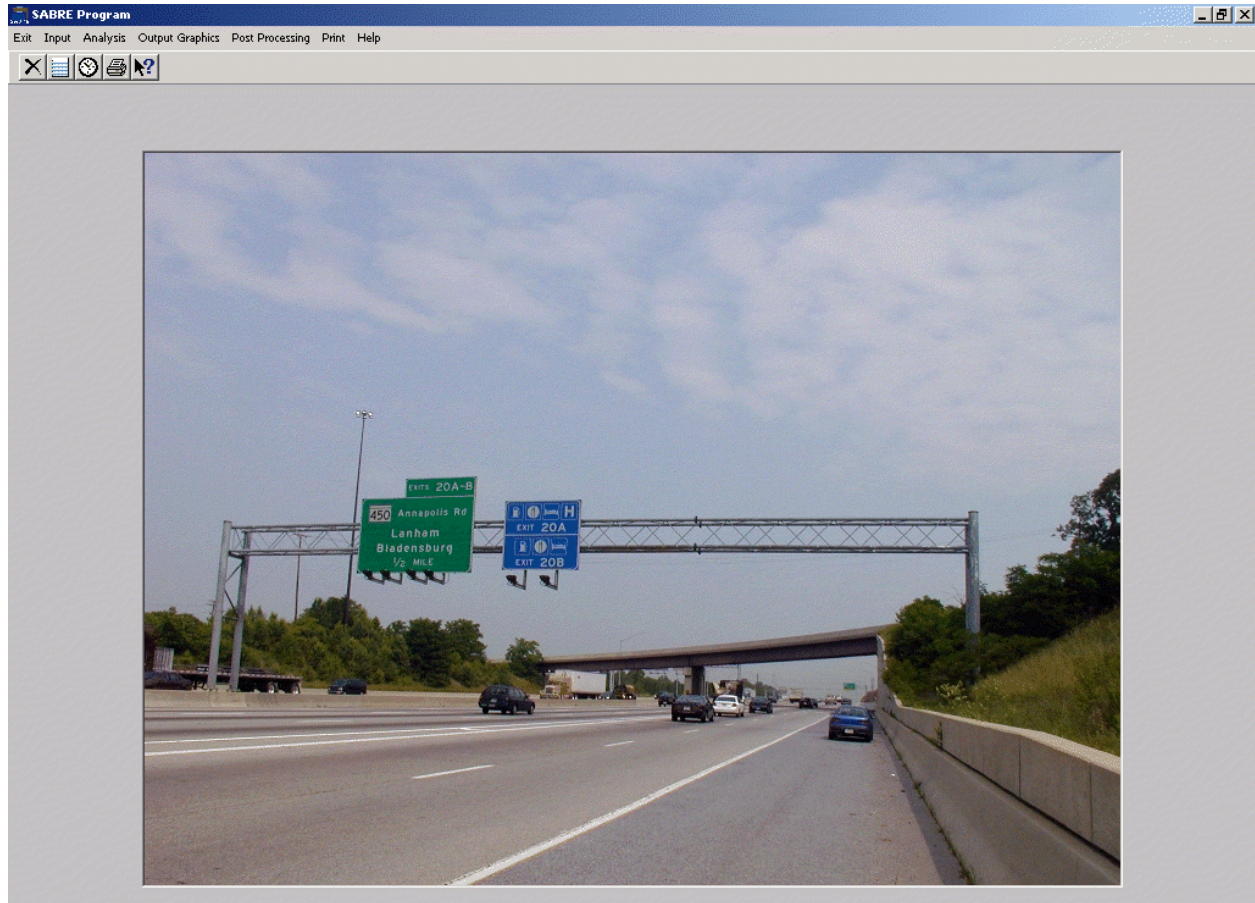
Input Utility – allows you to create new sign bridge data files or to edit existing files. Once you have entered the details of a structure, then you can save it for later use.

Analysis Utility – allows you to execute the **SABRE** program using the data stored in any of your input data files.

Output Graphic Utility – allows you to view and print the shape of a **SABRE** structure previously analyzed.

Post Processing Utility – allows you to do base plate and other details' fatigue check and design base plates and splice plates.

Print Utility – allows you to view and print output files and tables. It also provides a directory of available tables for your convenience.

Exit – allows you to exit **SABRE** simply by clicking on the word 'Exit' in the **SABRE** Main Menu or by typing Alt-x on your keyboard.

Help Utility – allows you to view help for the Help basics, commands, and buttons. Help Utility also may be accessed from Input Utility.

6.5   DETAILED USAGE OF THE UTILITY PROGRAMS

Input Utility – is accessed by clicking on the Input in the main menu. It allows you to create new sign bridge data files or to edit existing files. Once you have entered the details of a structure, you can then save it for later use.

To open a data file:

1.   On the **File** menu, click **Open**.
2.   In the Look in box, click the drive that contains the file.
3.   Below the look in box, click the folder that you want.
4.   Double_click the data file, or type it in the File Name box.

To create a new data file:

On the **File** menu, click **New**.

To save a new, unnamed data file:

1.   On the **File** menu, click **Save As**.
2.   In the File name box, type a name for the data file.
3.   Click **Save**.

To save an existing data file:

On the **File** menu, click **Save**.

Input Screens:

The available input categories are System, Structure Generation, Structure Lookup, Options. Each category has its own submenu(s) which include related bridge input data screens.

Using the keyboard with input screens:

To move in a table	Press
To the next cell in the row	ENTER or TAB or Right arrow
To the previous cell in the row	Left arrow

Up one row in a table	Up arrow
Down one row in a table	Down arrow
To move in individual fields	Press space bar
To the next field	ENTER or TAB or Right arrow or Down arrow
To the previous field	Left arrow or Up arrow

To delete, cut, copy, and paste data in a field:

To delete data, select them. Then on the Edit menu, click **Delete**.

To cut data so you can move it to another field, select the data. Then on the Edit menu, click **Cut**.

To copy data so you can paste a copy of it in another field, select the data. Then on the Edit menu, click **Copy**.

To paste data you have cut or copied, click the place where you want to put the data. Then on the Edit menu, click **Paste**.

To undo your last action, on the Edit menu, click **Undo**.

To delete, cut, copy and paste data in a row:

To delete a row of data, double click the gray area on the leftmost side of the table to highlight the row. Then on the Edit menu, click **Delete**.

To cut a row of data so you can move it to another place, double click the gray area on the leftmost side of the table to highlight the row. Then on the Edit menu, click **Cut**.

To copy a row of data so you can paste it in another place, double click the gray area on the leftmost side of the table to highlight the row. Then on the Edit menu, click **Copy**.

To paste a row of data you have cut or copied, double click the gray area on the leftmost side of the table to highlight the row where you want to put the data, Then on the Edit menu, click **Paste**.

To undo your last action, on the Edit menu, click **Undo**.

Going to the next or previous screen:

Clicking on the <up arrow> icon takes you back to the previous screen.

Clicking on the <down arrow> icon takes you down to the next screen.

Going to a specific screen:

On the **Input Screen** menu click any input screen title.

On the **Go To** menu click any input screen data type number.

Input Graphic:

Plots the joint location and member connectivity for the current data file, as inputted by the user or created by the **MESH** generator.

Automatic Joint and Member Renumbering:

The user may insert or delete an entire row of data, defined as a record. The insertion or deletion of a record may, however, affect data on another screen. For instance, the "left" and "right" ends of a member on Definition of Members represent joints defined on Definition of Joints. If a record is deleted on the Joints screen then all joints listed following that record will be shifted down the joint list. For example, if joint number 6 is deleted from the joint list then joint number 7 will become 6, 8 will become 7, and so on. All members connected to joints numbered greater than 6 will automatically be renumbered on the Member screen, thus saving the user the tedious task of renumbering by hand. Any member, however, that previously had been connected directly to joint 6, will become undefined. The user is responsible for redefining any such member. It is important to note that even when data has been created by the mesh generator, the user then can edit that data as if it had been entered by hand. This provides a considerable degree of flexibility.

Shape Files:

The input of the member properties for each element of a SABRE is accomplished on Definition of Sections. The data required on that screen consist of section types (tubes, WF, etc.) and dimensions (diameters, depths, thicknesses, etc.). The entry of this data can be accomplished in one of two ways, by the manual entry of the data or by the import of the data from a shape file. A shape file is a user-created formatted text file containing a list of dimensions for a particular type of structural shape. As an example, the user can create a shape file, using any word processor, which contains a list of the diameters, wall thicknesses, moments of inertia, etc. for round tubes. That file then can be called up from screen Definition of Sections by pressing the Section Lookup key. The data in that file will appear in a screen window which users can browse through at their convenience. If the user desires to use the data for a particular section, the user can highlight the data and click copy. Then click the record selector to highlight the row where you want to put the data and click paste. The relevant data from the selected section will be imported into the proper cells on the current line of Definition of Sections. Currently, two types of shape

files are recognized, tubes and wide flange sections. The tubes can be round, square, octagonal, or dodecagonal. The file formats are predefined and cannot be changed by the user.

Description of MESH:

MESH generates the joint and member data used in the analysis. In order to generate the mesh, certain data which defines the structure are required. These data are summarized in Table 6.3. With that data MESH is able to generate the data listed in Table 6.4, which defines the frame joints and members. After generation of all mesh data the structural frame is completely defined and can be analyzed, assuming that all other required data also has been entered.

Analysis Utility – is accessed by clicking on the Analysis in the main menu. It allows you to execute the SABRE program using the data stored in any of your input data files.

To select an input data file:

1.   Click the **Input File** button.
2.   In the Look in box, click the drive that contains the file.
3.   Below the look in box, click the folder that you want.
4.   Double_click the data file, or type it in the File Name box.

NOTE:
The default output file will appear below the output file button, after an input data file is selected. To select a different output file, click the Output File button, then follow the same procedures.

To execute SABRE:

Clicking the **OK** button on the run utility screen will execute **SABRE**. After the execution starts, a separate window will appear on the screen with the program status shown.

Print Utility

To change printers and printing options:

1.   On the **Print** menu, click **Print Setup**.
2.   To change printers, paper size, or page orientation, make the appropriate modifications.
3.   Click **OK**.

To Print screen:

On the **Print** menu, click **Print Screen**.

Print Utility is accessed by clicking on **Print** in the main menu.
It allows you to view and print output files and tables. It also provides a directory of available tables for your convenience.

To open a result file:

1.   Click the drive that contains the file.
2.   Click the directory folder that you want.
3.   Double_click the file in the file list box or type it in the file name box.
4.   Click **OK**.

To view the whole result file:

Click the **View/Print File** tab to view the whole file.

To find a string:

1.   Click **Find String**.
2.   In the **Enter Search String** box, enter the text you want to search for.
3.   Click **OK**.

To find next string:

Click **Find Next** to search the next string.

To print the whole result file:

1.   Click the **Print button**.
2.   Chose a printout option.
3.   Click **OK**.

To change printing options:

1.   Click **Print Setup**.
2.   To change printers, paper size, or page orientation, make the appropriate modifications.
3.   Click **OK**.

To view the tables:

1.   Click the **View Tables** tab.
2.   Click on a table you would like to view from the list on the top.
3.   The selected table will be displayed on the bottom.

To print the tables:

1.   Click the **Print Tables** tab.
2.   Click on the tables you would like to print from the list on the bottom.
3.   The selected tables will appear on the top list.
4.   Click **Print**.

Note: To delete a table from the top list, click on the table.

Output Graphics Utility

By clicking on the word 'Graphic,' shown in the menu bar of the Main Menu screen, you will be transferred to the **SABRE** Graphic Utility screen. The Output option, in addition to plotting the joints and members, provides several other tools. First, the deflected structural shape for the various loading conditions can be plotted, thus revealing obvious problem areas. Also, any member having been analyzed as overstressed (CSR, 1.0) will be highlighted on the screen, making quick identification possible. The user may also review the analysis results of any particular member in the structure, including dimensions, deflections, and CSR value. The user has the ability to view the image from any arbitrary viewing angle, zoom in on any part of the image, and toggle the joint and member numbering on and off.

Post Processing Utility

The POST PROCESSING module has four special features of **SABRE**s, base plates design, splice plates design and base plate and other details' fatigue check. The Base Plate Design option is used to design the plate connecting the bottom of a **SABRE** tower to the concrete foundation. The Splice Plate Deign option is used to design the plate connecting two abutting members of a beam chord. The data required for base plate design and splice plate design options can be entered by hand or imported from an output file. When imported, the forces and moments yielding the most conservative design are used. For base plate fatigue check option, the required data should be imported from the fatigue file. An analysis can be performed when all other desired data is entered. The results are displayed in a screen window and can then be sent to a printer or viewed graphically. Fatigue detail check is an independent Excel template for several most popular details. This Excel template contains macros; to run the macros, change the macro security level to a lower setting. The security setting is under Options on the Tools menu.

TABLE 6.1.1 – LIST OF SABRE.EXE MENU SELECTIONS				
MAIN CHOICES	SUBCHOICES	SUB SUBCHOICES	SUB SUB SUBCHOICES	FUNCTIONS
Exit				Exit the system
Input	File	New		Create a new input data file
		Open		Open an existing input data file
		Save		Save an existing data file
		Save As		Save a new unnamed data file
	Edit	Undo		Undo the action
		Copy		Copy the selected data
		Cut		Cut the selected data
		Paste		Paste the selected data
		Delete		Delete the selected data
	Input Screen	System	Project Data	Entry of project identification data
			Program Options	Entry of output level
		Structure Generation	Configuration	Choice of basic structural configuration
			Elements	Entry of number of posts, chords, and segments
			Dimensions	Entry of fame dimensions
			Cross Sections	Entry of main member properties
			Bracing	Entry of truss types and secondary member properties
			Yield Stresses	Entry of material yield stresses
			Sections	Entry of section properties
			VAMs	Entry of vertical attachment member data
			Signs	Entry of sign data
		Structure Lookup	Joints	Entry of joint data
			Members	Entry of member data

MAIN CHOICES	SUBCHOICES	SUB SUBCHOICES	SUB SUB SUBCHOICES	FUNCTIONS
		Options	Parameters	Override of wind, ice, dead load default data
			Joint Loads	Entry of arbitrary joint leads
			Walkways	Entry of walkway data
			Connections	Chord-post joint numbers
			Height Coefficients	Override of wind load data
			Hinges	Entry of chord-post hinge data
			Member Option	Override wind data, allowable and eff. length K
	Go To			Go to a specific Screen
	Input Graphic			On-Screen plotting of current data file
	Help	**Contents and Index**		SABRE on-line help
		About SABRE		Display SABRE information
Analysis				Analysis of current data
Output Graphics				On-screen plotting of a SIGN BRIDGE output
Post Processing	Base Plate Design			Design a tower base plate
	Splice Plate Design			Design a beam splice plate
	Base Plate Fatigue Check			Perform base plate fatigue check
	Fatigue Detail Check			Perform fatigue check of 7 types
Print	Open File			Open a sign bridge output file
	View/Print File			On screen viewing or printing a sign bridge output file
	View Tables			View selected tables from a sign bridge output file
	Print Table			Print selected table from a sign bridge output file
	Exit			Exit print utility
Help	**Contents and Index**			SABRE on-line help
	About SABRE			Display SABRE information

SCREEN TITLE	TYPE OF INPUT
	TABLE 6.2 – DATA REQUIRED FOR MESH GENERATION
Configuration	One of the five basic structural types to be analyzed (e.g., cantilever, span, etc.)
Elements	The number of beam chords, tower posts, and segments in both
Dimensions	Beam lengths, widths, depths, heights, and tower widths and heights
Cross Sections	The shapes and dimensions of each tower, beam and bracing element
Bracing	The type of truss pattern (e.g., Pratt, Warren, etc.) or each beam and tower and the shape and dimensions of the bracing members
Yield Stresses	The material yield stress for each type of beam, tower, and bracing element
Hinges	The type of member end conditions desired at the beam-to-tower connections, i.e., fixed, hinged, etc.

SCREEN TITLE	TYPE OF OUTPUT
TABLE 6.3 – DATA CREATED DURING MESH GENERATION	
Joint Data	Joint numbers, coordinates, boundary conditions, and beam-chord connection numbers
Member Data	Member numbers, section numbers, joint numbers, principle axis, unit number, unit type, material yield stress, and member type

7 - METHODOLOGY

7.1 GENERAL

The methodologies of topics specifically related to the SABRE program are discussed in this Chapter.

7.2 CALCULATION OF TORSIONAL CONCENTRATION STRESS FACTOR

As described in Chapter 6, the program can calculate the torsional stress concentration factor, K_t, for tubular members. For nonround tubular members the shear stress due to torsion is constant about the periphery of the section with higher concentrations at the corners. Torsional shear stress can be expressed as

$$T = K_t T_0 = K_t \frac{M_x}{cR^2 t} \qquad \text{(Eq. 7.1)}$$

where
   K_t = stress concentration factor,
   τ_0 = average torsional stress,
   M_x = torsional moment about the local X-X axis,
   c = shape coefficient for the particular tube,
   R = tube radius to the midline of the wall,
and t = wall thickness.

All of these parameters, other than K_t, are defined easily during input or calculated during analysis. The calculation of K_t, however, requires some effort.

The calculation of K_t is taken from Reference 4 where the stress due to torsion at the corner of a tubular member is given as

$$T = \frac{T_0 t}{r_i}\left[\frac{1 - s(r_a + r_i)/4A}{\log(r_a / r_i)} \right] + \frac{T_0 sr}{2} A \qquad \text{(Eq. 7.2)}$$

where
   τ_0 and t are as previously defined,
   r_i = inner corner radius,
   r_a = outer corner radius,
   r = distance to the point of interest
    = r_i for maximum stress,
   s = tube perimeter to the midline of the wall,
and A = tube area to the midline of the wall.

These factors are illustrated in Figure 7.1. By relating equations 7.1 and 7.2, it can be shown that

$$K_t = \frac{T}{T_0} = \frac{t}{r_i}\left[\frac{1 - s(r_a + r_i)/4A}{\log(r_a/r_i)}\right] + \frac{sr_i}{2A}$$

When the sectional dimensions of a tubular member are imported into Screen 9, the parameters D, t, and r_i are read directly from the shape file. In order to calculate K_t, then, it is necessary to first calculate r_a, s, and A by

	r_a	$= r_i + t$,
	s	$= 2 \cdot n \cdot h$,
and	A	$= n \cdot R \cdot h$,

where	n	= number of sides to the section,
	D	= outside-to-outside diameter,
	R	= (D - t) / 2,
	t	= wall thickness,
	Θ	= 360° / n,
and	h	= R sin Θ.

Knowing these values, K_t can be determined.

Examples

Several example problems have been performed for comparison with AASHTO Figure 1.3.1B(3).

In Example 1, a square tube with an outside-to-outside diameter of 50.25" (1276.4 mm), a wall thickness of .25" (6.4 mm), and an inner corner radius of .25" (6.4 mm) is selected. So

English Units

	n	= 4
	D	= 50.25"
	t	= .25"
	r_i	= .25"
	R	= (50.25 - .25) / 2 = 25"
	Θ	= 360° / 4 = 90°
	h	= 25 sin (90°) = 25"
	s	= 2 (4) (25) = 200"
	A	= 4 (25) (25) = 2500 sq.in.
and	r_a	= .25 + .25 = .5"

Therefore,

$$K_t = \frac{.25}{.25}\left[\frac{1-200(.5+.25)/4(2500)}{\log(.5/.25)}\right] + \frac{(200)(.25)}{2(2500)} = 1.4311$$

Metric Units

n	= 4
D	= 1276.4 mm
t	= 6.4 mm
r_i	= 6.4 mm
R	= (1276.4 - 6.4) / 2 = 635 mm
ı	= 360° / 4 = 90°
h	= 635 sin (90°) = 635 mm
s	= 2 (4) (635) = 5080 mm
A	= 4 (635) (635) = 1612900 mm²
r_a	= 6.4 + 6.4 = 12.8 mm

Therefore,

$$K_t = \frac{6.4}{6.4}\left[\frac{1-5080(12.8+6.4)/4(1,612,900)}{\log(.5/.25)}\right] + \frac{(5080)(6.4)}{2(1,612,900)} = 1.4311$$

From AASHTO Figure 1.3.1B(3), K_t is found to be about 1.44. The results for this and two other examples can be found in Table 7.1. As can be seen in that table, the results compare favorably. Note that the dimensions used in the examples are chosen for convenience only and do not represent any real shapes.

7.3 MESH GENERATION

Special methods for this program to streamline the mesh generation process are described in this section.

Joint Numbering

One of the primary functions of mesh generation is the creation of the joints and members in such a manner as to minimize the size of the required stiffness matrix. In programs which use the banded matrix method the size of the stiffness matrix is related directly to the total number of degrees of freedom and the semi-band width. For space frames, the size of the stiffness matrix is an N × N_B array where,

N	= total number of degrees of freedom
N_B	= semi-band width = 6 *JK_i - JJ_i + 1* $_{max}$,
6	= number of degrees of freedom per joint
JK_i	= the joint number at the "right" member end
JJ_i	= the joint number at the "left" member end

7-3

and $_i$ = 1 to the total number of members

As indicated by the $_{max}$ notation, N_B is defined by the largest difference between the end joint numbers for all members. It is evident, then, that to minimize the amount of required memory, the maximum difference between the joint numbers at either end of a member must be minimized for all members. A disadvantage of this method is that many zero terms in the stiffness matrix are included within the bandwidth and, consequently, are stored and operated on unnecessarily. SABRE, however, makes use of the "skyline" method, a more efficient storage scheme where the zero terms outside of the skyline are not used. The result is a method which is less sensitive to the joint numbering sequence and, therefore, requires less memory and processing time. Detailed discussions of the band width and skyline methods can be found in References 2 and 5, respectively. Despite the benefits in the use of the skyline method, it is still desirable to number the joints in such a manner as to minimize the joint number differences. As an example, in Figure 7.2, the maximum difference in the joint numbers of the diagonal members is equal to 6 (7-1) for Alternate A and 3 (4-1) for Alternate B. Because it is desirable to minimize these differences it is obvious that the numbering scheme used in Alternate B is preferable. Alternate B will require a smaller stiffness matrix, less memory, and less processing time. In SABRE this reasoning is expanded to cover all possible cases, which can be summarized in the three following rules:

1. The joints for a particular tower are numbered alternating from one post to the other. See Table 2.2.
2. The joints for a particular beam are numbered in a rotating fashion from one chord to the other. See Tables 2.3.1 and 2.3.2.
3. Structures which contain more than one tower or beam are numbered in a manner that minimizes the differences in joint numbers at connection joints, where beam chords are attached to tower posts. See Table 2.1.

The joints for all structural configurations and beam-tower combinations, then, are automatically numbered using the preceding rules, whichever are applicable. The result is reduced memory requirements and faster solution time.

SABRE Constructions

Because of the large number of different SABRE configurations available, a family of subroutines was developed and they are used for all configurations. These subroutines are used to define a basic structure which can be combined in various ways to form any configuration.

The basic structure consists of a right-handed cantilevered tower-beam combination, as seen in Table 7.2. The five available configurations can be constructed with all or part of this structure. For instance, a single cantilevered beam SABRE (configuration 1) can be formed from only one basic structure, as seen in Table 7.2. A single span type SABRE can be formed with two structures, one right-handed and one left-handed, as seen in the same table. Some configurations (3 and 4) make use of just the beam portion of the basic

structure as a component. Again, the advantage of this method is that only the programming of the basic structure is required, thus reducing development time and memory requirements for the program code.

SABRE Modeling

Another important consideration in mesh generation is the creation of a structural model to represent the actual structure. A real sign bridge consists of a number of three-dimensional elements (beams, plates, connections, etc.) which together act as a unit. In this program, however, all elements are assumed to behave as beam elements.

Of particular concern are the special connections which attach the beam chord to the tower post. For cantilevered structures (beams supported at only one end) connection members are treated as rigid members with relatively high moments of inertia in order to provide structural continuity and to avoid stability problems. U-bolts, hinges, etc. would create stability problems in a cantilevered beam during analysis and so are not allowed. The connections are given a special designation during generation to mark them as connection members. For span type structures (beams supported at both ends) connection members are treated as regular beam members; u-bolts, hinges, etc. may be defined at the user's discretion. Tables 2.4.2 through 2.4.7 illustrate each of the six SABRE types and the associated models.

7.4   SCREEN GRAPHICS

SABRE includes an extensive library of special graphics functions which allow for easy data review. Several of these functions are discussed in this section.

Scaling

When initially displayed, the SABRE image is plotted at a scale that ensures it fits within the screen limits. Scaling is accomplished by requiring that the ratio of the span length to tower height of the image is the same as that of the actual structure. This is done automatically within the program. The origin of the screen coordinate system is set at the center of the computer screen and the structure is plotted so that it is centered about the origin. If the structure is then rotated it will appear to rotate about a point at the screen center.

Zooming

Any arbitrary part of the structure may be magnified for closer inspection by creating a zoom window. A zoom window is created by defining the opposite corners of a rectangular area on the screen. After these corners are defined, the portion of the structure enclosed by the zoom window is rescaled in proportion to the window dimensions so that it fills the entire screen. The enlarged structure can then be rotated, zoomed again, etc. The structure can be returned to the unzoomed (original) scale at any time with the touch of a key.

Viewing of Rotated Structure

A useful feature of the program is the ability to view the plotted SABRE structure from an arbitrary viewpoint by rotating the structure about the system origin at the center of the screen. The structure may be rotated about the vertical or horizontal screen axes using the four keyboard arrow keys, left, right, up, and down. The left and right arrow keys rotate the structure about the vertical screen axis, where a rotation to the right is defined as positive. The up and down arrow keys rotate the structure about the horizontal screen axis, where a rotation up is defined as positive.

The rotations about the two axes are defined by the angles ι_y and ι_x. Each time an arrow key is pressed, the corresponding angle is redefined by the current angle increment (set by the user) and the sign (positive or negative), each joint is mathematically rotated about the associated axis, and the structure is replotted. It is important to emphasize that the angles ι_y and ι_x are not cumulative values but, rather, incremental values measured from the previous position.

The mathematical rotation of each joint is performed using a matrix transformation, whereby the coordinates of the new joint are products of the old joint coordinates and a rotation matrix. The two rotation axes have different associated matrices.

When a point in three dimensional space is rotated about the Y axis only the X and Z coordinates are affected. For example, as can be seen in Figure 7.3, the point originally at [1, Y, 0] and then rotated ι_y radians about the Y axis would have the coordinates [cos ι_y, Y, -sin ι_y]. Similarly, as seen in Figure 7.3, the point at [0, Y, 1] which is rotated ι_y about the Y axis would have the resulting coordinates [sin ι_y, Y, cos ι_y]. This transformation can be represented by the multiplication of a vector and a 3 × 3 matrix

or   P R$_y$ = P′   (Eq. 7.3)

where   P   = the vector coordinates of the original point,
             R$_y$ = a 3 x 3 rotation matrix,
             P′   = the vector coordinates of the rotated point.

The matrix R$_y$ can be found using the two previously mentioned points. In matrix notation,

$$[1, Y, 0] \, R_y = [\cos \iota_y, Y, -\sin \iota_y]$$

and

$$[0, Y, 1] \, R_y = [\sin \iota_y, Y, \cos \iota_y],$$

where

$$R_y = \begin{vmatrix} R_{11} & R_{12} & R_{13} \\ R_{21} & R_{22} & R_{23} \\ R_{31} & R_{32} & R_{33} \end{vmatrix}$$

Solving for the unknowns R_{11} ..... R_{33} yields

$$R_{11} \quad = R_{33} = \cos \gamma_y$$
$$R_{13} \quad = -\sin \theta_y$$
$$R_{31} \quad = \ \sin \theta_y$$
$$R_{22} \quad = 1$$
$$R_{21} \quad = R_{12} = R_{23} = R_{32} = 0,$$

so that
$$R_y = \begin{vmatrix} \cos\theta_y & 0 & -\sin\theta_y \\ 0 & 1 & 0 \\ \sin\theta_y & 0 & \cos\theta_y \end{vmatrix} \qquad \text{(Eq. 7.4)}$$

The coordinates of any point rotated about the Y axis can be found by equation 7.3, where R_y is given by equation 7.4.

The rotation about the X axis is calculated in a similar fashion. It can be seen in Figure 7.4 that the point originally at [X, 0, 1] and then rotated ι_x radians about the X axis would have the resulting coordinates [X, sin ι_x, cos ι_x]. Also, as seen in Figure 7.4, the point at [X, 1, 0] would have the rotated coordinates [X, cos ι_x, -sin ι_x]. Using these two points,

$$[\ X, 0, 1 \ ] \ R_x = [\ X, \sin \iota_x, \cos \iota_x \ ]$$

and

$$[\ X, 1, 0 \ ] \ R_x = [\ X, \cos \iota_x, -\sin \iota_x \ ]$$

and solving, yields

$$R_x = \begin{vmatrix} 1 & 0 & 0 \\ 0 & \cos\theta_x & -\sin\theta_x \\ 0 & \sin\theta_x & \cos\theta_x \end{vmatrix} \qquad \text{(Eq. 7.5)}$$

The two matrices, R_y and R_x, are used to update the cumulative rotation matrix R. The cumulative rotation matrix represents the effects of all previous rotation operations. In the initial unrotated state the R matrix is defined by a 3 × 3 identity matrix. When an arrow key is pressed, R is multiplied by either R_y or R_x, depending on the key pressed. This is represented by

$$R = R_1 \ R_2 \ .... \ R_n,$$

where R_1, R_2, ... R_n equal either R_y or R_x and n equals the total number of rotations to date. The original coordinates of all joints are multiplied by R, and replotted, each time R is updated. When the structure is returned to the unrotated ("home") position, the R matrix is reinitialized to an identity matrix.

<u>Example</u>

Say that the point [1, 1, 1] is to be rotated $\pi/2$ radians about the Y axis. If the point is initially in the unrotated position then the cumulative R matrix before the rotation is given by

$$R = \begin{vmatrix} 1 & 0 & 0 \\ 0 & 1 & 0 \\ 0 & 0 & 1 \end{vmatrix}$$

Rotation about the Y axis requires multiplication by R_y, where

$$R_y = \begin{vmatrix} \cos \pi/2 & 0 & -\sin \pi/2 \\ 0 & 1 & 0 \\ \sin \pi/2 & 0 & \cos \pi/2 \end{vmatrix} = \begin{vmatrix} 0 & 0 & -1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{vmatrix}$$

So, $$R = \begin{vmatrix} 0 & 0 & -1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{vmatrix}\begin{vmatrix} 1 & 0 & 0 \\ 0 & 0 & -1 \\ 0 & 1 & 0 \end{vmatrix} = \begin{vmatrix} 0 & -1 & 0 \\ 0 & 0 & -1 \\ 1 & 0 & 0 \end{vmatrix}$$

The coordinates of the rotated point then would be

$$[1,1,1]\begin{vmatrix} 0 & 0 & -1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{vmatrix} = [1,1,-1]$$

as seen in Figure 7.5. If the point is then rotated $\pi/2$ radians about the X axis another matrix operation is required. Rotation about the X axis requires multiplication by R_x, where

$$R_x = \begin{vmatrix} 1 & 0 & 0 \\ 0 & \cos \pi/2 & -\sin \pi/2 \\ 0 & \sin \pi/2 & \cos \pi/2 \end{vmatrix} = \begin{vmatrix} 1 & 0 & 0 \\ 0 & 0 & -1 \\ 0 & 1 & 0 \end{vmatrix}$$

So,
$$R = \begin{vmatrix} 0 & 0 & -1 \\ 0 & 1 & 0 \\ 1 & 0 & 0 \end{vmatrix} \begin{vmatrix} 1 & 0 & 0 \\ 0 & 0 & -1 \\ 0 & 1 & 0 \end{vmatrix} = \begin{vmatrix} 0 & -1 & 0 \\ 0 & 0 & -1 \\ 1 & 0 & 0 \end{vmatrix}$$

The coordinates of the new rotated point then would be

$$[1, 1, 1] \begin{vmatrix} 0 & -1 & 0 \\ 0 & 0 & -1 \\ 1 & 0 & 0 \end{vmatrix} = [1, -1, -1]$$

as seen in Figure 7.5.

7.5   ANALYSIS

The operations performed during analysis were listed in Chapter 6 and are repeated here:
1.   Input of data.
2.   Calculation of member section properties.
3.   Formation and decomposition of stiffness matrix.
4.   Performance of AASHTO code checks and calculation of allowable stresses.
5.   Distribution of dead loads to joints.
6.   Calculation of dead load displacements and member end actions.
7.   Distribution of ice loads to joints.
8.   Calculation of ice load displacements and member end actions.
9.   Distribution of wind loads to joints.
10.   Calculation of wind load displacements and member end actions.
11.   Distribution of user-defined loads to joints.
12.   Combination of displacements, member end actions, stresses, and reactions for AASHTO group loadings.
13.   Calculation of maximum stresses and CSRs.
14.   Printing of results.

The operations unique to the program are explained in greater detail in the remainder of this chapter.

Section Properties

The section properties for tubular shapes are given in Table B-1 in the AASHTO Specifications (Reference 1). The section properties for tapered members are calculated from the average of the dimensions at the two ends of the member.

Dead Loads

Dead loads are automatically generated within the program by applying the member weights to the corresponding joints. This is shown as:

$$DL_i = A_i L_i DF$$

where DL_i = weight of ith member,
.   = 490 pcf (7849 Kg/m^3) = unit weight of steel,
A_i = cross sectional area of ith member,
L_i = length of ith member,
and   DF = detail factor (connections, etc.).

Ice Loads

An ice loading with a default value of 3 psf (143.64 Pa) is applied to the surfaces of all structural supports and on the face of all sign panels. An alternate ice load value may be input at the user's discretion. The ice load may be applied to both sides of a sign or only to one side.

Wind Loads

Wind loads, in accordance with AASHTO requirements, are generated automatically by calculating the wind pressure on the structure elements. This can be shown as

W_i = $P_{zi} A_i L_i$ = wind load due to ith member, where P_{zi} = 0.00256 $K_{zi}GV^2 I_r C_{di}$
P_i = wind pressure on ith member,
A_i = average area normal to axis of ith member,
L_i = length of ith member,
K_{zi} = exposure factor of ith member (Table 3.5 of Reference 1 or $2.01(z/z_g)^{2/\alpha}$),
C_{di} = Drag coefficient of ith member (Table 3.6 of Reference 1), and
I_r, G, V = Importance Factor (Table 3-2 of Reference 1), Gust factor (maximum of 1.14) and wind speed (Figure 3.2 of Reference 1).

Height coefficients are used to account for variation of wind velocity with respect to height. If a member falls within more than one zone then a weighted average is used. For example, if a member with a length of 20 ft (6.096 m) has 16.4 ft (5.0 m) which fall into a zone with a coefficient of 0.87 and 3.6 ft (1.096 m) which falls into a zone with a coefficient of 0.94 then the average coefficient is calculated to be

English Units - (0.87) (16.4/20) + (0.94) (3.6/20) = 0.88.

Metric Units - (0.87)(5.0/6.096) + (0.94)(1.096/6.096) = 0.88

Pressure due to wind loading is applied to the face of the sign but not to members which are shielded over more than 65% of their area. For members with less than 35% shielded,

the entire member is assumed to be completely exposed. If one member is shielded by another then the drag coefficients are modified in accordance with Ref. 1. The drag coefficients for the signs are also taken from Ref. 1.

Wind areas for tapered members are taken on the average area of the member normal to the wind direction.

User-Defined Loads

In addition to the automatically calculated wind, dead, and ice loads the user has the capability to define arbitrary joint loads. These loads may be forces in the X, Y, or Z directions or moments about the X-X, Y-Y, or Z-Z axes. Joint loads are always applied in the global direction.

Live Loads

Optional walkway live loads can be defined at the user's discretion. The magnitude of the load is input as well as the load's location on the structure. The live load is then distributed to the structure via the VAMs. Because walkways are not considered structural elements, the effect of the live load on the walkway is not checked.

Joint Loads

The forces due to the wind, dead, and ice loads are distributed to the joints under the assumption that the members are simply supported. <u>Fixed-end moments are not considered; it is assumed that the moments will cancel internally over the length of the unit</u>. The appropriate shear components, determined by the direction cosines, are applied to the corresponding degrees of freedom. The various loads are combined in accordance with AASHTO requirements, as summarized in Table 7.3, and the maximum effects calculated for each member individually.

Structural Analysis

The method employed for structural analysis is that of the general stiffness method for three-dimensional frames. This method is described in detail in Reference 2 and assumes the following:

1)   Linearly elastic material.
2)   Small deformations.
3)   No shear deformations.
4)   6 kinematic degrees of freedom at each joint -
        1 axial, 2 shears, 2 moments, and 1 torsion.

See Figure 7.6.

Special Members

Special members, such as connections, have been created in order to make the structural model complete. These special members are simplified representations of complex assemblages so stress results and code checks are not performed. Connection members are required to ensure structural continuity. A high moment of inertia is assigned automatically to represent their relatively high stiffness compared to that of the adjacent members.

Other special members are the VAMs and the horizontal members which compose the walkways and lighting support units. These are not considered structural elements, only as attachment points and dead load sources.

Stress Analysis

As part of the analysis procedure, SABRE performs a complete stress analysis of the structural members of the structure. This analysis is described as follows:

Allowable Member Stresses

Allowable stresses are required in the computation of the Combined Stress Ratio (CSR) of each member. The allowable stresses are a function of the member's usage (tower element, truss element, etc.), stress condition (tension or compression), effective length, and cross sectional properties. The equations used are taken from Reference 1 and are summarized in Tables 7.4 through 7.6.

Member Unit Stresses

The unit stress in a member is found by performing a complete stress analysis for each loading condition. The loading resulting in the largest stress for that member is then used for all subsequent computations. The specific formulas used in the stress computations for the various shapes are given in Table B-1 in Reference 1.

Interaction Relationships

The combined effects of moment, shear, and axial stresses are considered with the CSR interaction equations. These are given in Reference 1 and are summarized in Tables 7.5.1 and 7.5.2. The AASHTO specifications require that all members be designed so that their individual CSRs are less than or equal to one. This requirement restricts the member stress to less than that of the material yield stress divided by some safety factor.

7.6   BASE PLATE DESIGN

The program can calculate the quantities and dimensions of certain details pertaining to column base plates when given the following data:

1.   Yield stresses - bolts, base plate, and welds
2.   Column parameters - outside diameter, wall thickness, and shape (round, square, etc.)
3.   Base forces - X, Y, and Z directions

4.   Base moments - about the X-X, Y-Y, and Z-Z axes
5.   AASHTO governing load combination (DL, DL+W, etc.)

Given this data, the program calculates the following:

1.   Anchor bolts - quantity, diameter, area, separation, and embedment length
2.   Weld - throat and leg lengths
3.   Base plate - shape, diameter, and thickness

It is important to note that the design is based on finding a minimally sized plate and that it is up to the design engineer to verify the suitability of the results. The methodology for each of these is described in the following:

Data Entry

The column parameters, base forces, and base moments can be entered by hand or imported from an output file. When imported from an output file the results yielding the most conservative design are used. This is governed by the following rules:

1.   Column parameters are taken from the largest column in the structure.
2.   Base Forces - Base shears, V_x and V_z, are taken from the largest shear reactions found for all

        AASHTO load combinations - i.e. $[*V_x* + *V_z*]_{max}$.

        Base axial force, V_y, is taken from the smallest axial force found for all load combinations.
3.   Base Moments - Base biaxial moments, M_x and M_z, are taken from the largest moment reactions of all AASHTO combinations - i.e. $[*M_x* + *M_z*]_{max}$. Base torsion, M_y, is taken from the largest torsional reaction found for all load combinations.
4.   Governing Load Combination - The load combination resulting in the largest base reactions is used.

Anchor Bolt Calculation

Design of the base plate anchor bolts is an iterative process which starts with a minimum number of bolts (four) and proceeds through a series of calculations until either an acceptable design is found or a maximum number of bolts (eight) is exceeded. Each time an unacceptable design is found the number of bolts is incremented by two and the process is repeated.

According to Reference 1, anchor bolts subject to combined shear and tension loads may be proportioned using

$$F_v^2 > f_v^2 + (k\, f_t)^2$$

where f_v $= V / A_b =$ actual bolt shear stress,
 f_t $= P / A_b =$ actual bolt tensile stress,
 k $= F_v / F_t =$ combined shear/tension constant,
 F_v $= 0.30\, F_y =$ allowable bolt shear stress,
 F_t $= 0.50\, F_y =$ allowable bolt tensile stress,

$$V = \sqrt{\left(V_x + V_{tx}\right)^2 + \left(V_z + V_{tz}\right)^2} = \text{ actual shear,}$$

 V_x $=$ shear in X direction,
 V_z $=$ shear in Z direction,
 V_{tx} $=$ shear in X direction due to torsion,
 V_{tz} $=$ shear in Z direction due to torsion,

$$P = \frac{|M_x| + |M_z|}{B_c\, B_n / 2} - \frac{V_y}{B_n} = \text{ actual bolt tension, ignore } V_y \text{ term if in compression.}$$

or 50% of the maximum column allowable load

 M_x $=$ moment about X axis,
 M_z $=$ moment about Z axis,
 V_y $=$ axial force,
 A_b $=$ bolt area,
 B_c $=$ bolt separation, (or opposite bolt distance, i.e. column diameter + 6")
 B_n $=$ number of bolts,
and F_y $=$ bolt yield stress.

As can be seen, the total shear force on a bolt, V, is found considering the shear forces in both the X and Z directions as well as the shear due to torsion. The total tensile force on a bolt, P, is found considering biaxial bending, M_x and M_z, as well as the axial force, V_y. These reactions are illustrated in Figure 7.7.

A three-inch separation distance is initially set from the outside column wall to the bolt center. If this distance is subsequently found to be insufficient it is increased and the process is repeated until a satisfactory design is found or the assumed limits of the design are exceeded.

The embedment length of an anchor bolt, L, is based upon bond stress and is given by

$$ L = \frac{P}{135\pi D_b (SF)} $$

where 135 = .045 f'$_c$ = allowable bond stress (Ref. 6),
        f'$_c$ = 3000 psi (20.684 MPa) (assumed),
and   D$_b$ = bolt diameter.
        SF = 1 for group load 1 and 1.33 for others.

Weld Design

The column-plate connection is made with a continuous fillet weld which is designed for shear. The force on the weld is found by uniformly distributing all reactions (forces and moments) to equivalent forces per unit length of weld and finding the resultant force (Reference 7). The design force, f$_r$, is the maximum force of f$_1$ and f$_2$, which can be shown as

$$ f_1 = \sqrt{\left(f_{vx} + f_t\right)^2 + \left(f_{vz} + f_t\right)^2 + \left(f_a + f_{bz} + f_{bx}\right)^2} $$

$$ f_2 = \sqrt{\left(f_{vxz}\right)^2 + \left(f_{mxz}\right)^2} $$

where f_{vx} = V$_x$/C = shear force, X direction,

        f_{vz} = V$_z$/C = shear force, Z direction,

        f_t = M$_y$ D$_c$ / 2 J$_w$ = torsional force,

        f_a = V$_y$/C = axial force,

        f_{bx} = M$_x$/S$_w$ = bending force, X-X axis,

        f_{bz} = M$_z$/S$_w$ = bending force, Z-Z axis,

        f_{vxz} = f$_{vc}$ * A$_c$ / C = capacity of shear force,

        f_{mxz} = f$_{bc}$ * S$_c$ / S$_w$ = capacity of moment,

$$f_{vc} \quad = 0.33 * f_{yc} * SF \text{ (safety factor)} * 0.5$$

$$f_{bc} \quad = 0.66 * f_{yc} * SF \text{ (safety factor)} * 0.5$$

$$C \quad = \text{weld length} = \text{column circumference,}$$

$$D_c \quad = \text{column outside diameter,}$$

$$A_c \quad = \text{area of column,}$$

$$S_c \quad = \text{section modulus of column,}$$

$$J_w \quad = \text{polar moment of inertia per unit length,}$$

and $\quad S_w \quad = \text{section modulus per unit length.}$

The weld dimensions are found from

$$W_l \quad = f_r / (.707 \, F_w) = \text{leg length}$$

and $\quad W_t \quad = .707 \, W_l = \text{throat length,}$

where $\quad F_w \quad = \text{allowable weld stress.}$

The calculated weld dimensions are checked against minimum and maximum AISC requirements, Reference 8. If the dimensions are found to be not in accordance with AISC requirements then the dimensions are adjusted accordingly.

Plate Design

Shape of the base plate depends on the number of anchor bolts required; four bolts require a square plate, 6 bolts require a hexagonal plate, and eight bolts an octagonal plate.

Size of the plate is based on the diagonal separation distance of the bolts plus an additional five inches of clearance to the plate edge.

Thickness of the plate is governed by bending along the critical section shown in Figure 7.7. The effective width of the plate at that point along the critical section is given by

English Units

$$B = 2d\sqrt{X} - (D_b + .25) \qquad \text{-- - - for square, hexagonal, octagonal}$$

where $d = 2.5$ inches,
$X = 1$ for Square plate, $\cong 2$ for Hexagonal, $= \tan 67.5$ for Octagonal

and D_b + .25 = bolt hole diameter.
$B = \Pi\, B_c\, /\, B_n\, - (D_b + .25)$ - - - for round

Metric Units

$$B = 2d\sqrt{X} - \left(D_b + 6.4\right)$$ - - - for square, hexagonal, octagonal

where d = 63.5 mm
X = 1 for Square plate, $\cong$ 2 for Hexagonal, = tan 67.5 for Octagonal
and D_b + 6.4 = bolt hole diameter.
$B = \Pi\, B_c\, /\, B_n\, - (D_b + 6.4)$ - - - for round

Plate thickness is found from $t = \sqrt{6S/B}$

where S $= M_e\, /\, F_b$ = required section modulus,
       M_e $= P\, e$ = moment due to eccentricity,
       e $= (B_c - D_c - 2\, W_1)\, /\, 4$ = eccentricity,
and F_b = .66 Fy = allowable bending stress of plate.

Eccentricity, e, is measured to the assumed point of contraflexure located midway between the bolt and the outside edge of the weld.

For all of the preceding, the allowable values are increased by 40% for load combinations including wind effects, in accordance with AASHTO recommendations.

7.7   FATIGUE CHECK

The SABRE program can do fatigue check for cantilevered steel and aluminum sign structures when importing the following data from the fatigue file:

1.   Base moment ranges – about X-X and Z-Z axes; and
2.   Column parameters – outside diameter, wall thickness, and cross section shape.

The user also needs to provide the following data:

1.   Base plate parameters – bolt diameter, bolt thread pitch, farthest bolt diameter to the center and number of bolts; and
2.   Stiffener parameters – height, width, thickness and total number.

Given these data, the program calculates the following:

1.   Anchor bolt stress range;
2.   Stress range at column-to-baseplate connection;
3.   Stress range at stiffener-to-baseplate connection; and
4.   Stress range at termination of stiffener.

Base Moment Range Calculation

Three types of loading (galloping, natural wind gust and truck-induced gust) should be considered for cantilevered steel and aluminum sign structures based upon the new (5th edition) AASHTO Standard Specifications for Structural Supports for Highway Signs, Luminaries and Traffic Signals.

Galloping

Overhead cantilevered sign structures shall be designed for galloping-induced cyclic loads by applying an equivalent static shear pressure vertically to the surface area as viewed in normal elevation of all sign panels. The magnitude of this vertical shear pressure range shall be equal to the following:

$$P_G = 1000 \cdot I_F (P_a) \qquad \text{(Metric Units)}$$
$$P_G = 21 \cdot I_F (\text{psf}) \qquad \text{(English Units)}$$

where I_F is the importance factor applied to limit state wind load effects to adjust for the desired level of structural reliability.

Natural Wind Gust

Cantilevered overhead sign structures shall be designed to resist an equivalent static natural wind gust pressure range of

$$P_{NW} = 250 C_d I_F (P_a) \qquad \text{(Metric Units)}$$
$$P_{NW} = 5.2 C_d I_F (\text{psf}) \qquad \text{(English Units)}$$

Where C_d is the wind drag coefficient specified in Table 3-6 (AASHTO Specifications).

The design natural wind gust pressure range is based on a yearly mean speed of 5 m/s (11.2 mph). For locations with more detailed wind records, particularly sites with higher wind speeds, the natural wind gust may be modified.

Truck-Induced Gust

Overhead sign structures shall be designed to resist an equivalent static truck gust pressure range of

$$P_{TG} = 900 C_d I_F (P_a) \qquad \text{(Metric Units)}$$
$$P_{TG} = 18.8 C_d I_F (psf) \qquad \text{(English Units)}$$

The equivalent static truck pressure range may be reduced for locations where vehicle speeds are less than 30 m/s (65 mph).

7.8   Fatigue Design Examples for Highway Sign Details

This section deals with the fatigue design of connections used for highway sign structures and luminaries. Some excel templates have been prepared for this paper. Following is a description of each sheet.

a. Built-up box

Description: This consists of two small gussets fillet welded to the main member which are connected to a main gusset plate. Another gusset plate with mast arm connected to it is bolted with the main gusset. The connections and diagrams are described as Details # 5, 17 and 19 in NCHRP report 469, pg 11-15, Table 11-2. Diagram is given on pg 11-21. For above calculations, NCHRP report 469, pg B14 is used as reference.

Maximum Permissible Stress: The maximum permissible CAFL (Constant-Amplitude Fatigue Limit) for the welded connection as per Detail 19 is corresponding to category 'ET' and Detail 17 to category E'. Hence the stress induced should be less than 1.2 ksi and 2.6 ksi for steel connection categories 'ET" and E', respectively. For bolted connections, as per Detail 5, the category is 'D' and the CAFL is 7 ksi.

b. Fillet welded tube-to-tube pass through connection

Description: This connection consists of two tubes. A stub passes through the column and it is welded along the perimeter on both sides of the column. The connections and diagrams are described as Details # 18 and 19 (pass-through & no-pass-through fillet welded tube-to-tube connections, respectively) in NCHRP report 469, pg 11-15, Table 11-2. The diagram is given on pg 11-21 (Example 9). For above calculations, NCHRP report 469, pg B75 is used as reference.

Maximum Permissible Stress: As per the note f, pg11-17 in the NCHRP report, stresses at the bottom of the connection should be checked. The maximum permissible CAFL at the bottom of the column is corresponding to category E. Hence the stress should be less than 4.5 ksi.   For the stub, the CAFL should be less than permitted in category E' for detail 18 for pass-through (or ET for Detail 19 for no pass-through). Hence the stress should be less than 2.6 ksi for Detail #18 (or 1.2 ksi for Detail #19.)

c. Fillet welded socket connection

Description: This connection is used for base plate – column connection. The plate has a hole equal to the external diameter of the column with some clearance. The column and plate are welded from inside and from outside as shown in the figure. The connections and diagrams are described as Detail # 16 (fillet welded connection) in NCHRP report 469, pg 11-15, Table 11-2. Diagram is given on pg 11-20 (Example 7). For the above calculations, NCHRP report 469, pg B94 is used as reference.

Maximum Permissible Stress: The maximum permissible CAFL for the connection corresponds to category E'. Hence the stress calculated should be less than 2.6 ksi. For the bolts, CAFL corresponds to category D. Hence the stress induced should be less than 7 ksi.

d. Reinforced handhole connection

Description: In case of a column with a handhole, with width and the length as 'W' and 'Lh', it has to be checked whether the dimensions of hole are in permissible limit or not. The connections and diagrams are described as Detail # 20 in NCHRP report 469, pg 11-15, Table 11-2. Diagram is given on pg 11-20.
For above calculations, NCHRP report 469, pg B94 is used as reference.

Maximum Permissible Stress: To calculate the maximum permissible stress, the limitations given in NCHRP report pg11-15 should be referred. Depending on the dimensions of the hole, the stress category may be C, D or E (10 ksi, 7 ksi or 4.5 ksi, respectively.) The stress category depends on length of hole and thickness of column. This excel sheet calculates the stress category and hence the permissible stress automatically depends on the input.

e. Slotted tube-to-gusset connection

Description: This connection consists of a main member attached to a stub through a gusset plate. Gusset plate slides into the slot in the stub and then it is welded. The connections and diagrams are described as Details # 15 and 24 in NCHRP report 469, Pg 11-15, Table 11-2. Diagrams are given on pgs 11-20 and 23.

Maximum Permissible Stress: The maximum permissible CAFL at the connection between stub and the gusset plate (Detail #15) corresponds to category E'. Hence the stress should be less than 2.6 ksi. For the connection between main member and column, the stress category depends on R (Detail #24 of category D of 7 ksi with R > 2" and E of 4.5 ksi with R ≤ 2", respectively.) This sheet calculates the stress category and corresponding CAFL as described in pg11-16 of NCHRP report.

f. Stiffened base plate

Description: This consists of a base plate stiffened by stiffener pates. Column is connected to base plate through stiffener plates. Base plate is connected to abutments through anchor bolts. Anchor bolt stress calculation and permissible stress in anchor bolts are similar to sheet #3 hence it is not discussed again in this document. In case of stiffener plate connections, the connections and diagrams are described as Details # 12, 21 and 23 in NCHRP report 469, pg 11-15, Table 11-2. Diagrams are given on pgs 11-22 and 23.

Maximum Permissible Stress: The maximum permissible CAFL at the connection between column and stiffener group corresponds to category E' (Detail #12). Hence the CAFL should not exceed 2.6 ksi. The maximum permissible CAFL at the connection between stiffener plate and base plate corresponds to category C and it should not exceed 10 ksi (Detail #23). The maximum

permissible stress at the termination of stiffener plate corresponds to category E' and should not exceed 2.6 ksi (Detail #21).

TABLE 7.1 – COMPARISONS OF CALCULATED AND TABLE VALUES OF K_t							
PROB. NO.	SHAPE	NO. OF SIDES	OUTER DIAM.	WALL THICK.	INNER RAD.	K_t	
						CALC.	AASHTO
1	Square	4	50.25	.25	.25	1.43	1.44 ±
2	Octagonal	8	16.92	.25	.50	1.20	1.20 ±
3	Dodecagonal	12	5.25	.25	.75	1.05	1.06 ±

Configuration	STRUCTURE	COMPONENTS
1 Basic Structure		
2		
3		
4		
5		

TABLE 7.2 – SIGN BRIDGE CONSTRUCTIONS

TABLE 7.3 – AASHTO GROUP LOAD COMBINATIONS			
GROUP	LOADS	% ALL. STRESS	DESCRIPTION AND LOAD COMBINATION
1	DL	100	Case 1: (Detail Factor) × (Member Weight) only.
2	DL + W	133	Case 1: Dead load + wind in the +z direction with wind combination 1 (100% normal + 20% transverse components)
			Case 2: Dead load + wind in the +z direction with wind combination 2 (60% normal + 30% transverse components)
			Case 3: Dead load + wind in the −z direction with wind combination 1 (100% normal + 20% transverse components)
			Case 4: Dead load + wind in the −z direction with wind combination 2 (60% normal + 30% transverse components)
3	DL + ½ W + ICE	133	Ice loads assumed acting on all exposed surfaces of all members other than signs. Ice loads assumed acting only on one face of signs.
4	FATIGUE	See Note	Galloping, natural wind gust and truck-induced gust considered for cantilevered steel and aluminum sign structures.
Note: See Section II (AASHTO SPECIFICATIONS) fro fatigue loads and stress range limits.			

<table>

STRESS TYPE	EQUATION ID	EQUATION	RANGE	COMMENTS
Axial F_a	Tensile A1	$0.60\,F_y$ $0.50\,F_u$	Gross area A_g Effective area A_e	Untrussed pole type tower members
	Compres. A2	$$\dfrac{12\pi^2 E}{23\left(\dfrac{KL}{r}\right)^2}$$	$$\dfrac{KL}{r} \geq \left[\dfrac{2\pi^2 E}{F_y}\right]^{\frac{1}{2}}$$ $$= C_c$$	All tower members except un-trussed poles. All other primary and secondary members.
	A3	$$\dfrac{\left[1-\dfrac{(KL/r)^2}{2C_c^2}\right]F_y}{\dfrac{5}{3}+\dfrac{3(KL/r)}{8C_c}-\dfrac{(KL/r)^3}{8C_c^3}}$$	$$\dfrac{KL}{r} < \left[\dfrac{2\pi^2 E}{F_y}\right]^{\frac{1}{2}}$$ $$= C_c$$	Reference 1,

<p align="center">TABLE 7.4.1 – ALLOWABLE UNIT STRESS RELATIONSHIPS</p>
</table>

Note: Case 1: Vertical support (pole type), K is assumed as 2.0 by the program
Case 2: Vertical support (truss type), K is assumed as 1.2 by the program
Case 3: Horizontal supports (pole and truss type), K is assumed as 0.65 by the program
Case 4: All secondary members, K is assumed as 0.65 by the program

TABLE 7.4.2 – ALLOWABLE UNIT STRESS RELATIONSHIPS

STRESS TYPE	EQUATION ID	EQUATION	RANGE[♦]	COMMENTS
Bending F_b	B1	$0.66\,F_y$	$\lambda \leq \lambda_p$	Round Compact Tubes
	B2	$\left[1 + \dfrac{0.09\left(E/F_y\right)}{\left(D/t\right)}\right]0.39F_y$	$\lambda_p < \lambda \leq \lambda_r$	Round Noncompact Tubes
	B3	$\left[1 + \dfrac{0.09\left(E/F_y\right)}{\left(D/t\right)}\right]0.39F_y$	$\lambda > \lambda_r$	
	B4	$a_1 F_y$	$\lambda \leq \lambda_p$	Polygonal Compact Tubes
	B5	$a_2 F_y\left[1 - \dfrac{a_3}{\sqrt{E/F_y}}\dfrac{b}{t}\right]$	$\lambda_p < \lambda \leq \lambda_r$	Polygonal Noncompact Tubes
	B6	$a_4 F_y\left[1 - \dfrac{a_5}{\sqrt{E/F_y}}\dfrac{b}{t}\right]$	$\lambda > \lambda_r$	

Tube	a_1	a_2	a_3	a_4	a_5
Hexdecagonal	0.66	1.71	0.55	0.74	0.23
Dodecagonal	0.65	1.15	0.39	0.75	0.22
Octagonal	0.64	0.96	0.30	0.73	0.19
Rectangular	0.60	0.82	0.24	0.74	0.19

♦ See TABLE 5-3 of 2009 AASHTO "Standard Specifications for Structural Supports for Highway Signs, Luminaires and Traffic Signals"

TABLE 7.4.3 – ALLOWABLE UNIT STRESS RELATIONSHIPS

STRESS TYPE	EQUATION ID	EQUATION	RANGE	COMMENTS
Shear F_v	V1	$0.33\,F_y$	$\dfrac{D}{t} \le 1.16\left(\dfrac{E}{F_y}\right)^{2/3}$	Round Tubes
	V2	$\dfrac{0.41E}{\left(\dfrac{D}{t}\right)^{3/2}}$	$\dfrac{D}{t} > 1.16\left(\dfrac{E}{F_y}\right)^{2/3}$	
	V3	$0.33\,F_y$	$\dfrac{b}{t} \le 2.23\sqrt{\dfrac{E}{F_y}}$	Polygonal Tubes
	V4	$\dfrac{1.64E}{\left(b/t\right)^2}$	$\dfrac{b}{t} > 2.23\sqrt{\dfrac{E}{F_y}}$	
	V5	$0.33\,F_y$	$\dfrac{h}{t_w} \le 2.23\sqrt{\dfrac{E}{F_y}}$	Other shapes

		TABLE 7.5.1 – INTERACTION EQUATIONS		
COMPONENT	EQUATION ID.	CSR	COMMENTS	FIGURE
Pole Type Tower Member, Untrussed	Compres. I1	$\dfrac{f_a}{.6F_y} + \dfrac{f_b}{C_A F_b} + \left[\dfrac{f_v}{F_v}\right]^2$	C_A = Coeff. Of Amplification See Ref. 1	
Trussed Tower Member	Compres. I2	$\dfrac{f_a}{F_a} + \dfrac{f_b}{(1 - f_a/F_a)F_b} + \left[\dfrac{f_v}{F_v}\right]^2$	Sect. Stabil. $F_a = .52\dfrac{\pi^2 E}{(K1/r)^2}$ Overall Stab. $\underline{K = 2}$	
Trussed Tower Member	Tensile I3	$\dfrac{f_a}{.6F_y} + \dfrac{f_b}{C_A F_b} + \left[\dfrac{f_v}{F_v}\right]^2$		
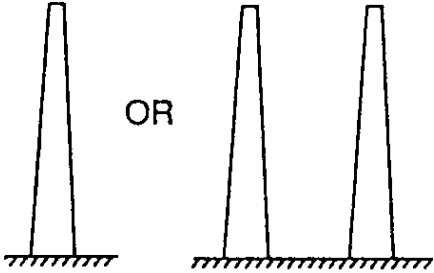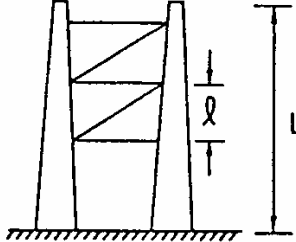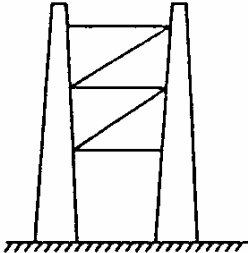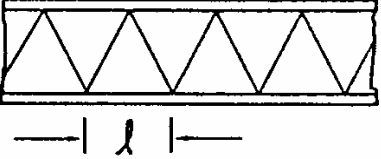
		TABLE 7.5.2 – INTERACTION EQUATIONS		
COMPONENT	EQUATION ID.	CSR	COMMENTS	FIGURE
Horizontal and inclined truss support members	Compres. I4	$\dfrac{f_a}{F_a} + \dfrac{f_b}{(1 - f_a/F_a)F_b} + \left[\dfrac{f_v}{F_v}\right]^2$	$F_a = .52 \dfrac{\pi^2 E}{(K1/r)^2}$ where K = ¾	
Horizontal and inclined truss support members	Tensile I5	$\dfrac{f_a}{.6F_y} + \dfrac{f_b}{F_b} + \left[\dfrac{f_v}{F_v}\right]^2$		
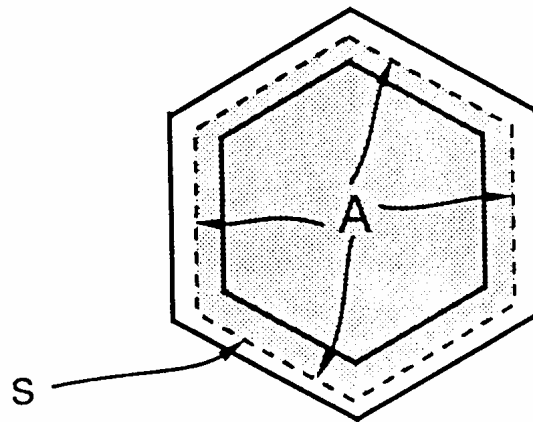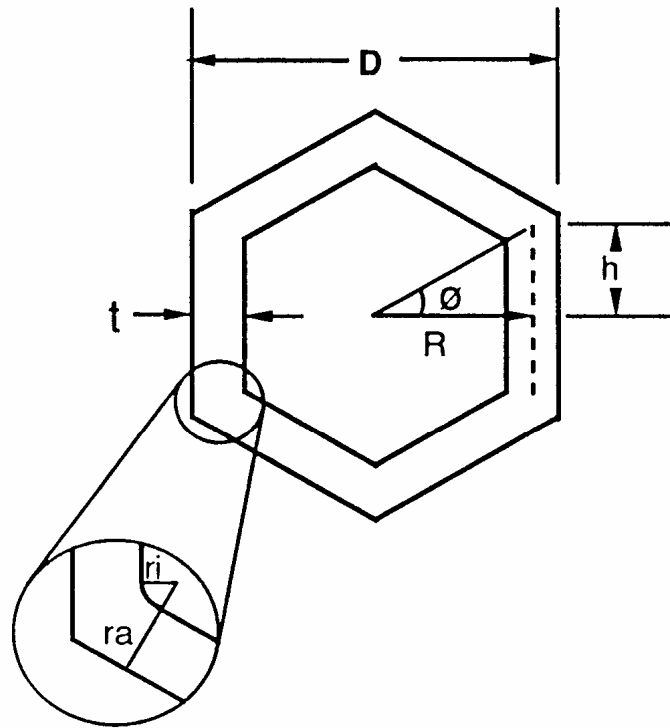
Figure 7.1 Torsional Factors

Alternate A
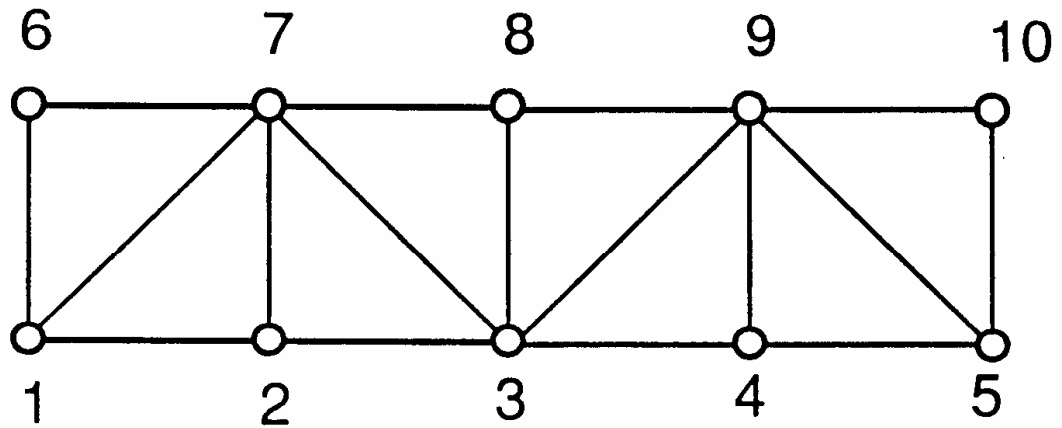
Alternate B
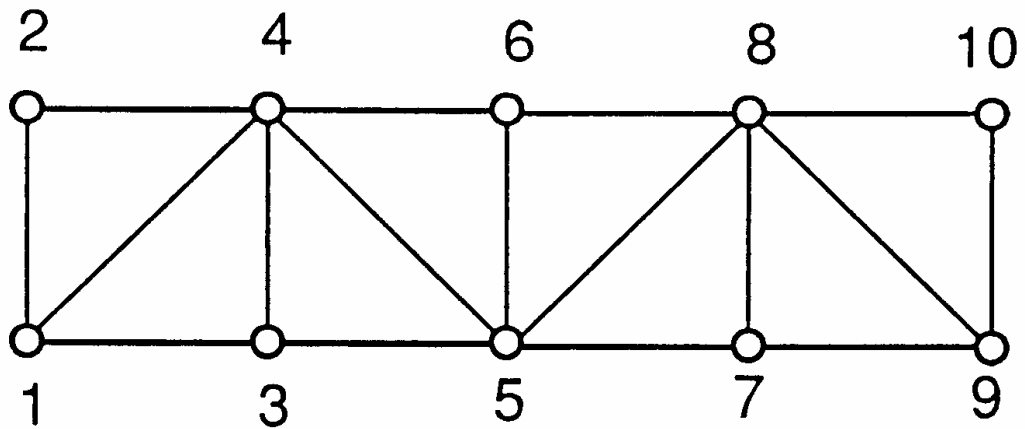
Figure 7.2 Example of Possible Joint Numbering Schemes
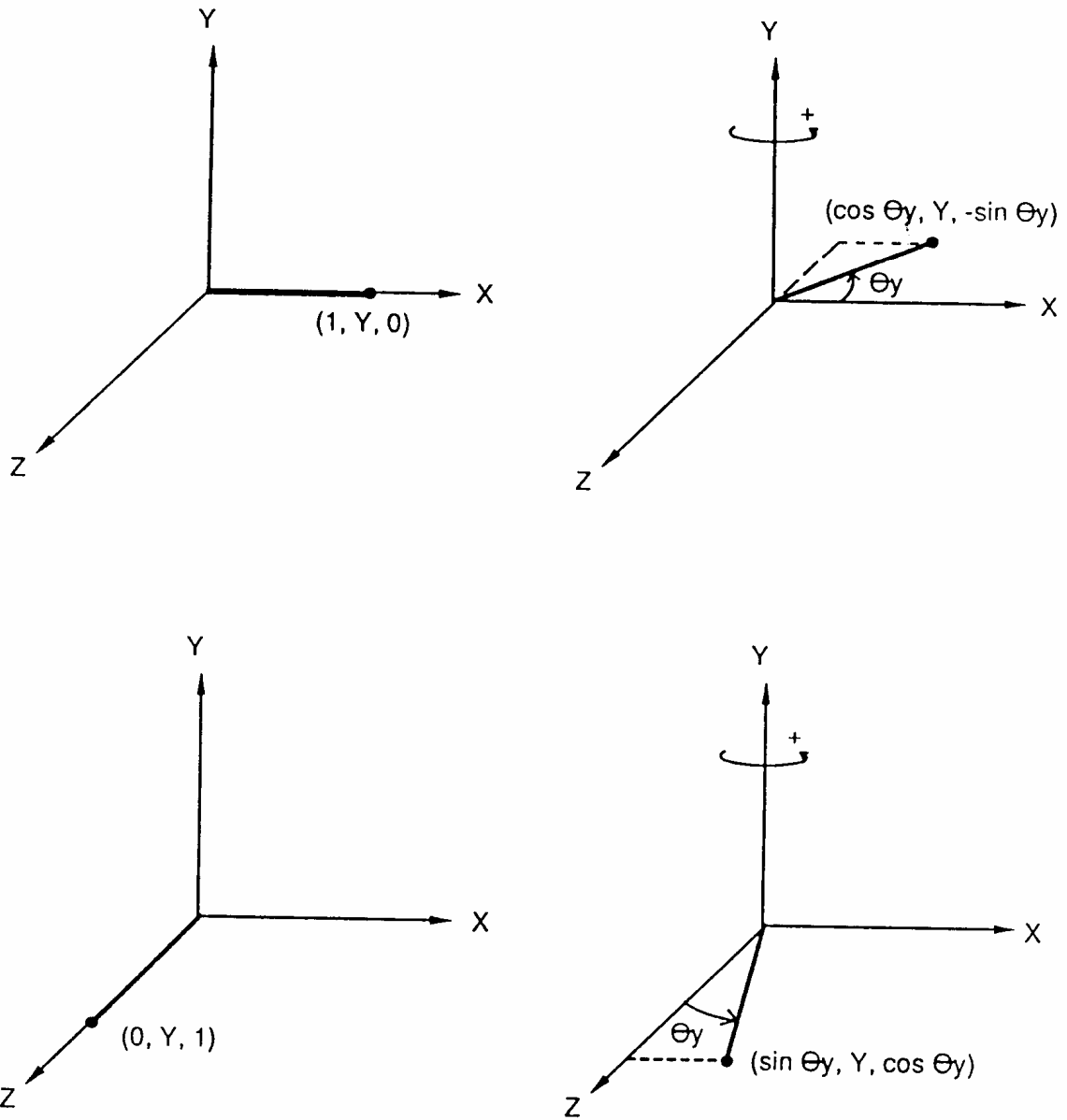
Figure 7.3 Example Rotation about Y Axis
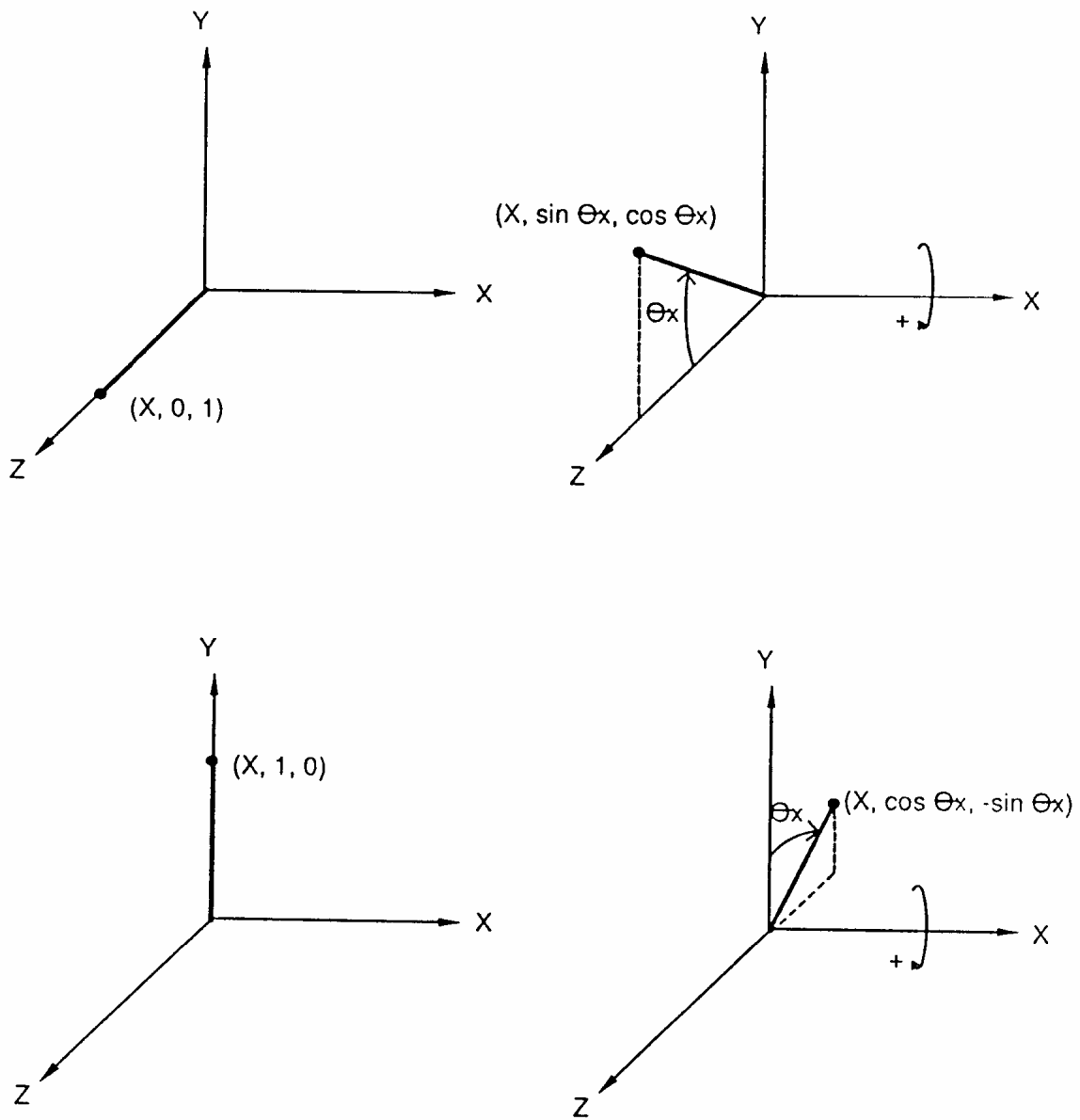
Figure 7.4 Example Rotation about X Axis
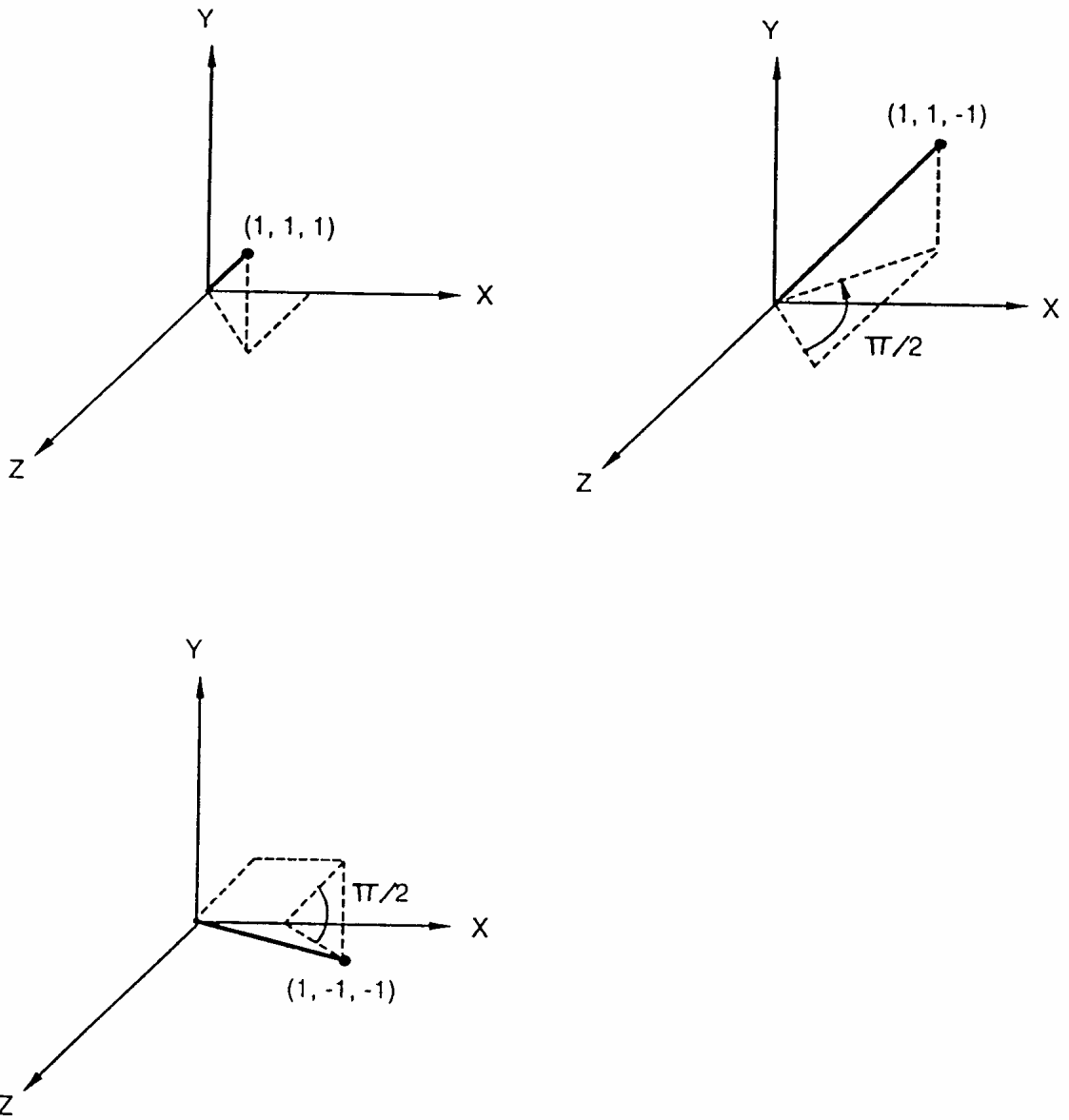
Figure 7.5 Example Rotation about Y and X Axes
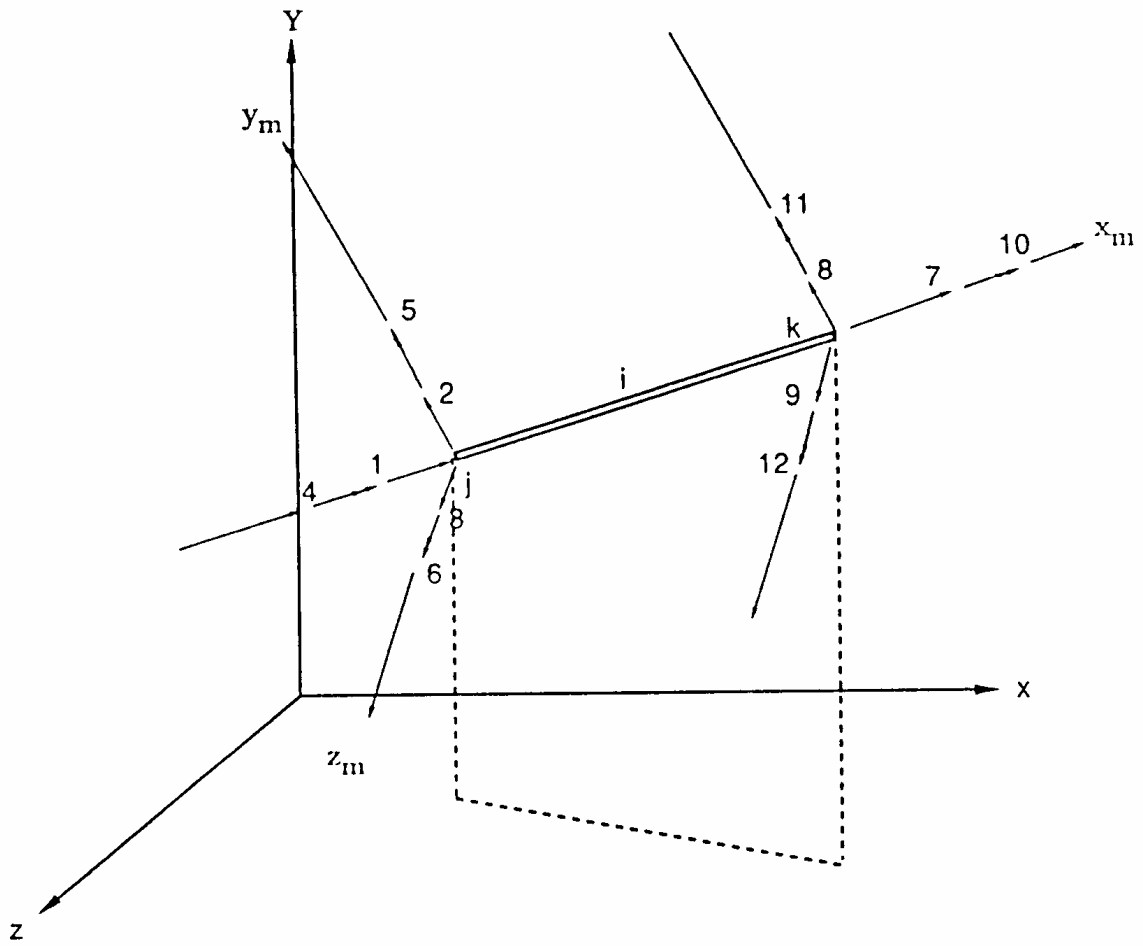
Figure 7.6 Space Frame Member Numbering System
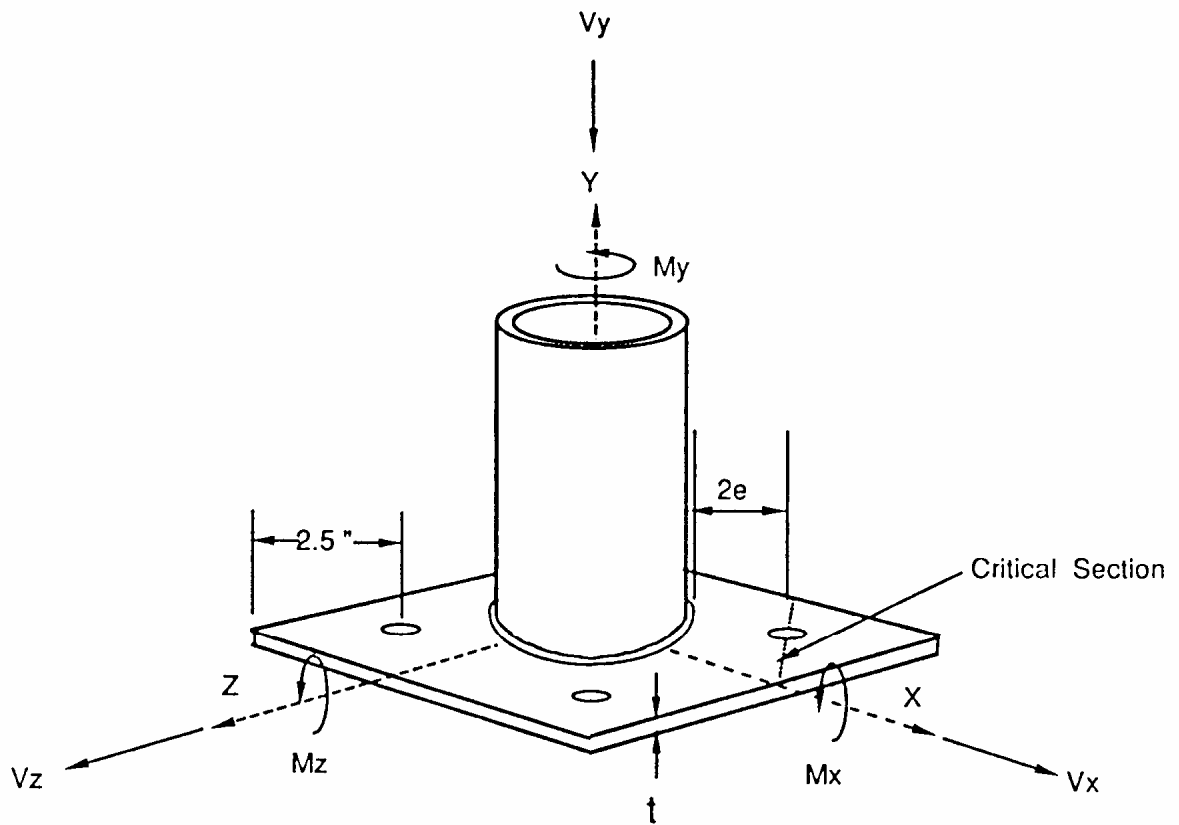
Figure 7.7 Base Plate Design Factors

Appendix A – Input Screens

A.1 System Input

Data Type 01012, 01022 – Project Data

ITEM	UNIT	FORMAT	DESCRIPTION
DESCRIPTION	none	alphanumeric	Enter the project description, location, or any other pertinent information
DATE	none	alphanumeric	Enter the date of the data entry
CONTRACT NUMBER	none	alphanumeric	Enter the project contract number
STRUCTURE NUMBER	none	alphanumeric	Enter the structure identification number
STRUCTURE UNIT	none	alphanumeric	Enter the structure unit number
DESIGNED BY	none	alphanumeric	Enter the designer's name
CHECKED BY	none	alphanumeric	Enter the name of the person who checked the input data
SPECIFICATION	none	alphanumeric	Enter the project specification number

Data Type 01032 – Program Options

ITEM	UNIT	FORMAT	DESCRIPTION
OUTPUT LEVEL	none	integer	Enter 1 if the basic level of output is desired; Enter 2 if the basic level of output plus additional detail is desired.
TYPE OF UNIT	none	integer	Enter 0 for English Units; Enter 1 for Metric Units.
MATERIAL ID	none	integer	Enter 0 for Steel; Enter 1 for Aluminum. Enter 2 for Steel Posts and Aluminum Span
MODULUS OF ELASTICITY	none	integer	Default 206,850 Mpa (30,000 ksi) for steel Default 68,950 Mpa (10,000 ksi) for aluminum
ALUMINUM TYPE ID	none	integer	Enter 1 for 6061-T651 Enter 2 for 6061-T6 Enter 3 for 6063-T6 Enter 4 for 6065 Enter 5 for 5086-H34 Enter 6 for 6061-T651/T6 welded Enter 7 for 6063-T6 welded Enter 8 for 6005-T6 welded Enter 9 for 5086-H34 welded
6061 – T6 WELDED W/4043 ALLOY FILLER WIRE	none	integer	Enter 0 for no; Enter 1 for yes

A-3

A.2 Structure Generation Input
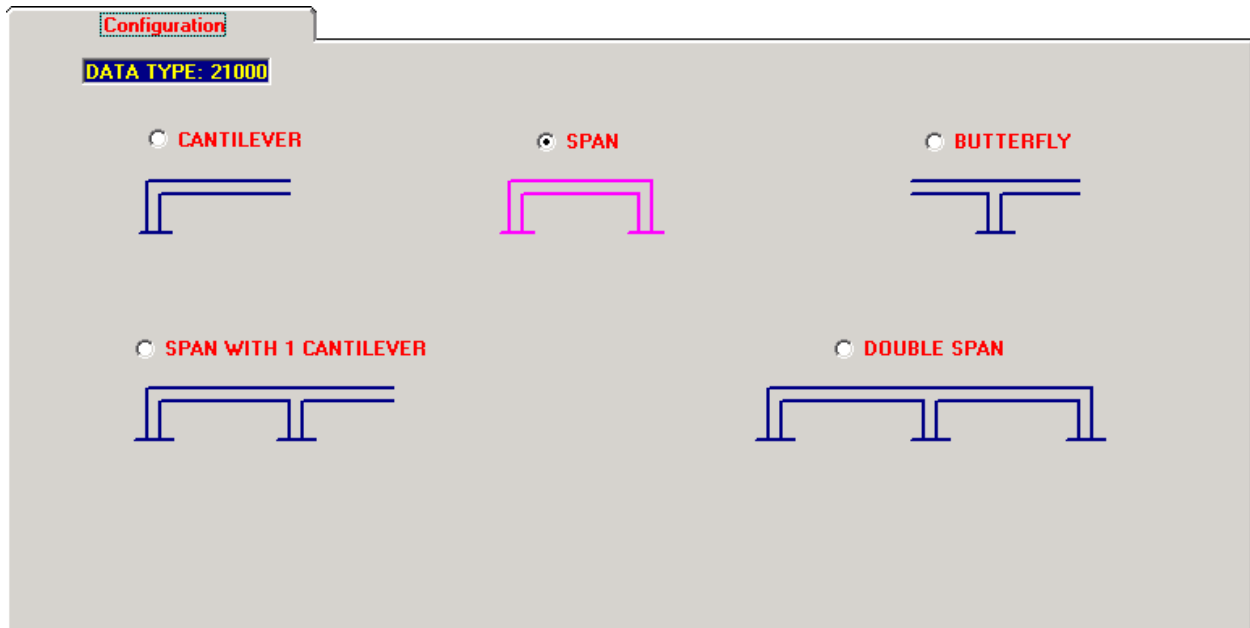
Data Type 21000 – Configuration

ITEM	UNIT	FORMAT	DESCRIPTION
CONFIGURATION	none		To select a configuration, click the mouse to the desired choice. The current configuration is highlighted with a bright color. A configuration must be chosen if automatic joint and member generation is desired.

Data Type 31000 – Element Definition

ITEM	UNIT	FORMAT	DESCRIPTION
CHORDS	none	integer	Enter one of the following numbers: 1 – monotube 2 – plane truss 3 – tri-chord truss 4 – box truss
SEGMENTS	none	integer	Enter the number of segments desired in each beam chord or tower post. For a span type beam (i.e., supported by a tower on each end) only an even number of segments are allowed. No more than 16 segments are allowed for a span type beam chord.
POSTS	none	integer	Enter one of the following numbers: 1 – single post; 2 – two trussed posts

Data Type 34000 – Dimensions

ITEM	UNIT	FORMAT	DESCRIPTION
BEAM LENGTH	feet(m)	real	For a cantilevered beam, enter the length of the beam, measured from the center of the supporting tower to the end of the beam. For a span beam enter the length of the beam, measured from the centers of both, measured from the centers of both supporting towers.
BEAM ELEV.	feet(m)	real	Enter the elevation of the lowest chord of the beam.
BEAM DEPTH	feet(m)	real	Enter the depth of the trussed beam; i.e., the distance between the highest chord(s) and the lowest chord(s).
CONNECTION LENGTH	feet(m)	real	Enter the length of the connection member which attaches the beam chord(s) to the tower post(s). This length should not exceed 2.5ft (.76m) for accurate results. For a trussed tower with two posts, this value may equal 0. For all other structures, this value must exceed 0.
TOP ELEV.	feet(m)	real	Enter the elevation of the top of the tower. For a trussed tower with two posts, both posts will be assumed to have the same top elevation.
BOT. ELEV.	feet(m)	real	Enter the elevation of the bottom of the tower. For a trussed tower with two posts, both posts will be assumed to have the same bottom elevation.
TOWER WIDTH	feet(m)	real	Enter the width of the trussed tower(s); i.e., the distance between the from post and the rear post.
LAST BRACE	feet(m)	real	Enter the vertical distance from the bottom of the beam to the point below where the tower bracing is to end.
FIRST BRACE	feet(m)	real	Enter the vertical distance from the base of the tower to the point above where the tower bracing is to begin.

A-6

DATA TYPE: 33000

Cross Sections
Single Span

LCT   BL   BC   BR   RCT
[1]   [1]   [1]   [1]   [1]

[1] BLC   BRC [1]

LCB   RCB
[1]   [1]

CB =Column Bottom section no
CCB =Center Column Bottom section no
CCT =Center Column Top section no
CT =Column Top section no
BC =Beam Center
BL =Beam Left section no
BLC =Beam Left Connection
BR =Beam Right section no
BRC =Beam Right Connection
LBC =Left Beam Center
LBL =Left Beam Left section no
LBLC =Left Beam Left Connection
LBR =Left Beam Right section no

Data Type 33000 – Cross Sections

ITEM	UNIT	FORMAT	DESCRIPTION
CROSS SECTION	none	integer	Enter the section number (from Screen "Definition of Sections").

A-7

Data Type 38000 – Bracing

ITEM	UNIT	FORMAT	DESCRIPTION
FRONT PATTERN	none	integer	Front truss face
TOP PATTERN	none	integer	Top truss face
REAR PATTERN	none	integer	Rear truss face
BOTTOM PATTERN	none	integer	Bottom truss face
PERP. SECTION	none	integer	Enter the section number (from Screen "Definition of Sections"), which defines the cross section properties for the truss members attached perpendicularly to the main members. (Only tubular sections are allowed for perpendicular members.)
DIAG. SECTION	none	integer	Enter the section number (from Screen "Definition of Sections"), which defines the cross section properties for the truss members attached diagonally to the main members. (Only tubular sections are allowed for diagonal members.)
PATTERN, TOWER	none	integer	Tower bracing 1 – Pratt truss 2 – Pratt truss reversed 3 – Warren truss 4 – Warren truss reversed Note: If there is no perpendicular member for any pattern, leave perpendicular section blank.

Data Type 39000 – Yield Stresses (Steel only)

ITEM	UNIT	FORMAT	DESCRIPTION
TOWER YIELD STRESS (MAIN MEMBER)	ksi (Mpa)	real	Enter the material yield stress for the main (primary) members of the tower.
TOWER YIELD STRESS (PERP. BRACING)	ksi (Mpa)	real	Enter the material yield stress for the bracing members attached perpendicularly to the main members of the towers.
TOWER YIELD STRESS (DIAG. BRACING)	ksi (Mpa)	real	Enter the material yield stress for the bracing members attached diagonally to the main members of the towers.
BEAM YIELD STRESS (MAIN MEMBER)	ksi (Mpa)	real	Enter the material yield stress for the main (primary) members of the beams.
BEAM YIELD STRESS (PERP. BRACING)	ksi (Mpa)	real	Enter the material yield stress for the bracing members attached perpendicularly to the main members of the beams.
BEAM YIELD STRESS (DIAG. BRACING)	ksi (Mpa)	real	Enter the material yield stress for the bracing members attached diagonally to the main members of the beams.

DATA TYPE: 04012

Section Lookup

| Sec No | Std. Sect. | | | Tubular Shapes | | | | General Sections | | | | | | |
	ID	DTH inch (mm)	WT/FT (Lb./ft)	ID	OUT D inch (mm)	Tw inch (mm)	Kt	ID	WT/FT Kip/ft (KN/m)	S1 inch (mm)	S2 inch (mm)	S3 inch (mm)	S4 inch (mm)	S5 inch (mm)
1				2	254.	25.4								
2	W	4	13					9	.1897	103.1	105.6	8.636		7.112
3				2	127.	19.05								
4														
5														
6														
7														
8														
9														
10														
11														
12														
13														
14														
15														
16														
17														
18														
19														
20														

Click on SECTION LOOKUP to select a shape file first. Then use the scroll bar to view the selected file. Click on the desired data to import the section data.

Data Type 04012 – Definition of Sections

ITEM	UNIT	FORMAT	DESCRIPTION
STANDARD SECTION NO.	none	alphanumeric	Enter W for a wide flange section, or L for an angle. This data is for information purposes only and its input is optional
STANDARD SECTION DEPTH	inch (mm)	integer	Enter the nominal depth of the section. This data is for information purposes only and its input is optional.
STANDARD SECTION WEIGHT	lb/ft	integer	Enter the nominal weight of the section. This data is for information purposes only and its input is optional.
TUBULAR SHAPE I.D. NO.	none	integer	Enter one of the following cross section numbers: 2 – round cross section 3 – dodecagonal cross section 4 – octagonal cross section 5 – square cross section 999 – connection
TUBULAR OUTSIDE DIAM.	inch (mm)	real	Enter the outside diameter of the tubular cross sections.
TUBULAR WALL THICK$_{(TW)}$	inch (mm)	real	Enter the wall thickness of the tubular section.
TORSIONAL STRESS	none	real	Enter the torsional stress concentration factor for the particular cross section. This value may be inputted directly or calculated by the program upon leaving the [Sections] screen.
GENERAL SECT. I.D. NO.	none	integer	Enter one of the following cross section numbers: 6 – rectangular cross section (or cruciform) 7 – angle cross section 8 – channel (currently unavailable) 9 – W cross section 10 – Z cross section
GENERAL SECT. WEIGHT	kip/ft(KN/m)	real	Enter the weight per unit foot for the general section.
S1, S2, S3, S4, S5 PARAMETERS	inch (mm)	real	Enter the S1, S2, S3, S4, S5 parameters.

VAMS

VAM ID		Top VAM Coords.				* Supporting Members			Attached Sign Units			Walkway Units		
No	Sect.	X feet (m)	Y feet (m)	Z feet (m)	Length feet (m)	No.	No.	No.	No.	No.	No.	No.	No.	No.
1														
2														
3														
4														
5														
6														
7														
8														
9														
10														
11														
12														
13														
14														
15														
16														
17														
18														
19														
20														
21														
22														
23														
24														
25														

* Supporting Members are generated by Mesh.

Data Type 07012 – VAMS

ITEM	UNIT	FORMAT	DESCRIPTION
VAM SECTION I.D. NUMBER	none	integer	Enter the desired section number (from Screen "Definitions of Sections") for the vertical attachment member (VAM). Only general sections are allowed (angles, WF's, and Z's). A vertical attachment member is a member which attaches a sign to the sign structure.
TOP VAM X-COORDINATES	feet (m)	real	Enter the x-coordinate of the top of the vertical attachment member. The x-coordinate of the VAM cannot coincide with the x-coordinate of any joint, as defined in Screen "Definition of Joints".
TOP VAM Y-COORDINATES	feet (m)	real	Enter the y-coordinate of the top of the vertical attachment member.
TOP VAM Z-COORDINATES	feet (m)	real	Enter the z-coordinate of the top of the vertical attachment member.
LENGTH	feet (m)	real	Enter the length of the vertical attachment member.
SUPPORTING MEMBERS	none	integer	Enter the member numbers (from Screen "Definition of Members), which support each vertical attachment member. Supporting members are generated by Mesh.
ATTACHED SIGN UNITS	none	integer	Enter the sign numbers (from Screen "Sign Data"), which are supported by each vertical attachment member.
WALKWAY UNITS	none	integer	Enter the walkway numbers (from Screen "Walkways"), which are supported by each vertical attachment member.

A-11

	Sign No.	Dimensions					Lower Left Coord.			Drag Coeff.
		Width feet (m)	Height feet (m)	Thick. inch (mm)	Slope in/ft (mm/m)	Density k/cf (Kg/m^3)	X feet (m)	Y feet (m)	Z feet (m)	Optional Cd
	1					0.175				
	2									
	3									
	4									
	5									
	6									
	7									
	8									
	9									
	10									

Data Type 09012 – Signs

ITEM	UNIT	FORMAT	DESCRIPTION
SIGN WIDTH	feet (m)	real	Enter the width of the sign panel.
SIGN HEIGHT	feet (m)	real	Enter the height of the sign panel.
SIGN THICKNESS	inch (mm)	real	Enter the thickness of the sign panel.
SLOPE	in/ft (mm/m)	real	Enter the slope of the sign panel horizontal (parallel to the z-axis) per foot (meter) vertical (parallel to the y-axis).
DENSITY	k/cg (Kg/m^3)	real	Enter the density of the sign material. The value of 0.175 k/cf (2803 Kg/m^3) is the density for aluminum signs.
X-COORDINATE	feet (m)	real	Enter the x-coordinate of the lower left corner of the sign panel.
Y-COORDINATE	feet (m)	real	Enter the y-coordinate of the lower left corner of the sign panel.
Z-COORDINATE	feet (m)	real	Enter the z-coordinate of the lower left corner of the sign panel.
SIGN DRAG COEFFICIENT	none	real	Enter the overridden sign drag coefficient (SCD). (Internally set SCD for W/H $\leq$ 1, SCD = 1.12; 1 < W/H $\leq$ 2, SCD = 1.19; 2 < W/H $\leq$ 5, SCD = 1.2; 5 < W/H $\leq$ 10, SCD 1.23; 10 < W/H $\leq$ 15, SCD = 1.3)

A-12

A.3 Structure Lookup Input

Definition of Joints

DATA TYPE :03012

	JNT No	Coordinates			Supp. Translation			Supp. Rotation		
		X ft(m)	Y ft(m)	Z ft(m)	X	Y	Z	XX	YY	ZZ
✏	1	0.0	0.0	0.0	1	1	1	1	1	1
	2									
	3									
	4									
	5									
	6									
	7									
	8									
	9									
	10									
	11									
	12									
	13									
	14									
	15									
	16									
	17									
	18									
	19									
	20									

Data Type 03012 – Definition of Joints

ITEM	UNIT	FORMAT	DESCRIPTION
X-COORDINATES	feet (m)	real	Enter the x-coordinate of the point in space for the corresponding structure joint.
Y-COORDINATES	feet (m)	real	Enter the y-coordinate of the point in space for the corresponding structure joint.
Z-COORDINATES	feet (m)	real	Enter the z-coordinate of the point in space for the corresponding structure joint.
X-AXIS TRANSLATION	none	integer	Enter 0 if the joint is not supported against translation in the x direction; Enter 1 if the joint is supported against translation in the x direction.
Y-AXIS TRANSLATION	none	integer	Enter 0 if the joint is not supported against translation in the y direction; Enter 1 if the joint is supported against translation if the y direction.
Z-AXIS TRANSLATION	none	integer	Enter 0 if the joint is not supported against translation in the z direction; Enter 1 if the joint is supported against translation in the z direction.
X-X AXIS ROTATION	none	integer	Enter 0 if the joint is not support against rotation about the x-x axis ; Enter 1 if the joint is supported against rotation about the x-x axis.
Y-Y AXIS ROTATION	none	integer	Enter 0 if the joint is not supported against rotation about the y-y axis; Enter 1 if the joint is supported against rotation about the y-y axis.
Z-Z AXIS ROTATION	none	integer	Enter 0 if the joint is not supported against rotation about the z-z axis; Enter 1 if the joint is supported against rotation about the z-z axis.

Definition of Members

DATA TYPE : 05012

	Mem. No.	Sect. From	Sect. To	Joints From	Joints To	Angle Degree	Unit No.	Unit Type	Mat'l Yield ksi(MPa)	Mem. Type	Stress Cat
	1										
	2										
	3										
	4										
	5										
	6										
	7										
	8										
	9										
	10										
	11										
	12										
	13										
	14										
	15										
	16										
	17										
	18										
	19										
	20										

Excel Work Sheet

Note: If user specifies his own members please

1) Use one "prismatic" or "tapered" section within one unit.

2) Maximum unit number is 40. User may define multiple units within one unit type.

3) Only 4 unit types are allowed: (1) chord interior truss, (2) exterior truss - main chord members, (3) tower - main vertical members, (4) tower truss

4) If member types are blank, fixed-end members are assumed.

Data Type 05012 – Definition of Members

ITEM	UNIT	FORMAT	DESCRIPTION
SECTION, FROM	none	integer	If this cell is for a primary member (part of a unit) and the member is the first member in that unit (exists at the J-end of the unit) then enter the section number (from Screen "Definition of Sections") for the J-end of the unit. If this cell is for a secondary member (not part of a unit) then simply enter the section number for the J-end of the member.
SECTION, TO	none	integer	If this cell is for a primary member (part of a unit) and the member is the last member in that unit (exists at the K-end of the unit) then enter the section number (from Screen "Definition of Sections") for the K-end of the unit. All intermediate cells between the cell defining the unit J-end and the unit K-end must be left blank. The program will interpolate the section properties for the intermediate members. If this cell is for a secondary member (not part of a unit) then simply enter the section number from the K-end of the member.
JOINTS, FROM	none	integer	Enter the joint number (from Screen "Definition of Joints") of the J-end of the member
JOINTS, TO	none	integer	Enter the joint number (from Screen "Definition of Joints") of the K-end of the member
ANGLE	degree	real	Enter the angle defining the principal axis of the cross section. Refer to the figure below For tubular sections this angle may be ignored since any axis is a principal axis. For wide flange or other sections, the default angle is the strong axis along the Z axis (out of plane). For the calculation of the allowable, changing angle will affect the stress calculation but not the post KL/r computation where the program assumes the strong axis as r with the total height as L and K of 1.2 to calculate the allowable. This can be used for majority of the case since the weak axis is usually the direction of the tower's post(s). If not, user may use the optional screen to enter their own desired allowable.

A-14

UNIT NUMBER	none	integer	Enter the unit member (starting with 1) in which the member exists. A unit is a major continuous structural member made up of smaller members defined between joints. This cell should be left blank for secondary members. See Table 2.5 in the User Manual for definitions of primary and secondary members.
UNIT TYPE	none	integer	Enter 1, if interior truss members for boxed and trichords; Enter 2, if exterior truss member for boxed and trichords; Enter 3, if main vertical members (towers); Enter 4, if truss members for tower units.
METERIAL YIELD (Fy)	ksi (MPa)	real	Enter the yield stress for the member material. See Screen 8 for the yield stress default values.
MEMBER TYPE	none	integer	Enter the member identifier. Blank or "0" is assuming a fixed-end member. "14" is assuming a truss member. For other member types, see Table 2.13.
STRESS CAT	none	Alphanumeric	Fatigue stress category at the joint. For details, see Appendix A2.

Note: Members and joints are generated by Mesh. User may add, delete or alter members and joints on their respective screens after Mesh.
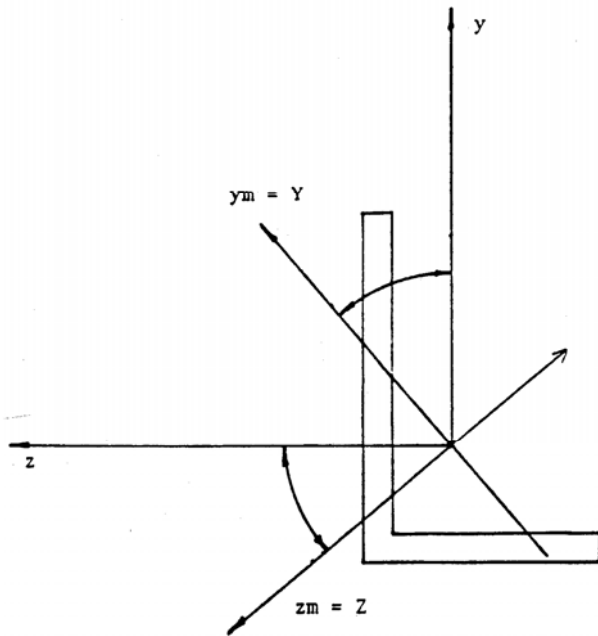
Definition of Principal Axis

A.4 Options Input

Data Type 01042 – Parameters

ITEM	UNIT	FORMAT	DESCRIPTION
WIND VELOCITY	mph (m/s)	real	Enter the annual extreme wind velocity at 30' (9.144 m) above the ground surface. The default value of 80 mph (128 Kmph) is based upon a 50-year recurrence interval for all overhead sign structures in Maryland.
MEAN REGULAR INTERVAL	years	integer	Enter the mean recurrence interval for wind velocity. This data is optional and is used only as a label.
GUST FACTOR	none	real	Enter the factor by which all wind levels will be multiplied to account for gust effects. Regular method – 1.14 (4th or 5th ed.), alternate method in Spec. Appendix C – 1.69 (5th ed. only). This information is optional.
ICE LOADS	psf (KPa)	real	Enter the value of the ice load which is to be applied to the surface of the structural supports and to one face of the sign panels. The default value is 3 psf (143.6 KPa).
ICE LOAD OPTIONS	none	integer	Enter 0 if the ice load is to be considered only on one side of the signs. Enter 1 if the ice load is to be considered on both sides of the signs.
TRUSS BRACING OPTIONS	none	integer	Default or "0" is Moment Bracing by assuming all braced members are fixed at both ends. Enter "1" for Truss Bracing by assuming truss action for all braced members.
DEAD LOAD DETAIL FACTOR	none	real	Enter the factor by which the structure dead load is multiplied to account for the weight of connections, bolts, etc.
WIND IMPORTANCE FACTOR	none	real	Default = 1.0, see AASHTO Table 3-2.
YEARLY MEAN VELOCITY FOR NATURAL WIND GUST	mps (m/s)	real	Enter the yearly mean velocity. The default value of 11.2 mph (5 m/s) is a reasonable upper-bound of yearly mean wind velocities for most locations in the country.
VEHICLE SPEED FOR TRUCK-INDUCED GUST	mps (m/s)	real	Enter the vehicle speed. The default value of 65 mph (30 m/s) wind to approximately coincide with existing vehicle speed limit.
FATIGUE IMPORTANCE FACTOR OPTIONS	none		AASHTO 4th Ed. I & II, 5th Ed. Cantilever I & II, Non-Cantilever, I & II.
IMPORTANCE FACTORS (GALLOPING, NATURAL WIND, TRUCK - INDUCED GUST)	none	real	Enter the value of the importance factors. Importance factors are introduced into the specifications to adjust the level of structural reliability of cantilevered support structures. Please see user manual Appendix B, Table B-1.1 (Table 11-1 of Ref. 1)

Additional Joint Loads

No	Joint No.	Load Types	Desc.	Forces X kip(KN)	Forces Y kip(KN)	Forces Z kip(KN)	Moments X-X k-ft (KN-m)	Moments Y-Y k-ft (KN-m)	Moments Z-Z k-ft (KN-m)
1									
2									
3									
4									
5									
6									
7									
8									
9									
10									
11									
12									
13									
14									
15									
16									
17									
18									
19									
20									

DATA TYPE : 10012

* For Load Types 3, 4 and 5, enter force range on their designated direction only.

Data Type 10012 – Additional Joint Loads

ITEM	UNIT	FORMAT	DESCRIPTION
JOINT NUMBER	none	integer	Enter the joint number (from Screen "Definition of Joints") where the load is to be applied
LOAD TYPE	none	integer	Type of added load (0 or blank = Dead load, 1 = Wind load, 2 = Ice load, 3 = Galloping – Y only, 4 = Natural wind gusts – Z only, 5 = Truck gusts – Y only. For load types 3, 4 and 5, enter force range on their designated direction only.
DESCRIPTION	none	alphanumeric	Enter a description of the load (e.g., ICE, SPEC1, etc.).
FORCES, X	kip (KN)	real	Enter concentrated force that is to be applied to the joint in the x-direction.
FORCES, Y	kip (KN)	real	Enter concentrated force that is to be applied to the joint in the y-direction.
FORCES, Z	kip (KN)	real	Enter concentrated force that is to be applied to the joint in the z-direction.
MOMENTS, X-X	k-ft (KN-m)	real	Enter the concentrated moment that is to be applied to the joint about the x-x axis.
MOMENTS, Y-Y	k-ft (KN-m)	real	Enter the concentrated moment that is to be applied to the joint about the y-y axis.
MOMENTS, Z-Z	k-ft (KN-m)	real	Enter the concentrated moment that is to be applied to the joint about the z-z axis.

Note: Dead, wind and ice loads will be generated by the program internally. These are additional loads other than default.

DATA TYPE: 08012

Walkway	X Coord. Ends		Load Type			Uniform Loading		Wind Areas	
No	Left feet(m)	Right feet(m)	DL	ICE	LL	LOAD klf (KN/m)	Z Coord. feet(m)	Area ft^2/ft (m^2/m)	Y Coord. feet(m)

Data Type 08012 – Walkways

ITEM	UNIT	FORMAT	DESCRIPTION
WALKWAY NO.	none	integer	Enter the walkway number for the load type under consideration. If more than one load type is to be applied to a particular walkway, each load type must be defined separately and on different lines.
X-COORDINATE END, LEFT	feet (m)	real	Enter the x-coordinate of the left end of the walkway.
X-COORDINATE END, RIGHT	feet (m)	real	Enter the x-coordinate of the right end of the walkway.
LOAD TYPE, DL	none	integer	Enter a [1] to indicate the presence of a uniform dead load on the walkway. If you enter a [1] in this column and also wish to apply an additional ice or live load to the walkway, you must enter the additional load on another line.
LOAD TYPE, ICE	none	integer	Enter a [1] to indicate the presence of a uniform ice load on the walkway. If you enter a [1] in this column and also wish to apply an additional dead or live load to the walkway, you must enter the additional load on another line.
LOAD TYPE, LL	none	integer	Enter a [1] to indicate the presence of a uniform live load on the walkway. If you enter a [1] in this column and also wish to apply an additional dead or ice load to the walkway, you must enter the additional load on another line.
UNIFORM LOAD	klf (KN/m)	real	Enter the intensity of the uniform load applied to the walkway.
Z-COORDINATE	feet (m)	real	Enter the z-coordinate of the location of the resultant of the uniform load.
WIND AREA	ft^2/ft (m^2/m)	real	Enter the area in the z-direction which defines the wind on the walkway.
Y-COORDINATE	feet (m)	real	Enter the y-coordinate of the centroid of the wind area.

CONNECTION   JOINTS

| 3 | 5 | 6 | 4 | 22 | 20 | 19 | 21 | | | | | | | |

* data is generated by Mesh

Data Type 01052 – Connections

ITEM	UNIT	FORMAT	DESCRIPTION
CONNECTION JOINTS	none	integer	Enter the number of each structure joint where a beam is connected to a tower. Normally, this information is created by the program when the joints and members are generated; therefore, the manual entry of the data is not required.

Data Type 09112 – Height Coeff.

ITEM	UNIT	FORMAT	DESCRIPTION
HEIGHT	feet (m)	integer	Enter the height (upper or lower limit) for the corresponding wind coefficient. The heights must be in ascending order from the bottom of the screen to the top of the screen.
COEFFICIENT (Ch)	none	real	Enter the height coefficient for the corresponding upper and lower heights.
HEIGHT COEFFICIENT	none		- 3rd edition - 4th/5th edition (default)

DATA TYPE : 40000

	LEFT BEAM END		RIGHT BEAM END	
	Y-Y	Z-Z	Y-Y	Z-Z
	(0 or 1)		(0 or 1)	
MONOTUBE		1		1
PLANE TRUSS				
TRICHORD TRUSS ON 1 POST				
BOX TRUSS ON 1 POST				
TRICHORD TRUSS ON 2 POSTS	1	1	1	1
BOX TRUSS ON 2 POSTS	1	1	1	1

Data Type 40000 – Hinges

ITEM	**UNIT**	**FORMAT**	**DESCRIPTION**
HINGES	none	integer	Enter [1] if a hinge (moment release) is desired about the structure y-y axis at the corresponding beam end or the structure z-z axis at the corresponding beam end. Enter [0] (or space) if a hinge is not desired. This data will be used to model bolts during mesh generation for span-type beams only. Cantilevered beams will not be affected.

Data Type 06012 – Member Option

ITEM	UNIT	FORMAT	DESCRIPTION
MEMBER NO.	none	integer	Enter member number (from screen "Definition of Members) where default values to be overridden.
WIND DATA SHIELD ID	none	integer	(Shield ID = 1 shielding on the –Z direction; = 2 shielding on the +Z direction; = 3 no shielding; = 4 both shielding)
WIND DATA WIND DRAG COEFFICIENT	none	real	Override the AASHTO specified values Cd (internally set by the shape of the member see AASHTO specification Table 3-6)
WIND DATA HEIGHT COEFICIENT	none	real	Override the AASHTO specification values Ch (as set on Screen "Height Coefficient")
AXIAL TENSION ALLOWABLE Ft	Ksi (MPa)	real	As per AASHTO Specifications (See Chapter 7)
AXIAL COMPRESSION ALLOWABLE Fa	Ksi (MPa)	real	As per AASHTO Specifications (See Chapter 7)
BENDING ALLOWABLE Fb	Ksi (MPa)	real	As per AASHTO Specifications (See Chapter 7)
SHEAR ALLOWABLE Fv	Ksi (MPa)	real	As per AASHTO Specifications (See Chapter 7)
EFFECTIVE LENGTH K FACTOR	none	real	The default factors: For single vertical post case, K = 2.0; For double vertical post with truss member connected, K = 1.2, Chords and truss members, K = 0.65

A.5 Post-Processing Input Screens

Splice Plate Design Parameters

ITEM	UNIT	FORMAT	DESCRIPTION
ENGLISH/S.I. UNIT	none	integer	Enter the English or SI unit used in this design.
BOLT YIELD STRESS	ksi (MPa)	real	Enter the yield stress for the splice plate anchor bolts.
PLATE YIELD STRESS	ksi (MPa)	real	Enter the yield stress for the splice plate.
CHORD YIELD STRESS	ksi (MPa)	real	Enter the yield stress for the chord.
CHORD OUTSIDE DIAM.	in (mm)	real	Enter the outside diameter of the chord at the level of the chord at the desired joint.
CHORD WALL THICKNESS	in (mm)	real	Enter the wall thickness of the chord at the desired joint.
CHORD SHAPE I.D. NO	none	integer	Enter one of the following tubular shape numbers: 2 – round cross section 3 – dodecagonal cross section 4 – octagonal cross section 5 – square cross section
JOINT FORCE, X DIRECTION	kips (KN)	real	Enter the shear reaction force at the desired joint. The x direction refers to the structure global axis.
JOINT FORCE, Y DIRECTION	kips (KN)	real	Enter the axial force at the desired joint. The y direction refers to the structure global axis.
JOINT FORCE, Z DIRECTION	kips (KN)	real	Enter the shear force at the desired joint. The z direction refers to the structure global axis.
JOINT MOMENT, X-X AXIS	k-ft (KN-m)	real	Enter the moment about the x-x axis at the desired joint. The x-x refers to the structure global axis.
JOINT MOMENT, Y-Y AXIS	k-ft (KN-m)	real	Enter the moment about the y-y axis at the desired joint. The y-y refers to the structure global axis.
JOINT MOMENT, Z-Z AXIS	k-ft (KN-m)	real	Enter the moment about the z-z axis at the desired joint. The z-z refers to the structure global axis.
ALLOWABLE WELD STRESS	kis (MPa)	real	Enter the allowable stress for the weld connecting the column to the splice plate to the chord.

GROUP LOAD NO.	none	integer	Enter the applicable group load number associated with the forces and moments entered previously. Refer to Table 3-1 in the AASHTO Standard Specs. for Structural Supports for Highway Signs, Luminaries and Traffic Signals.
SPLICE PLATE SHAPE	none	integer	Enter the shape number (as defined above for the chord shape)
DESIRED NO. OF BOLTS	none	integer	Enter number of bolts desired by the user. If nothing is input, program will design the number of bolts. Note: Number of bolts should be an even number.
JOINT NUMBER	none	integer	Enter the number of the joint on the beam where the splice will be located. A splice can be located only at a joint.
COEFFICIENT	none		Select 3rd Edition or 4th Edition-Default.

Base Plate Design Parameters

ITEM	UNIT	FORMAT	DESCRIPTION
ENGLISH/S.I. UNIT	none	integer	Enter the English or SI unit used in this design.
BOLT YIELD STRESS	ksi (MPa)	real	Enter the yield stress for the base plate anchor bolts.
BASE PLATE YIELD STRESS	ksi (MPa)	real	Enter the yield stress for the base plate.
COLUMN YIELD STRESS	ksi (MPa)	real	Enter the yield stress for the column
COLUMN OUTSIDE DIAM.	in (mm)	real	Enter the outside diameter of the column at the level of the base plate.
COLUMN WALL THICKNESS	in (mm)	real	Enter the wall thickness of the column at the level of the base plate.
COLUMN SHAPE I.D. NO	none	integer	Enter one of the following tubular shape numbers: 2 – round cross section 3 – dodecagonal cross section 4 – octagonal cross section 5 – square cross section
BASE FORCE, X DIRECTION	kips (KN)	real	Enter the shear reaction force at the column base. The x direction refers to the structure global axis.
BASE FORCE, Y DIRECTION	kips (KN)	real	Enter the axial force at the column base. The y direction refers to the structure global axis.
BASE FORCE, Z DIRECTION	kips (KN)	real	Enter the shear force at the column base. The z direction refers to the structure global axis.
BASE MOMENT, X-X AXIS	k-ft (KN-m)	real	Enter the moment about the x-x axis at the column base. The x-x refers to the structure global axis.
BASE MOMENT, Y-Y AXIS	k-ft (KN-m)	real	Enter the moment about the y-y axis at the column base. The y-y refers to the structure global axis.
BASE MOMENT, Z-Z AXIS	k-ft (KN-m)	real	Enter the moment about the z-z axis at the column base. The z-z refers to the structure global axis.
ALLOWABLE WELD STRESS	kis (MPa)	real	Enter the allowable stress for the weld connecting the column to the base plate.
GROUP LOAD NO.	none	integer	Enter the applicable group load number associated with

			the forces and moments entered previously. Refer to Table 1.2.6 in the AASHTO Standard Specs. for Structural Supports for Highway Signs, Luminaires and Traffic Signals.
BASE PLATE SHAPE	none	integer	Enter the shape number (as defined above for the column shape)
DESIRED NO. OF BOLTS	none	integer	Enter number of bolts desired by the user. If nothing is input, program will design the number of bolts. Note: Number of bolts should be an even number. Note: Minimum 6 bolts for cantilever structures. Minimum 4 bolts for overhead bridge structures.
COEFFICIENT	none		Select 3rd Edition or 4th Edition-Default.

Base Plate Fatigue Check

ITEM	UNIT	FORMAT	DESCRIPTION
ENGLISH/SI UNIT	none	integer	Enter the English or SI unit used in the design
BASE MOMENT RANGE X-X AXIS	K-ft (KN-m)	real	Enter the moment range about the x-x axis at the column base
BASE MOMENT RANGE Z-Z AXIS	K-ft (KN-m)	real	Enter the moment range about the x-x axis at the column base
COLUMN OUTSIDE DIAM.	in (mm)	real	Enter the outside diameter of the column at the level of the base plate.
COLUMN WALL THICKNESS	in (mm)	real	Enter the wall thickness of the column at the level of the base plate.
COLUMN SHAPE ID. NO.	none	integer	2 – round cross section 3 – dodecagonal cross section 4 – octagonal cross section 5 – square cross section
BASE PLATE SHAPE	none	integer	Enter the shape number (as defined above for the column shape)
BASE PLATE BOLT DIAM	in (mm)	real	Enter the bolt diameter for the base plate
BASE PLATE BOLT THREAD PITCH	in (mm)	real	Enter the bolt thread pitch
BOLT CIRCLE DIAMETER DISTANCE	in (mm)	real	Enter the diameter distance for the bolt circle
NO. OF BOLT	none	integer	Enter number of bolts desired by the user.
STIFFENER HEIGHT	in (mm)	real	Enter the height of the stiffener
STIFFENER WIDTH	in (mm)	real	Enter the width of the stiffener
STIFFENER THICKNESS	in (mm)	real	Enter the thickness of the stiffener
NO. OF STIFFENER	none	integer	Enter number of stiffeners desired by the user
COLUMN-TO-BASE PLATE CONNECTION	none	integer	1-Groove-Welded Tube-to-Transverse Plate Connection 2-Fillet-Welded Socket Connection

Appendix A2 – Presentation and Calculation of Fatigue Combine Stress Ratio

User has the option of overriding the predefine stress categories as defined in Table A-2.1 and demonstrated in Figure A-2.1 or enter their own stress categories on input screen for 'Definition of Members', Data Type 05012. (Input has to be after MESH command. Otherwise, input will be emptied.) Stress categories as defined in Table A-2.2 are allowed and their corresponding constant-amplitude fatigue limits are utilized by the program. Calculation of fatigue combined stress ratio (FCSR) is shown below:

1. Close section (2-round, 5-square,…sections)

$$\frac{fa + \sqrt{f_{bx}^2 + f_{by}^2}}{F_{sr}}$$

2. Open section (7-angle, 8-ST,…sections)

$$\frac{fa + f_{bx} + f_{by}}{F_{sr}}$$

Where F_{sr} = Allowable stress range based on tables above.

Table A-2.1- Predefined detail locations with their respective stress categories:

Type	Category	Detail No. and Description	AASHTO Examples
Post End	E'	16. Fillet-welded tube-to-transverse plate connections	7
Cantilever type – Post-to-chord connection	E'	16. Fillet-welded tube-to-transverse plate connections	8, 16
Span type - Post-to-chord connection	E'	16. Fillet-welded tube-to-transverse plate connections	8, 16
Truss-to-post or truss-to-chord connection	E	19. Fillet-welded T-,Y-, and K-tube-to-tube, angle-to-tube, or plate-to-tube connections.	10,11

Table A-2.2 – Constant-amplitude fatigue limits

	Steel	Aluminum
Stress Category	ksi	ksi
A	24	10.2
B	16	6
B′ (input as B1)	12	4.6
C	10	4
D	7	2.5
E	4.5	1.9
E' (input as E1)	2.6	1
ET	1.2	0.44
K	1	0.38

Stress category E - Truss-to-post or truss-to-chord connection

Stress category E' - Post-to-chord connections (both span- and Cantilever-types

Stress category E' - Post End
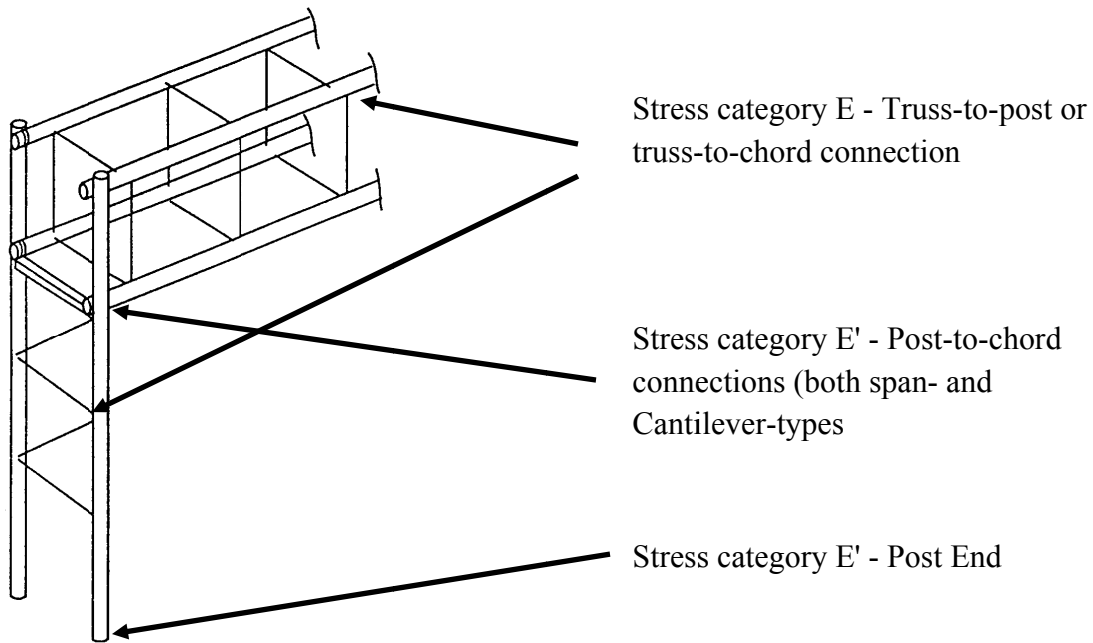
Figure A-2.1 - Predefined detail locations with their respective stress categories

Appendix B - Major changes for the 2009 AASHTO "Standard Specifications for Structural Supports for Highway Signs, Luminaries and Traffic Signals" 5th Edition.

Cantilevered and no cantilevered support structures shall be designed for fatigue to resist each of the applicable equivalent static wind load effects specified in Article 11.7 of Ref. 1, and modified by the appropriate importance factors given in Table 11.1 of Ref.1.

Table B-1.1 (Table 11-1 of Ref. 1)

Table 11–1—Fatigue Importance Factors, I_F

Fatigue Category			Importance Factor, I_F			
			Galloping	Vortex Shedding	Natural Wind Gusts	Truck-Induced Gusts
Cantilevered	I	Sign	1.0	x*	1.0	1.0
		Traffic Signal	1.0	x*	1.0	1.0
		Lighting	x	1.0	1.0	x
	II	Sign	0.70	x*	0.85	0.90
		Traffic Signal	0.65	x*	0.80	0.85
		Lighting	x	0.65	0.75	x
	III	Sign	0.40	x*	0.70	0.80
		Traffic Signal	0.30	x*	0.55	0.70
		Lighting	x	0.30	0.50	x
Noncantilevered	I	Sign	x	x*	1.0	1.0
		Traffic Signal	x	x*	1.0	1.0
	II	Sign	x	x*	0.85	0.90
		Traffic Signal	x	x*	0.80	0.85
	III	Sign	x	x*	0.70	0.80
		Traffic Signal	x	x*	0.55	0.70

Notes:

x Structure is not susceptible to this type of loading.

* Overhead cantilevered and noncantilevered sign and traffic signal components are susceptible to vortex shedding prior to placement of the signs and traffic signal heads, i.e., during construction.

The alternate method for wind pressures may be computed using the following formula:

$$P_z = 0.0473 \, (1.3V_m)^2 C_d C_h \quad \text{(Pa)} \qquad \text{(Eq. C-1 of Ref. 1)}$$

$$P_z = 0.00256 \, (1.3V_m)^2 C_d C_h \quad \text{(psf)}$$

Where:

P_z = Design wind pressure (Pa, psf)

V_m = Fastest-mile wind speed from map, for the design mean recurrence interval, see Figures C-1, C-2, and C-3 (Ref. 1) (km/h, mph)

Appendix C – Output Tables

TABLE C.1 – LIST OF OUTPUT TABLES			
TABLE NO.	OUTPUT PHASE	OUTPUT TABLE TITLE	OUTPUT LEVEL
		Input Verification	
1.1	Input. Verif.	Definition of Joints	1, 2
1.2	"	Definition of Sections	1, 2
1.3	"	Definition of Members	1, 2
1.4	"	Definition of Vertical Attachment Members	1, 2
1.5	"	Definition of Walkways and Conduits	1, 2
1.6	"	Definition of Signs	1, 2
1.7	"	Coefficients of Height	1,2
1.8	"	Joint Load Data	1, 2
		General Data	
2.1	"	General Member Data	1, 2
2.2	"	General Member Data	1, 2
		Dead Load Data	
3.1	Analysis	General Dead Load Information	1, 2
		Wind Load Data	
4.1	"	General Wind Load Information	1, 2
4.2	"	Group II Wind Load Joint Loads for Combination 1 in the +Z direction.	2
4.3	"	Group II Member-End-Actions for Wind Combination 1 in the +Z direction.	2
4.4	"	Group II Wind Load Joint Loads for Combination 2 in the +Z direction.	2
4.5	Analysis	Group II Member-End-Actions for Wind Combination 2 in the +Z direction.	2

TABLE C.2 – LIST OF OUTPUT TABLES			
TABLE NO.	OUTPUT PHASE	OUTPUT TABLE TITLE	OUTPUT LEVEL
4.6	"	**Group II Wind Load Joint Loads for Combination 1 in the –Z direction.**	2
4.7	"	Group II Member-End-Actions for Wind Combination 1 in the –Z direction.	2
4.8	"	Group II Wind Load Joint Loads for Combination 2 in the –Z direction.	2
4.9	"	Group II Member-End-Actions for Wind Combination 2 in the –Z direction.	2
4.10	"	Group III Wind Load Joint Loads for Combination 1 in the +Z direction.	2
4.11	"	Group III Member-End-Actions for Wind Combination 1 in the +Z direction.	2
4.12	"	Group III Wind Load Joint Loads for Combination 2 in the +Z direction.	2
4.13	"	Group III Member-End-Actions for Wind Combination 2 in the +Z direction.	2
4.14	"	Group III Wind Load Joint Loads for Combination 1 in the –Z direction.	2
4.15	"	Group III Member-End-Actions for Wind Combination 1 in the –Z Direction.	2
4.16	"	Group III Wind Load Joint Loads for Combination 2 in the –Z direction.	2
4.17	"	Group III Member-End-Actions for Wind Combination 2 in the –Z direction.	2
		<u>Ice Load Data</u>	
5.1	"	General Ice Load Information	1, 2
5.2	"	Ice Load Joint Loads	2
5.3	"	Member-End-Actions for Ice Loading	2
6.0	---	Not Used.	---

TABLE C.3 – LIST OF OUTPUT TABLES			
TABLE NO.	OUTPUT PHASE	OUTPUT TABLE TITLE	OUTPUT LEVEL
7.0	---	**Not Used.**	---
		<u>Group Joint Loads</u>	
8.1	Analysis	Joint Loads for Group I: DL	2
8.2	"	Joint Loads fro Group II: DL + W (Wind Combination 1 in the +Z direction)	2
8.3	"	Joint Loads for Group II: DL + W (Wind Combination 2 in the +Z direction)	2
8.4	"	Joint Loads for Group II: DL + W (Wind Combination 1 in the –Z direction)	2
8.5	"	Joint Loads for Group II: DL + W (Wind Combination 2 in the –Z direction)	2
8.6	"	Joint Loads for Group III: DL + ICE + .5W (Wind Combination 1 in the +Z direction)	2
8.7	"	Joint Loads for Group III: DL + ICE + .5W (Wind Combination 2 in the +Z direction)	2
8.8	"	Joint Loads for Group III: DL + ICE + .5W (Wind Combination 1 in the –Z direction)	2
8.9	"	Joint Loads for Group III: DL + ICE + .5W (Wind Combination 2 in the –Z direction)	2
		<u>Joint Displacements</u>	
9.1	"	**Joint Displacements for Group I:** DL	2
9.2	"	Joint Displacements for Group II: DL + W (Wind Combination 1 in the +Z direction)	2

TABLE C.4 – LIST OF OUTPUT TABLES			
TABLE NO.	OUTPUT PHASE	OUTPUT TABLE TITLE	OUTPUT LEVEL
9.3	Analysis	**Joint Displacements for Group II:** DL + W (Wind Combination 2 in the +Z direction)	2
9.4	"	**Joint Displacements for Group II:** DL + W (Wind Combination 1 in the –Z direction)	2
9.5	"	Joint Displacements for Group II: DL + W (Wind Combination 2 in the –Z direction)	2
9.6	"	Joint Displacements for Group III: DL + ICE + .5W (Wind Combination 1 in the +Z direction)	2
9.7	"	Joint Displacements for Group III: DL + ICE + .5W (Wind Combination 2 in the +Z direction)	2
9.8	"	Joint Displacements for Group III: DL + ICE + .5W (Wind Combination 1 in the –Z direction)	2
9.9	"	Joint Displacements for Group III: DL + ICE + .5W (Wind Combination 2 in the –Z direction)	2
9.10	"	Maximum Joint Displacements	1, 2
		Group Load Member-End-Actions	
10.1	"	Member-End-Actions for Group I: DL	2
10.2	"	Member-End-Actions for Group II: DL + W (Wind Combination 1 in the +Z direction)	2
10.3	"	**Member-End-Actions for Group II:** DL + W (Wind Combination 2 in the +Z direction)	2
10.4	"	Member-End-Actions for Group II: DL + W (Wind Combination 1 in the –Z direction)	2

TABLE C.5 – LIST OF OUTPUT TABLES
TABLE NO.	OUTPUT PHASE	OUTPUT TABLE TITLE	OUTPUT LEVEL
10.5	Analysis	**Member-End-Actions for Group II:** DL + W (Wind Combination 2 in the –Z direction)	2
10.6	"	**Member-End-Actions for Group III:** DL + ICE + .5W (Wind Combination 1 in the +Z direction)	2
10.7	"	Member-End-Actions for Group III: DL + ICE + .5W (Wind Combination 2 in the +Z direction)	2
10.8	"	Member-End-Actions for Group III: DL + ICE + .5W (Wind Combination 1 in the –Z direction)	2
10.9	"	Member-End-Actions for Group III: DL + ICE + .5W (Wind Combination 2 in the –Z direction)	2
		<u>**Tower Member Stress Information**</u>	
11.1	Code Check	Tower Member Details	1, 2
11.2	"	Tower Member Allowables at 100% (Group 1)	1, 2
11.3	"	Tower Member Allowables at 133% (Groups 2 + 3)	1, 2
11.4	"	Combined Stresses in Tower Members	1, 2
		<u>**Truss Member Stress Information**</u>	
12.1	"	**Truss Member Details**	1, 2
12.2	"	**Truss Member Allowables at 100% (Group 1)**	1, 2
12.3	"	Truss Member Allowables at 133% (Groups 2 + 3)	1, 2
12.4	"	Combined Stresses in Truss Members	1, 2

TABLE C.6 – LIST OF OUTPUT TABLES			
TABLE NO.	OUTPUT PHASE	OUTPUT TABLE TITLE	OUTPUT LEVEL
13.1	Code Check	**Support Reactions** **Support Reactions for Group I:** DL	1, 2
13.2	"	Support Reactions for Group II: DL + W (Wind Combination 1 in the +Z direction)	1, 2
13.3	"	Support Reactions for Group II: DL + W (Wind Combination 2 in the +Z direction)	1, 2
13.4	"	Support Reactions for Group II: DL + W (Wind Combination 1 in the –Z direction)	1, 2
13.5	"	Support Reactions for Group II: DL + W (Wind Combination 2 in the –Z direction)	1, 2
13.6	"	Support Reactions for Group III: DL + ICE + .5W (Wind Combination 1 in the +Z direction)	1, 2
13.7	"	Support Reactions for Group III: DL + ICE + .5W (Wind Combination 2 in the +Z direction)	1, 2
13.8	"	Support Reactions for Group III: DL + ICE + .5W (Wind Combination 1 in the –Z direction)	1, 2
13.9	"	Support Reactions for Group III: DL + ICE + .5W (Wind Combination 2 in the –Z direction)	1, 2

TABLE C.7 – LIST OF OUTPUT TABLES			
TABLE NO.	OUTPUT PHASE	OUTPUT TABLE TITLE	OUTPUT LEVEL
20.1	Code Check	Moment and Stress Range due to Galloping	1.2
20.2	"	Moment and Stress Range due to Natural Wind Gusts.	1.2
20.3	"	Moment and Stress Range due to Truck Gusts	1.2

Appendix D – Error and Warning Messages

TABLE D.1 – Error Message			
LEVEL	ERROR NO.	ERROR MASSAGE	REMEDY
Data Entry	104	No Library Loaded	Load a shape file from the LIBRARY selection of the pull-down menu.
"	106	Type of Shape File Not Recognized	All shape file must have either "ROU", "DOD", "OCT", "SQU" or "WF" in the file name.
"	108	Unable to Calculate Stress Factor! Data Missing.	Make sure that the diameter and thickness of the section being imported have positive values.
"	110	Decimal Point Required	Input a decimal point in the current cell.
"	115	File Name Required	Input a valid file name.
"	120	Not a Valid Sign Bridge File	Input either the name of a new data file or the name of an existing SIGN BRIDGE data file.
"	122	Not a Valid Sign Bridge Output File	Input the name of an existing SIGN BRIDGE output file saved under the current directory.
"	125	Can't Find Directory	Enter the path to an existing directory on a valid drive.
"	130	Can't Find Any Shape Files in LIBRARY.LST	The file names of all shape files must be entered in LIBRARY.LST.
"	134	VAM No. x Coincides with a Joint	Mover the X coordinate of VAM x so that it does not coincide with the X coordinate of a joint.

TABLE D.2 – ERROR MESSAGES			
LEVEL	ERROR NO.	ERROR MASSAGE	REMEDY
Mesh Generation	160	Error Generating Joints and/or Members	Review all input data for omissions or invalid data
"	162	Joints and Members Must be Generated First	Generate joints and members before proceeding
"	166	Number of Generated Joints > X	Reduce the number of tower and/or beam segments.
"	167	Number of Generated Member > X	Reduce the number of tower and/or beam segments.
"	168	Yield Stress Missing	Input yield stresses for all element types.
"	169	Element Number Missing	Input the number of beam chords and tower posts.
"	171	Segment Number Missing	Input the number of segments for each beam and tower.
"	172	Beam Segments Must Be an Even Number	Span-type beams must have an even number of segments to ensure symmetry about midspan.
"	173	Tower Segments Exceed 16	Reduce the number of tower segments.
"	174, 175	Beam Segments Exceed x	Reduce the number of beam segments.
"	176	Beam Length Cannot Equal 0	Input valid lengths for all beams
"	177	Connection Length Cannot Equal 0	Input valid lengths for all connections.
"	178	Tower Top Elevation Must Be Above Tower Bottom	Either increase the elevation of the top of the tower or decrease the elevation of the bottom.

TABLE D.3 – ERROR MESSAGES			
LEVEL	ERROR NO.	ERROR MASSAGE	REMEDY
Mesh Generation	179	Top Chord Elevation Cannot Be Below Tower Bottom	Increase the elevation of the top beam chord or decrease the elevation of the tower bottom.
"	180	Top Chord Elevation Cannot Be Above Tower Top	Increase the elevation of the top of the tower or decrease the elevation of the top beam chord.
"	181	Beam Width Must Exceed 0	Input a valid width for all box or trichords.
"	182	Tower Width Must Exceed 0	Input a valid width for all braced tower posts.
"	183	Beam Depth Must Exceed 0	Input a valid depth for all box or trichords.
"	184	Invalid Cross Section No.	Input a cross section that is defined on Screen "Definition of Sections".
"	152	Section Not Defined for Beam End Members	Enter the desired section numbers for the members comprising the bracing at the ends of the beams.

TABLE D.5 – ERROR MESSAGES			
LEVEL	ERROR NO.	ERROR MASSAGE	REMEDY
Analysis	F1	Input Data Sequence at Card No. X	Check sequence of card number given.
"	F2	Total of x is Inconsistent with Joint Number x	Check joint number given and joint input sequence.
"	F3	Total of x is Inconsistent with Section Number x	Check section number given and section input sequence.
"	F4	Total of x is Inconsistent with Member Number x	Check member number given and member input sequence
"	F6	Total of x is Inconsistent with VAM Number x	Check VAM number given and VAM input sequence.
"	F7	Total of x is Inconsistent with Walkway Number x	Check walkway number given and walkway input sequence.
"	F8	Total of x is Inconsistent with Sign Number x	Check sign number given and sign input sequence.
"	F9	Maximum Number of Units Exceeded (maximum = 40)	Check member input, number of units <= 40.
"	F10	Structure Type Error	Check configuration no. (1-5 only).
"	F11	Improper Section for Tower Member x	Check member number x tubular sections only for primary vertical members.
"	F12	Orientation Error for Unit Number x	Check orientation for unit number given. Member types 1 and 4 cannot be part of a unit.

TABLE D.6 – ERROR MESSAGES			
LEVEL	ERROR NO.	ERROR MASSAGE	REMEDY
Analysis	F13	Initial Data Errors of Maximum Height Zone	Check structure input height > 55.
"	F14	Procedure Fails	Check for proper number of support restraints at proper locations.
"	F15	VAM Type Number x Not Defined, VAM Number x	Check section of VAM number given.
"	F16	Section Number x is not Defined for VAM Number x	Check section number for VAM number given cannot be 1-6 (tubular).
"	F18	The Supporting Member For VAM Number x is Not Defined	Check input support member for VAM number given.

References

1. "Standard Specification for Structural Supports for Highway Signs, Luminaires, and Traffic Signals"; American Association of State Highway and Transportation Officials; Washington, D.C.; 1994, 2001, and 2009.

2. Gere, J. M., Weaver, W. Jr.; "Analysis of Framed Structures"; D. Van Nostrand Co., New York; 1980

3. "Microsoft Windows Family of Operating Systems"; Microsoft Corporation; 1987-2009

4. S. Timenshenko; "Strength of Materials" – Part II; D. Van Nostrand Co. New York; 1957

5. Bathe, K. L., Wilson, E.L.; "Numerical Methods in Finite Element Analysis"; Prentice-Hall, Inc., New Jersey; 1976

6. "Building Code Requirements for Reinforced Concrete"; American Concrete Institute; Detroit; 1999

7. Blodgett, O. W.; "Design of Welded Structures"; Lincoln Arc Welding Foundation; Cleveland; 1968

8. "ASD Manual of Steel Construction", 9th Edition; American Institute of Steel Construction; Chicago; 1980

9. "NCHR Report 469-Fatigue-Resistant Design of Cantilevered Signal, Sign and Light Supports," Transportation Research Board, Washington, D.C.; 2002