Technical Report Documentation Page

1. Report No. FHWA/TX-92+360-1	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle COMPUTER PROGRAM FOR THE ANALYSIS OF CURVED STEEL GIRDER BRIDGES		5. Report Date June 1991 6. Performing Organization Code
7. Author(s) Kristopher H. Hahn and C. Philip Johnson		8. Performing Organization Report No. Research Report 360-1
9. Performing Organization Name and Address Center for Transportation Research The University of Texas at Austin Austin, Texas 78712-1075		10. Work Unit No. (TRAIS) 11. Contract or Grant No. Research Study 3-5-85-360
12. Sponsoring Agency Name and Address Texas Department of Transportation (formerly State Department of Highways and Public Transportation) P. O. Box 5051 Austin, Texas 78763-5051		13. Type of Report and Period Covered Interim 14. Sponsoring Agency Code

15. Supplementary Notes

Study conducted in cooperation with the U. S. Department of Transportation, Federal Highway Administration

Research Study Title: "Analysis of Curved Steel Girder Units"

16. Abstract

The Kurv87 computer program was developed to provide an easy-to-use analysis tool for curved as well as straight steel girder bridges. The philosophy behind this objective was to set certain limitations to balance the goals of an easy-to-use program and one which is applicable to many bridge geometries. The program easily handles the following types of bridge problems: (1) erection procedure in which multiple stages and differing boundary conditions can be superimposed on each other; (2) S-curved bridges with intermittent straight segments; (3) support settlements and specified support stiffnesses; (4) multiple truck or lane loadings; (5) slight exterior support skew and severe or slight interior support skew; (6) dead loads superimposed on live loads; (7) orthotropic slab properties over negative moment regions; and (8) three diaphragm configurations with or without bottom lateral bracing.

The program also provides the capacity to add segments of the bridge, which do not correspond to the data generator requirements, between data generated ones so that most any bridge can be analyzed on varying degrees of input difficulty.

A parameter study was done to study the accuracy of the V-load method and to study curved girder behavior.

17. Key Words computer program, analysis, curved steel girder bridges, straight steel girder bridges, erection procedure, loads, stiffnesses, skew, diaphragms	18. Distribution Statement No restrictions. This document is available to the public through the National Technical Information Service, Springfield, Virginia 22161.

19. Security Classif. (of this report)	20. Security Classif. (of this page)	21. No. of Pages	22. Price
Unclassified	Unclassified	178	

Form DOT F 1700.7 (8-72) Reproduction of completed page authorized

COMPUTER PROGRAM FOR THE ANALYSIS OF CURVED STEEL GIRDER BRIDGES

by

Kristopher H. Hahn
C. Philip Johnson

Research Report Number 360-1

Analysis of Curved Steel Girder Units

Research Project 3-5-85-360

conducted for

Texas
State Department of Highways and Public Transportation

in cooperation with the
U. S. Department of Transportation
Federal Highway Administration

by the

CENTER FOR TRANSPORTATION RESEARCH
BUREAU OF ENGINEERING RESEARCH
THE UNIVERSITY OF TEXAS AT AUSTIN

June 1991

NOT INTENDED FOR CONSTRUCTION,
PERMIT, OR BIDDING PURPOSES

C. P. Johnson (Texas No. 49162)

Research Supervisor

ABSTRACT

The Kurv87 computer program was developed to provide an easy to use analysis tool for curved as well as straight steel girder bridges. The philosophy behind this objective was to set certain limitations to balance the goals of an easy to use program and one which is applicable to many bridge geometries. The program easily handles the following types of bridge problems :

1) Erection procedure in which multiple stages and differing boundary conditions can be superimposed on each other.
2) S curved bridges with intermittent straight segments.
3) Support settlements and specified support stiffnesses..
4) Multiple truck or lane loadings.
5) Slight exterior support skew and severe or slight interior support skew.
6) Dead loads superimposed on live loads.
7) Orthotropic slab properties over negative moment regions.
8) Three diaphragm configurations with or without bottom lateral bracing.

The program also provides the capacity to add segments of the bridge, which do not correspond to the data generator requirements, between data generated ones so that most any bridge can be analyzed on varying degrees of input difficulty.

A parameter study was done to study the accuracy of the V-load method and to study curved girder behavior. The examination of some of the results revealed:

1) When the radius decreases, the load transfer from the inner to the outer girder increases.
2) The diaphragm spacing had a profound effect on the warping stresses but much less of an influence on the bending stresses.
3) The V-load method was very good for predicting dead load stresses on noncomposite sections but it was not nearly as good at predicting live load stresses on composite sections since it ignored the stiffness of the slab.
4) The effect of the bottom lateral bracing was not seen to be too significant in the bridge studied even though a little less rotation was evident. The influence of the braces probably would be more evident for a bridge with a sharper degree of curvature.

TABLE OF CONTENTS

CHAPTER 1. INTRODUCTION

1.1 BACKGROUND

Expanding existing roads or interchanges presents the designer with many problems. First, there is generally much less right of way for a new bridge. Second, the two places that need to be connected are probably not alligned and must be connected with a curved bridge. Also, the new roadway probably passes over an existing road which puts great constraints on the additional road substructures. For these reasons, the choice of curved girders is the only feasible solution to a complex problem.

Curved girders offer several advantages over straight girders. Curved girders allow the use of longer spans which eliminate some of the substructure. This allows the designer to meet space restrictions below existing roadways. Curved girders also can be designed as continuous composite girders which results in a stiffer structure, fewer expansion details, and greater vertical clearance because of shallower girders. Curved girders are also aesthetically more pleasing than a series of straight girders along the chord of a roadway. Since the transverse spacing of the girders is constant relative to the edges of the road, more uniform and simpler details are possible for curved girder bridges.

The designer should be aware of disadvantages in using curved girders. Curved girder fabrication costs are generally higher than those for a straight girder. Depending on the curvature, the curved girder segments must be transported in smaller pieces which increases both the shipping and erection costs. Analyzing curved girders during the erection stages and the service life is also more complicated than for straight bridges.

Curved girder analysis must generally be done on the computer. There exists approximate methods such as the V-Load method to determine a rough design of the bridge. These approximate methods use many simplifications and are usually good only for a narrow range of bridges. The computer must be used to check the final design but the preparation of the input data for a more general program may be very tedious and time-consuming. There is a need, therefore, for an analysis tool in which designs for a wide variety of bridges can easily be modified while accuracy is maintained, and yet be relatively easy to use.

1.1.1 Curved Girder Behavior

The analysis of curved girders is more complex than that of straight ones. A simplified explanation of some of the differences in behavior follows. This explanation generally follows the reasoning behind the V-load method.

A single curved girder has forces acting on it which causes torsional loading on the girder as shown in Figure 1.1a. Assuming that the flanges resist the full moment, the longitudinal force in the flange at any point is equal to the vertical bending moment on a transverse section "M" in the girder at that point divided by the centerline distance "h" between flanges. Because of the girder curvature, these axial forces are not collinear along any given segment of the flange. Thus, to maintain equilibrium, radial forces must be developed along the girder as shown in Figure 1.1b. The radial component of the flange force is directed outward when the flange is in compression and inward when the flange is in tension. The radial forces are in opposite directions as shown in Figure 1.2. It is the moment of these radial forces times the depth "h" which causes the twisting of the girder about its longitudinal axis. Thus, the radial forces cause lateral bending of the girder flanges and this results in the development of warping stresses.

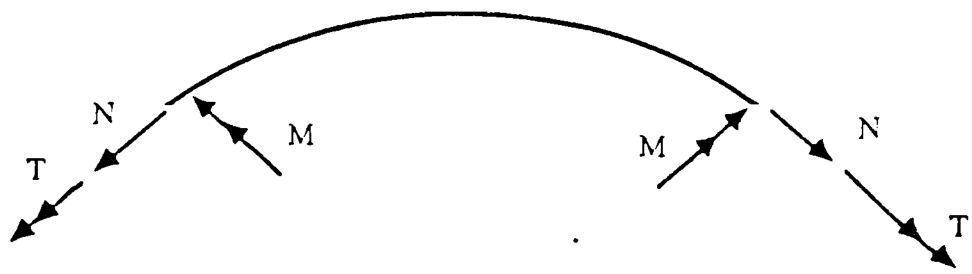


Figure 1.1a Forces on a curved element

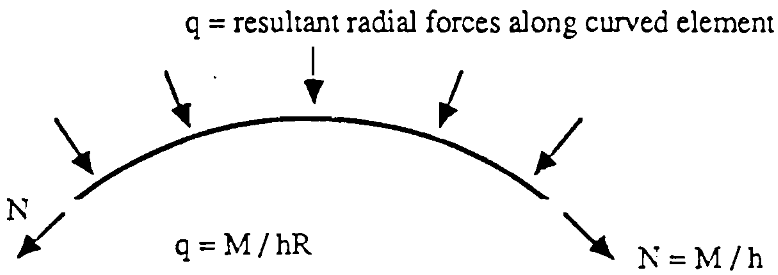


Figure 1.1b Radial forces developed on a curved element.

In a curved girder, the bending and torsional moments are coupled; physically, this means that the bending moments are influenced by the torsional moments along a girder and vice versa as shown in Figure 1.1a. A free body cross-section is shown in Figure 1.2. The horizontal forces are the previously described radial forces. The in-plane and torsional moments vary along the girder length and since they are interrelated, they cannot be solved directly. Unless the radius of curvature is sharp, the effect of the radial forces is much higher than the effect of the torsional moment.

The addition of cross-frames to multi-girder systems greatly alter the physical behavior of the bridge. A simple example of a two girder system is shown in Figure 1.3. and will help in the explanation. The diaphragms introduce concentrated torsional moments which are a function of the diaphragm and girder stiffness. In this simplified explanation, the diaphragms act as a concentrated reaction point for the radial forces developed because of the curvature as shown in Figure 1.4. The approximate value would be "q" times the length between diaphragms. Equal and opposite forces are thus developed at each cross-frame. To maintain equilibrium of the cross-frame, vertical shear forces must be developed at each end of the cross-frame as a result of the cross-frame rigidity. These shear forces then react on the girders and either increase or decrease the load on the girder as shown in Figure 1.5. The net effect of a braced, curved girder system is that a portion of the load is generally shifted from the inner to the outer girders.

The V-Load method uses this simplification to analyze curved bridges. First the girders are straightened out and the ordinary bending moments are determined at diaphragm locations. The second step is to apply fictional V-Loads to each girder to account for the shift in load from the inner and outer girders.

It is useful to discuss in a general way the effect of certain bridge parameters on the behavior of a curved bridge. As might be assumed, the shift of the load from the inner to outer girders increases as the radius decreases. The shift of the load from the inner to outer girder decreases as the bridge becomes torsionally stiffer. The things that increase the torsional stiffness of the bridge the most are the concrete slab, the diaphragm spacing and the diaphragm configuration. The concrete slab is very stiff compared to the steel framing and keeps the load transfer low. The main function of the diaphragm is to provide stiffness with a closed section during construction. Thus, closer spacing and proper configuration of the braces is needed during the erection stage. After the slab is placed, the diaphragms are still primary members but fewer are needed since the slab stiffness is so large.

Perhaps the most critical time of a curved bridge is the construction phase. During the construction of curved girders, the geometry and boundary conditions of the structure are changed at various stages of erection changing from girder units with cross-frames to a complete bridge with a composite deck. Due to the geometry of curved girders, they must generally be shipped in smaller lengths. This increases the amount of field erection. Generally, the girders are erected in pairs and many times this involves cantilevering pieces. The deflections and stresses at this stage must be determined since it may be the most critical time for the bridge. After subsequent girders and diaphragms are added, the bridge becomes much stiffer and less critical. Once the slab is provided, the behavior of the bridge is better since the slab provides a lot of stiffness. As a designer, it would be useful to try different erection schemes to determine the best construction process. In this way, the need for falsework or alternate erection schemes can be determined. What the designer actually needs is an easy way to analyze the construction process as well as the service life of the bridge.

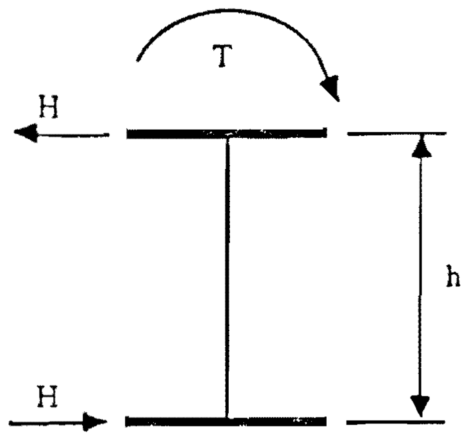


Figure 1.2 Forces produced on a single curved girder

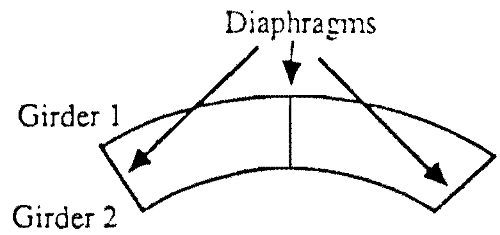


Figure 1.3
Diaphragms for a curved bridge

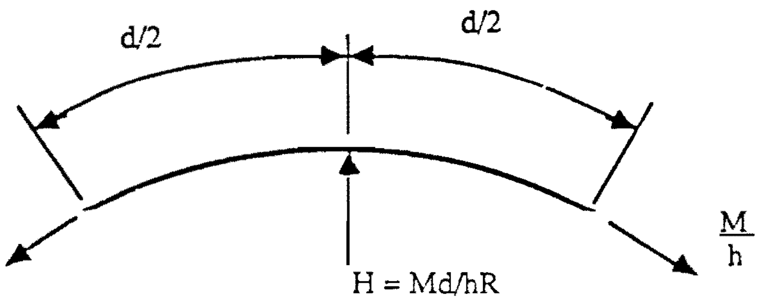


Figure 1.4 Diaphragm reaction on a curved flange

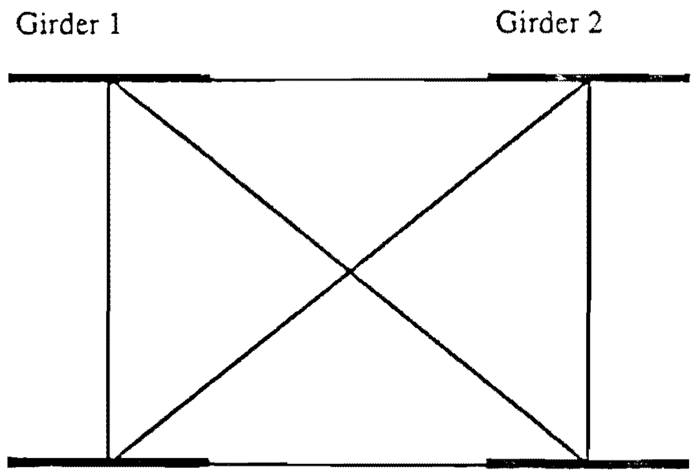


Figure 1.5a Diaphragm cross-section

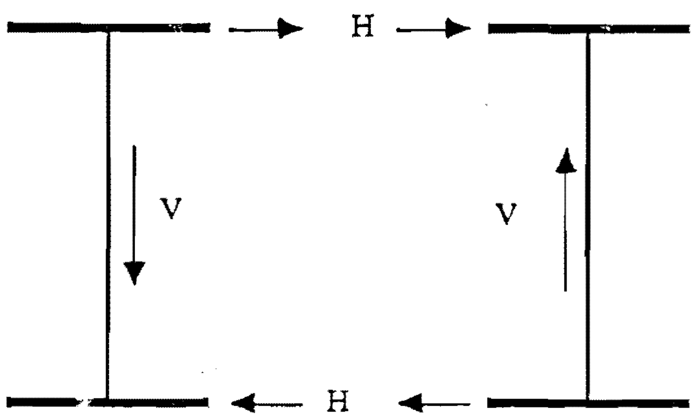


Figure 1.5b Additional loads due to girder curvature

1.1.2 Current Curved Girder Analysis Procedures

There are many different methods to analyze curved girders. Most of these methods involve the computer since the analysis of even a simple curved girder by hand gets extremely complicated. In general, these methods are categorized as follows:

1. Static Method - U.S. Steel (4)*
2. Computer Matrix Grid Method (8,9)
    a) Three DOF
    b) Five DOF
3. Space Frame Grid Method (8)
4. Finite Element Method (8)
5. Finite Difference Method (10,11,12,13)

* Number in parenthesis denotes items in References.

The first method is often called the "V-Load" method and generally follows the simplified explanation for curved girder behavior in section 1.1.1 . The V-load method allows for the direct determination of the forces in the longitudinal girders and diaphragms by making certain assumptions relative to load or force distribution. The V-load procedure is simple to use and is easily adapted to a personal computer for more complicated cases. Recently, finite element analyses have been done to verifiy the V-Load method for a wider range of bridges. (4) Many practicing engineers use the V-Load method for approximate designs because of its ease in use and then make final design checks with computer models.

The second method to be utilized consists of a computer oriented matrix method, which may consist of three degrees of freedom or five degrees of freedom (3). The three DOF grid accounts for torsional effects, but exclude warping. The five DOF gird , contains warping in the top and bottom flanges, thus increasing the degrees of freedom by two.

The third method to be utilized is the space frame gird method with six DOF at each node (3). The method models the girder flanges as beams and the webs as a series of cross and vertical elements. The modeling of the girders in this manner include warping influences by summing the effects of axial loads and vertical bending moments on the flanges.

The finite element method idealizes the bridge as a series of plate and beam elements with six DOF at each node. Many times the preparation of the data and solution time can be excessive for large problems even though the answers obtained are quite good.

The finite difference procedure permits direct solution of the differential curved girder equation. The solution of the differential equations can also be solved by a rigorous closed-form technique or a Fourier series method. Each of these three methods to solve the differential equation has its advantages and disadvantages depending on the boundary conditions, number of girders and loadings. In all these methods, the solution of the basic Vlasov (7) curved girder differential equations are utilized. Design aids and computer programs have been developed for the finite difference method so that it is much easier to use.

The ideal analysis tool should be easy to use, accurate for a wide range of possibilities and be relatively inexpensive to use. This research program was actually done in two separate parts by combining the first and fourth methods.

1.2 OUTLINE OF PRESENT RESEARCH

The research into the development of a curved girder analysis tool has been done in two parts. The first part developed a computer program for the V - Load method in which an approximate design could be determined. The second part, which is the basis of this thesis, involved adapting an exitsting finite element program to analyze curved steel girder bridges.

1.2.1 Objectives and Purpose

The objective of this research was to develop an easy to use analysis tool for curved as well as straight steel bridges. In addition, this research was to develop an easy to use analysis procedure of the erection process. The philosophy behind this objective was to set certain limitations to balance the goals of an easy to use program and one which is applicable to many bridge geometries.

The purpose of the research was to provide the designer with a new computer program design tool. This new design tool will allow the structural design and analysis of curved girder bridges to be more efficient and accurate than presently possible. The benefit will be a reduction in engingeering time and prevention of unforeseen construction problems.

1.2.2 Analysis Procedure

The main thrust of this research was to develop a pre-processor and a post-processor for existing finite element programs. The programs that were already developed were GENPUZ, PUZF83 and RECPUZ. The programs utilize one and two dimensional finite elements in a three-dimensional global assemblage with six degrees of freedom (DOF) at each nodal point. The two-dimensional elements have been used extensively and have been reported in previous CFHR reports from projects 23 and 155.

The novel feature of the computer program developed in this project is that of substructuring. Using the substructuring approach, bridge sections with identical properties that appear in sequence can be treated very effectively since only the first section in the sequence has to be dealt with. The computer program recognizes repeated substructures in both the stiffness calculations and the stress recovery. This, of course, results in savings of both human and computer resources.

The goal of this project was to tailor simplified inputs to this concept of substructuring for curved steel girder bridges. The individual substructure geometries and loadings are specified for the GENPUZ program. GENPUZ is executed prior to PUZF83. GENPUZ prepares disk files containing element stiffnesses and node numbers which are used in PUZF83. Information on the global assemblage of the substructures as well as boundary conditions are also specified for PUZF83. PUZF83 is a single level substructuring package which computes nodal displacements. RECPUZ is done last and it is the stress recovery program. The program was written in FORTRAN on the Cyber 750/170 of The University of Texas at Austin computational facilities. The computer program was subsequently adapted to the SDHPT computer facilities.

1.3 OUTLINE OF CHAPTERS

Chapter 2 describes the background for the computer program and the documentation for the program. Figures and small examples are also used to help the user understand the program quicker and easier.

Chapter 3 describes the output of the program. The user is given the background on the output and help in discovering errors in data preparation.

Chapter 4 describes six example problems that demonstrate the ease and versatility of the program.

Chapter 5 describes the comparison of this program to other methods of analysis along with a parameter study for different curved bridge geometries. The main purpose of this chapter is to demonstrate that the program has been verified by other programs so that the user may be confident in the solutions obtained. Also, the parameter investigation was used to study curved bridge behavior and to check both the finite element and V-load programs.

Chapter 6 presents conclusions and recommendations for other computer analysis programs.

CHAPTER 2. COMPUTER PROGRAM

2.1 GENERAL CONSIDERATIONS

The analysis of the bridge uses one- and two-dimensional finite elements in a three-dimensional grid. The use of the program requires a respectable level of competence in finite element analysis on the part of the user. The data generator takes simplified input and creates finite element meshes, loads them and determines the deflection and stresses. The user must decide how to best model the bridge so that these results are meaningful. Toward this end, this chapter describes how the data generator takes the simplified input and models the bridge. The sections that follow also discuss limitations and assumptions of the program with helpful suggestions on arranging the input data.

2.2 BACKGROUND

To perform an analysis, the bridge must be divided into substructures. A substructure can be thought of as a building block for a part of the structure. In this way, a wide variety of structural combinations are possible through different arrangements of these blocks. An example of this concept of substructuring might clarify things. Assume there is an S-shaped bridge as shown in Figure 2.1. If there was one substructure curved to the right, one straight and one substructure curved to the left then, by putting these substructures end to end, a reverse curvature structure can be built.

The choice of partitoning is arbitrary and up to the user. In this data generator a substructure is defined as part of the bridge between two diaphragms. These substructures can be repeated consecutively with the input being required for the first substructure only. When doing this the same node numbers can be assigned to the various substructures. In this case, nodes common to two substructures must be renumbered and these global nodes are termed Master Nodes. Further description is given in section 2.4.6.

In order to develop a data generator , the following assumptions were made:

1) There are a constant number of nodes for cross-sections which border on the edge of data generated substructures.
2) There are two or more girders.
3) The depth of the webs for all girders are the same.
4) Transverse diaphragms are connected at the intersection of the x-bracing.
5) Overhang elements of the slab span from the edge of the slab to the centerline of the girder.
6) There is no superelevation.
7) Transverse diaphragms are in every bay between girders.
8) There is a constant spacing between any two girders along the length of the members (no fanning out).
9) All supports are at the bottom of the flanges.
10) Each girder is vertically restrained at all supports and horizontally restrained at one support.
11) Curves are approximated by finite elements with straight edges.
12) In general, flange dimensions and material properties are assumed to change at diaphragm locations. This is the probably the most severe restriction of the data generator but it was done so that each substructure has one set of flange sizes and material properties. To make changes between

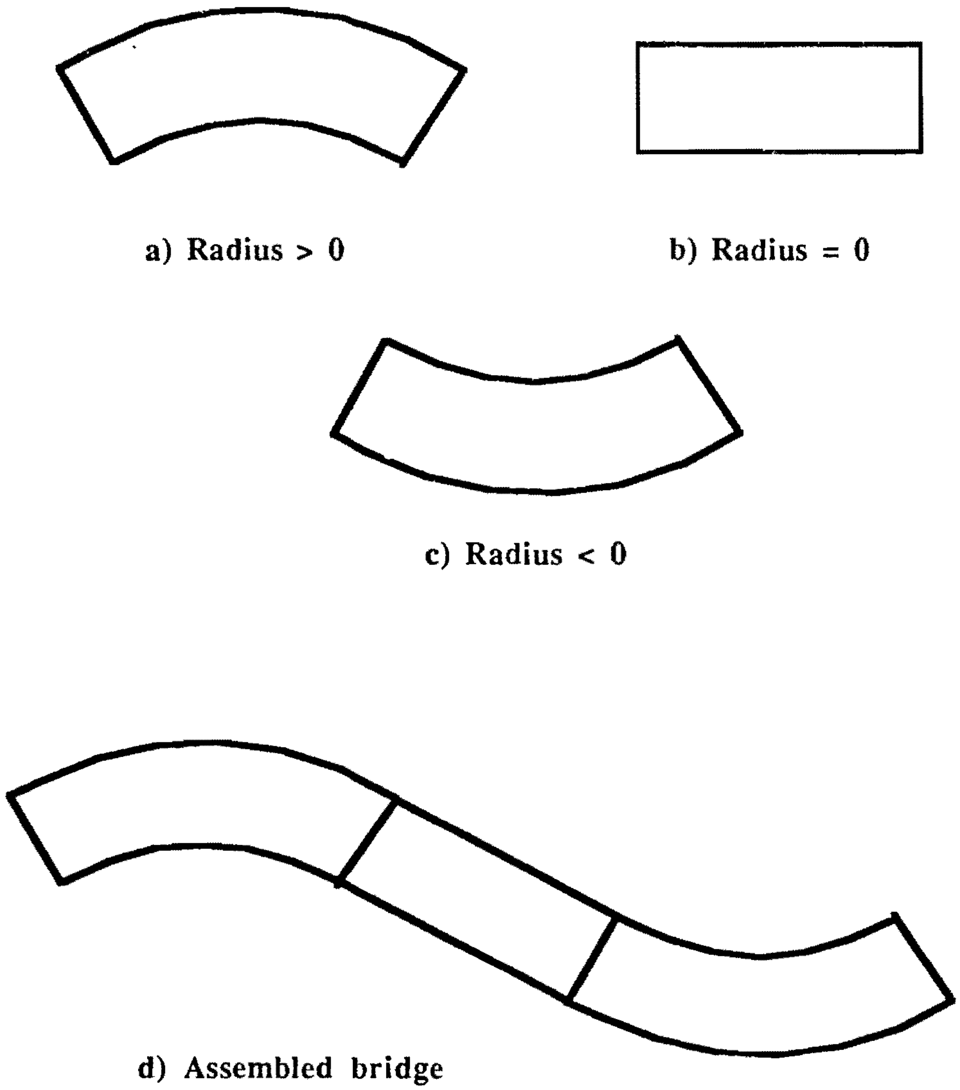


Figure 2.1 Three substructures with different radii

diaphragms, a substructure with diaphragms with zero physical properties can be added between the actual diaphragms. Also, the diaphragm spacing can be altered so that it coincides with the change in properties.

2.3 DATA GENERATOR

The bridge is modeled with one- and two-dimensional finite elements in a three-dimensional grid. A typical partial model is shown in Figure 2.2. The concrete slab is modeled as a two-dimensional plate element and is connected to the beam by rigid links. The rigid link is a beam element and goes from the center of the top flange to the center of the concrete slab. The top flange is modeled as a beam element which is connected to the web and the rigid link. The steel web spans between the top and bottom flange and is modeled as a plate element. The bottom flange is also modeled as a beam element and is connected to the web. The diaphragms are modeled as beam elements and are connected to the top and bottom flanges. This modeling is common and results have been quite good (6).

Two-dimensional plate elements are used for the concrete slab and the steel web. These plate elements can be either quadrilateral or triangular elements. A typical quadrilateral plate element is shown in Figure 2.3b. The x-axis bisects side il and jk with the direction going from node i to node j. The y-axis is placed perpendicular to the x-axis with the direction going from node j to node k. The z-axis completes a right hand coordinate system and projects upward from the top of the element as shown in Figure 2.3b. A typical triangular plate element is shown in Figure 2.3a. The x-axis is along line ij going from node i to node j. The y-axis and z-axis are then placed in a right hand coordinate system with the z-axis pointing upward. The plate elements were originally developed as shell elements with both membrane and bending stiffnesses. The plate elements have been used and described extensivly in previous CFHR reports from projects 23 and 155. A constant thickness and orthotropic properties have been assumed for the plate element. A plate element with orthotropic properties is especially convenient for the concrete slab in the negative moment region and for the wet slab load in the dead load case since the elastic modulii can be set to different values in the transverse and the longitudinal directions.

One-dimensional beam elements are used for the diaphragms, flanges and rigid links. A typical beam element is shown in Figure 2.3c. The x-axis goes from node i to node j. The y-axis is in the horizontal plane and the z-axis is in the vertical plane for horizontal beams. These beam elements have axial, bending and torsional stiffnesses. It may be assumed that the diaphragm is an axial element in a truss, in which case, the bending and the torsional stiffnesses can be set equal to zero. The flange elements are assumed to be rectangular and the bending and torsional stiffnesses are internally calclulated from the flange dimensions. The properties of the rigid link are internally set in the program and have been set so that it would act as a shear stud and undergo little, if any, deformations.

2.3.1 Nodal Coordinates

From the input for one substructure, the data generator determines the nodal coordinates, the element node numbers and element properties required by the analysis program. As stated earlier, the substructure is assumed to be bounded by diaphragms on each end. The data generator numbers the nodes of the substructure from left to right. The local axis is placed on the left edge of the substructure at the same level as the bottom flange. If the substructure is straight, the axis is placed at the midpoint between the first and last girder. If the substructure is curved then cylindrical coordinates are used and the axis is located at a the point of origin of the reference line.

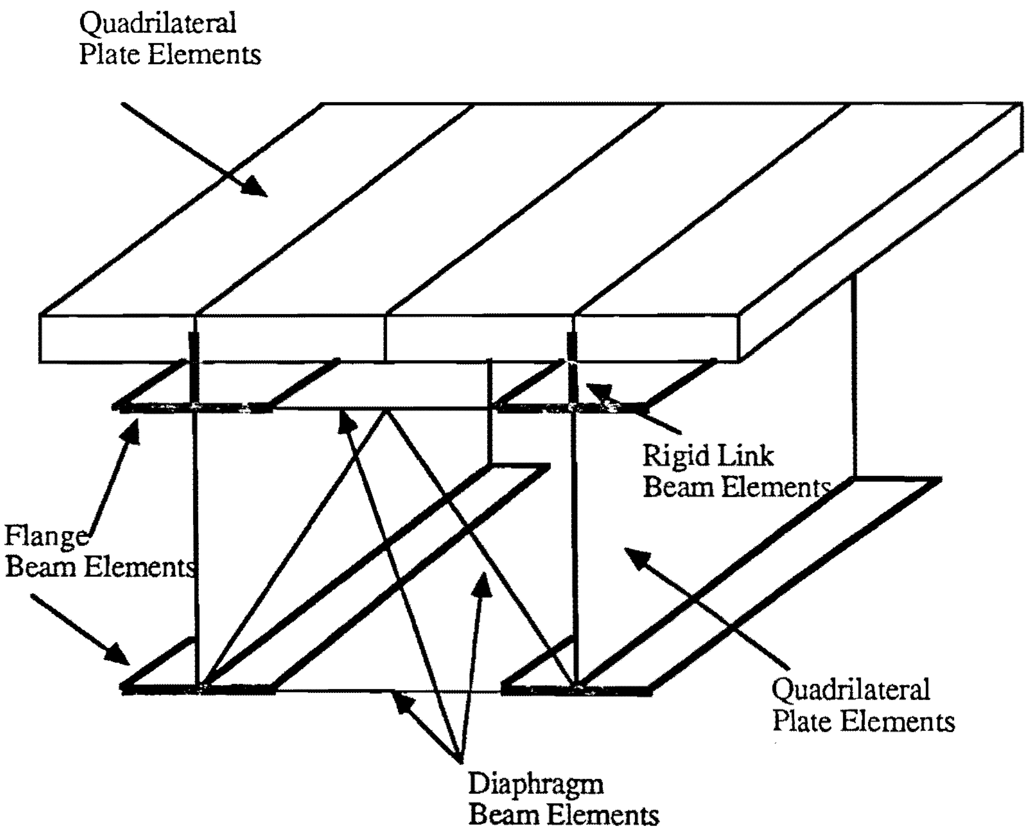


Figure 2.2 Substructure model

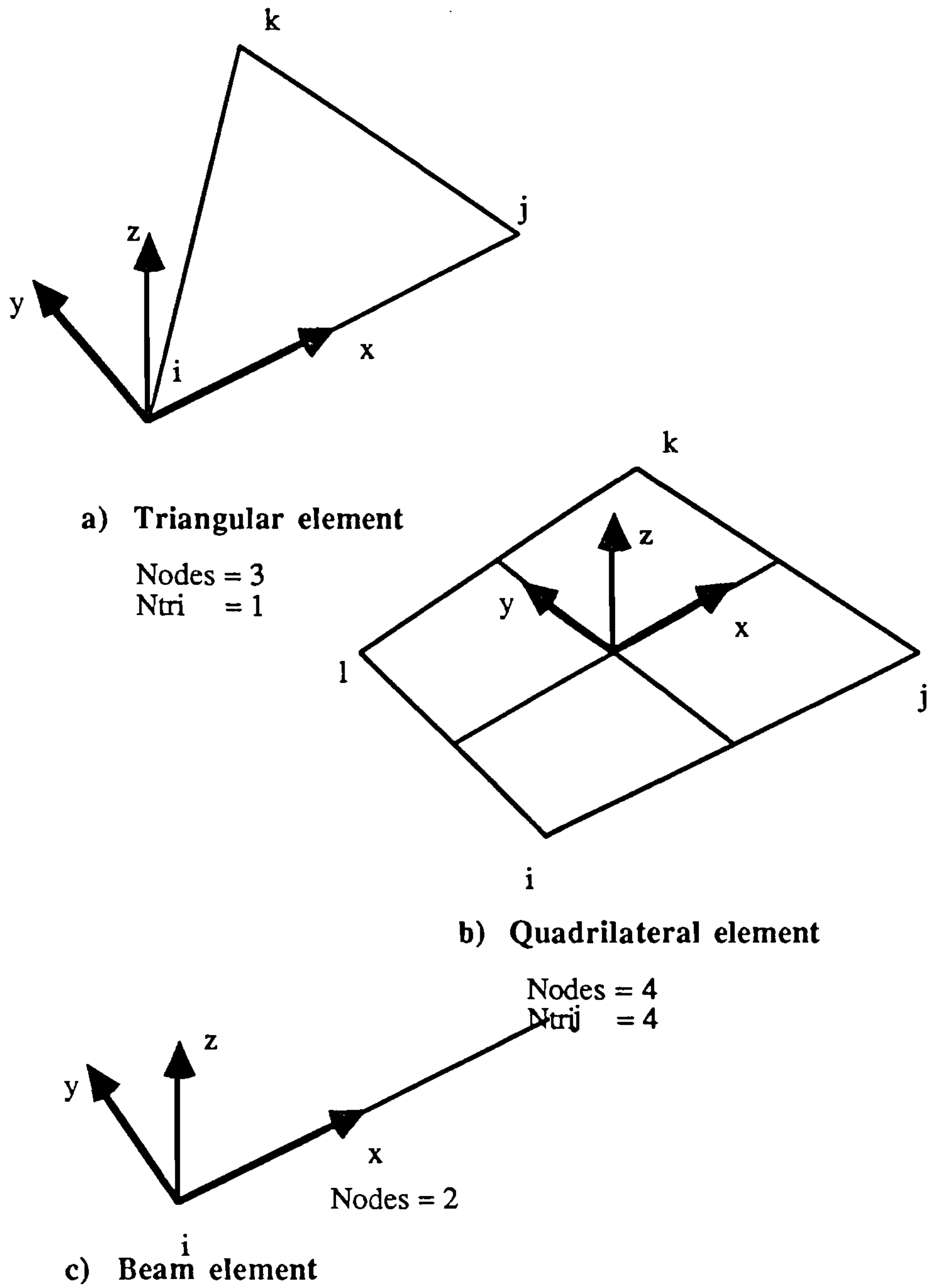


Figure 2.3 Finite elements

The locations of the axes are shown in Figures 2.4a and 2.4b. A positive radius of curvature is one in which the substructure curves to the right and a negative radius of curvature is one in which the substructure curves to the left. As an example, the curvature shown in Figure 2.4a is positive.

An example of the node numbering process will help clarify things as shown in Figure 2.5. The data generator builds from left to right starting with the slab nodes. The outside overhang node is numbered first and is placed at the mid-depth of the concrete slab. All the concrete slab nodes are place at the same vertical distance equal to h2. The value h2 equals half the average bottom flange thickness plus the web depth plus the average top flange thickness plus the soffit plus half the concrete thickness as shown in Figure 2.6a. The data generator then moves down and determines the top flange nodes placed at a vertical distance h1. The value h1 equals half the average bottom flange thickness plus the web depth plus half the average top flange thickness as shown in Figure 2.6a.

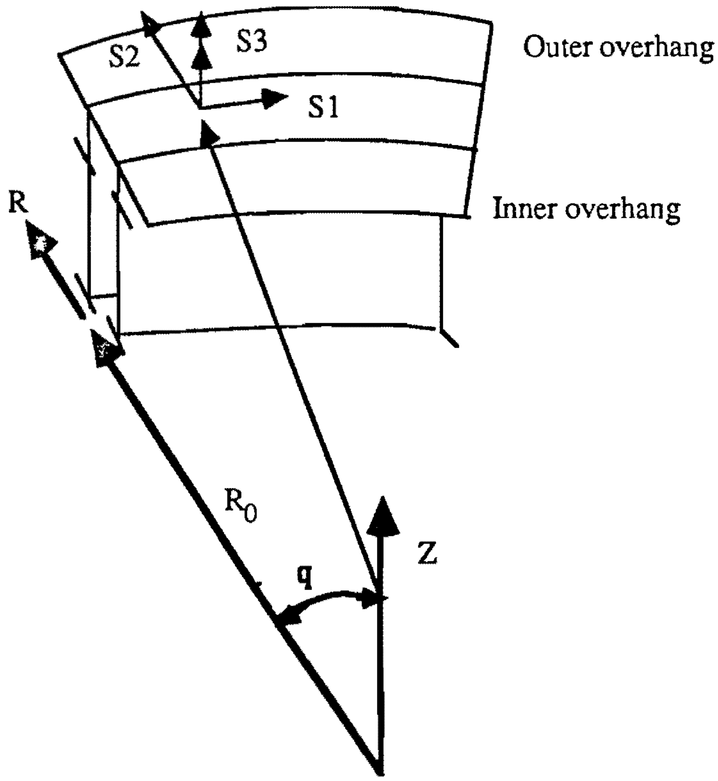
The bottom nodes are place at the mid-depth of the bottom flange. The average bottom flange and average top flange thicknesses were input so that the vertical placement of the nodes is constant for the entire bridge. This was a simplification that was done so that better advantage could be taken of repeated substructures. When flanges change dimensions, the vertical location of the node does not change but the new properties are centered on the node as shown in Figure 2.6b. Also, the girder spacings do not change relative to each other but the girder spacings do not have to equal each other.

Three types of diaphragm bracings can be used with this data generator and they are the K-brace (Figure 2.7c), the X-brace with a top horizontal member (Figure 2.7a), and the X-brace without a top horizontal member (Figure 2.7b). The node for the K-bracing is placed at h1(Fig. 2.7c) and at half of h1 for the X-bracing as shown in Figures 2.7a and 2.7b. A node was place at the intersection of the X-brace so that all brace configurations would require the same number of nodes.

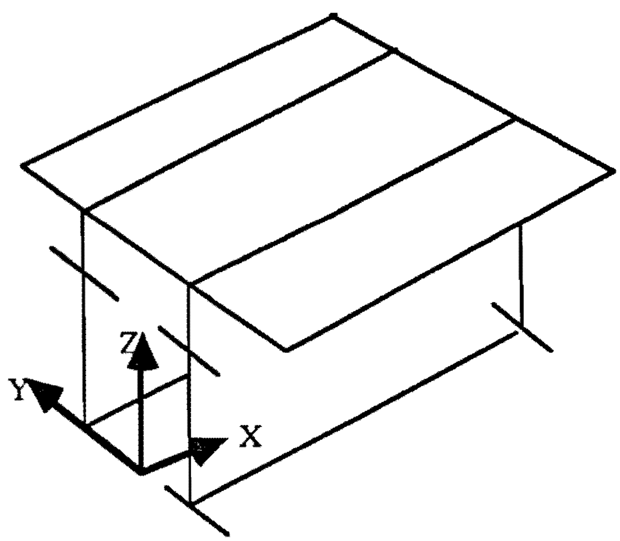
The next transverse row of nodes are placed at the location equal to the diaphragm spacing divided by the number of divisions between diaphragms. An example of this is section B-B of Figure 2.5. The diaphragms are added on the end and this is shown in section C-C of Figure 2.5.

2.3.2 Element Node Numbers

The element node numbers are determined after the nodal coordinates have been established as shown in Fig. 2.7. The top nodes of the diaphragm on the left of the substructure between the first and second girders are numbered first. The exact numbering sequence is dependant on the type of diaphragm used as shown in Figure 2.7. After the diaphragms are numbered, the nodes for a rigid link between the top flange and the concrete slab are determined. The slab element is numbered counter-clockwise as viewed from above starting with node two. In this way, the local x-axis and y-axis are generally in the same direction as the global axis. The slab elements are then numbered next from left to right. After the slab, the top flanges of the girders are numbered from left to right. The node numbers for the web elements are determined next with node i equal to the first bottom flange node. The nodes of the bottom flanges are then numbered from left to right. The process is repeated for each division between the diaphragms. The diaphragms are only added at the end of each substructure. The element node numbers for the bottom horizontal bracing are determined last and the bottom brace sequence is in the same order which they were input. The physical properties of the elements are determined at the same time as the node numbers.



a) Curved substructure with positive radius



b) Straight substructure

Figure 2.4 Straight and curved substructure axes

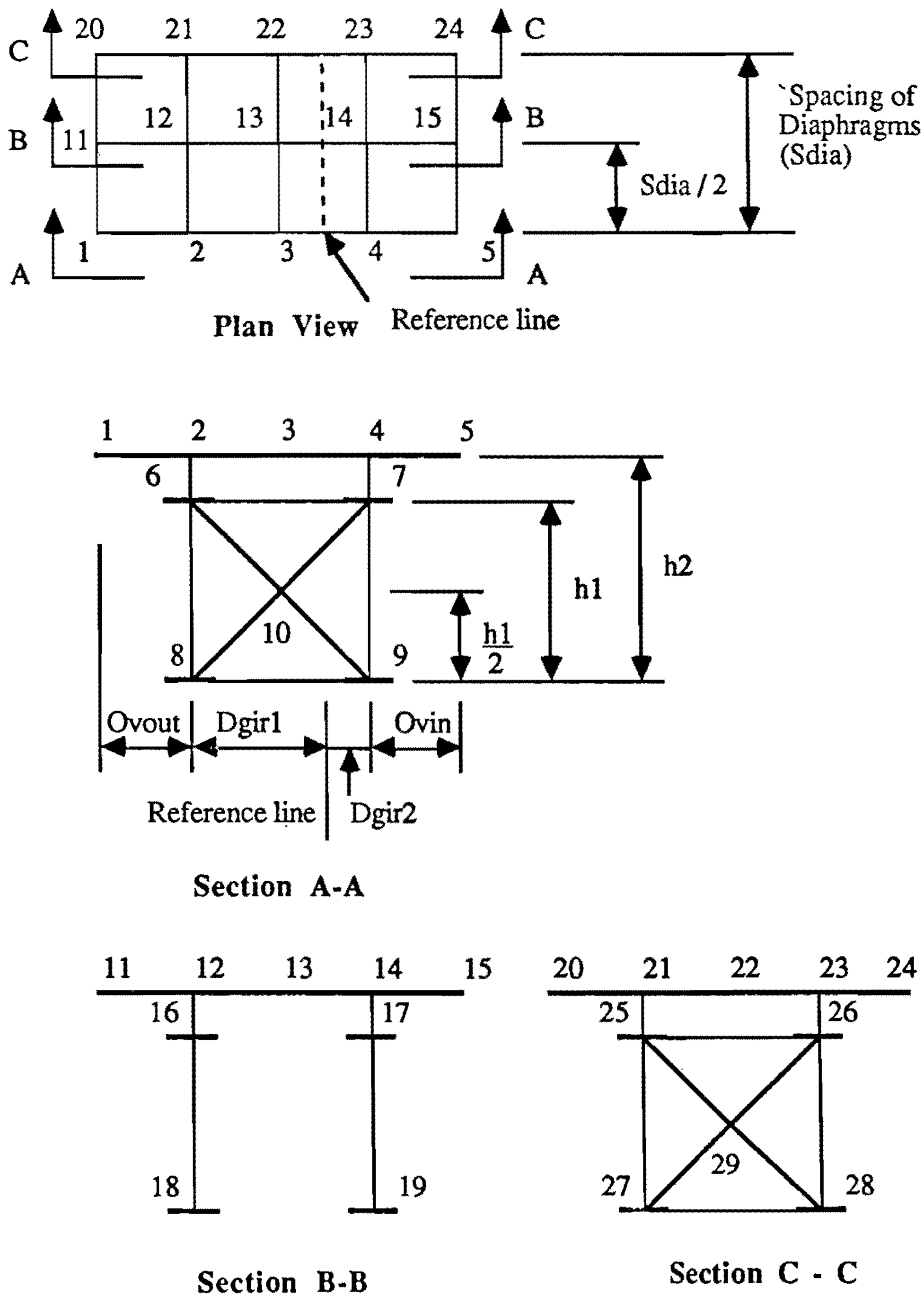


Figure 2.5 Element node numbering.

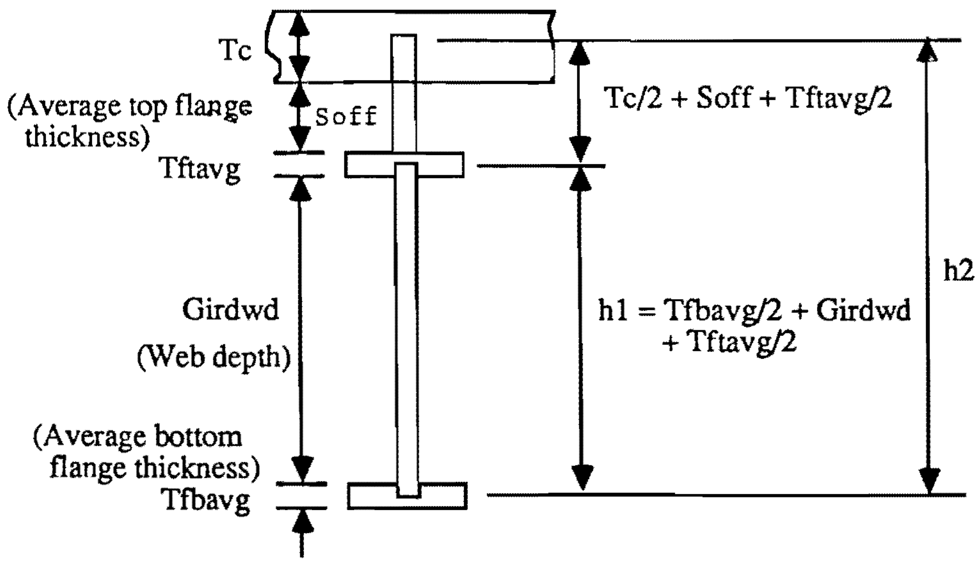


Figure 2.6a Node location.

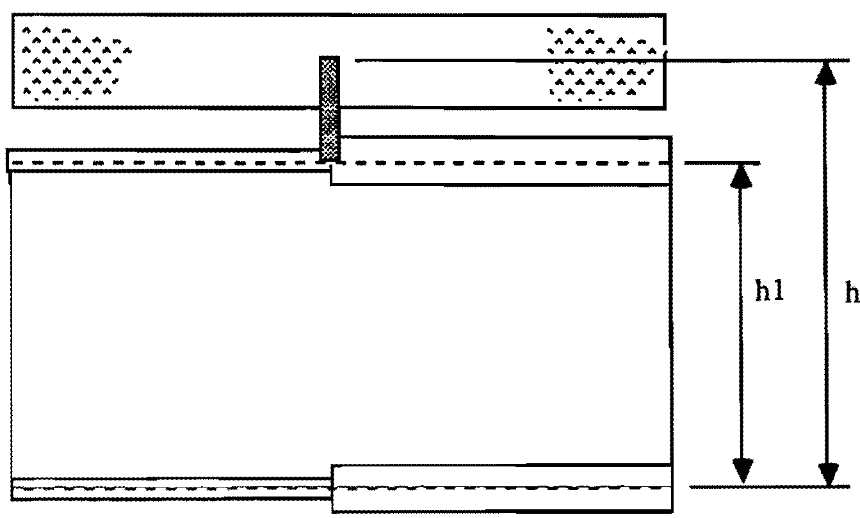


Figure 2.6b Model with change in flange dimension.

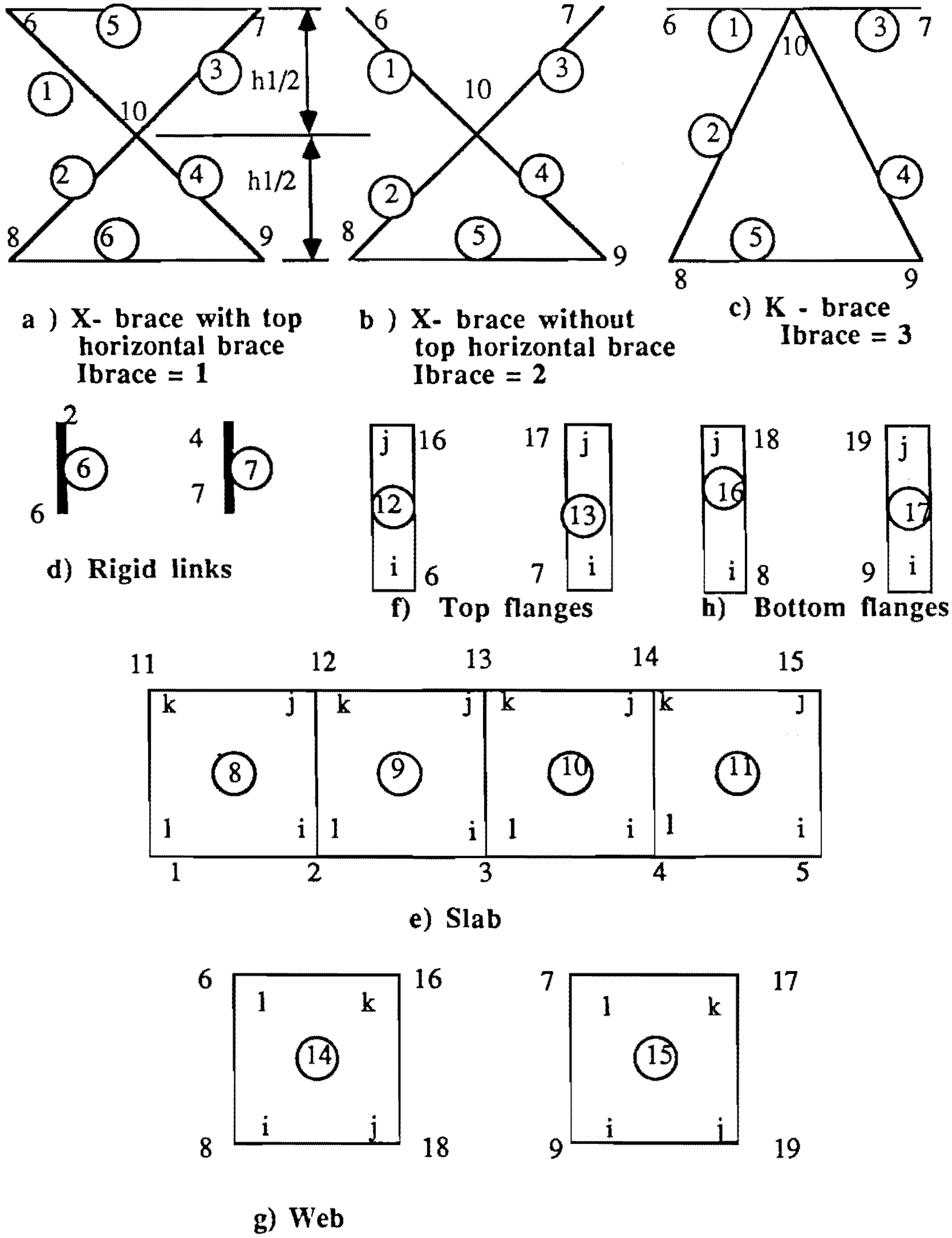


Figure 2.7 Element nodes

2.3.3 Automatic Mesh Size

The program has the capability to determine a first approximation for the refinement of the mesh. The coarseness of the mesh is done with the idea to keep the elements as square as possible. Since the web only has one plate element in the vertical direction, the program sets the size of the rest of the plate elements as close as possible to the web depth. This routine should be used with a cautious eye since the coarseness of the mesh is only done once for the bridge. If the diaphragm spacings change much then the size of the mesh may not be the most economical and worse yet the answers may not be reliable. The aspect ratio of the quadrilateral plate elements are given so that any problems can be detected.

2.3.4 Loads

All loads are placed as equivalent concentrated loads at the nodes. Dead loads due to surface loads on the concrete slab are superimposed on the concrete slab elements as an increase in density. Edge loads on the overhang are smeared over the overhang elements as an increase in density of the overhang elements. Live loads due to truck loading or lane loading are also superimposed on the nodes of the concrete slab elements. In addition, it is possible to add more loads from input data by hand onto the bridge. This would represent loadings not covered by the data generator such as wind loads and braking loads.

2.3.5 Changing Substructure Values

A new substructure needs to be defined if there is any difference between the previous substructure. For example, a new substructure is needed if the flange dimensions change. The only additional input required are the values that change from the previous substructure. When this update is done, the number of repeated substructures of this same type must always be included. The data generator then generates the substructure again. Also, changing loads between substructures do not require a new substructure.

2.3.6 Custom Substructures

If a part of the geometry of the bridge does not lend itself to the data generator routine then the user must input that substructure by hand. This input follows the documentation for section 2.4.6. After all the data are included, then this custom substructure can be placed as the next substructure in the bridge. The bridge can thus contain a mixture of custom and data-generated substructures.

2.3.7 Master Nodes

The arrangement of the substructures to form the entire bridge is done with master nodes. These master nodes are at the intersection of all substructures and at each end of the bridge. In a simplified sense, the master nodes are the global numbering of all the diaphragms. The local nodes of each substructure must be renumbered to match these master nodes. The data generator does all of this automatically but the user must understand the concept of master nodes in order to interpret the output.

2.3.8 Boundary Conditions

The boundary conditions may be imposed on either the master nodes or the internal nodes of a substructure. In most cases, master nodes are used because the boundary conditions are imposed at diaphragm locations. When the support is between diaphragms, then the boundary conditions must correspond to the local nodes for that substructure. The most common occurrence would be if the support is skewed.

The data generator has the capability to handle two kinds of substructures which have skewed supports. The distinction is made between a slight and a severe skew. A slight skew is one in which the mesh is slightly distorted to accomodate the support skew as shown in Figures 2.8 and 2.18. The supports can be at the diaphragms in which case the diaphragms follow the support line and the master nodes are used. The support can be placed between the diaphragms in which case local nodes are used. Usually diaphragms are required at support locations but this modeling allows radial supports when the support is only slightly skewed. The slight skew substructrue (IROUTE=3) is most versatile in that it can be used at both end and interior locations as shown in Figure 2.8b with little additional input. Also, the skewed substructure can be used without support conditions which may be useful next to skewed supports.

A severe skew is one in which the diaphragms intersect at the point where the girders are supported as shown in Figure 2.8c. The diaphragms still bound a single substructure but the supports can span one or more substructures as shown in Figure 2.19. Because of the difficulty in creating a general data generator for a severely skewed end support, the severely skewed substructure (IROUTE =4) only applies to interior supports. If the support conditions do not correspond to these categories then the boundary conditions must be input by hand. See the respective sections for further clarification.

The data generator allows the user to specifiy many things pertaining to the support conditions. All supports are vertically restrained but the user can specify which support is horizontally restrained. In addition, the user can input vertical support spring stiffnesses or intitial vertical displacements for any and all supports. This is especially useful if different elastomeric pads are used or a support undergoes settlement.

2.3.9 Summing Load Cases

The program has the capability to sum stresses for different load cases. In this way, it is convenient to run the dead load case and then run the live load for the worst case loading. The stresses for the dead load and live load can then be summed up with the stresses for the latest run printed out alongside the total stresses.

2.3.10 Erection Stages

The erection procedure can be approximated by making separate computer runs for each stage. Each erection stage is input with the girders and the length to be erected. The program then uses the density and the elastic modulus to model the bridge with the stiffness of all the previous stages but only the load of the last stage. Superposition is then valid to sum up the stresses for the erection procedure in the same way different load cases are summed.

2.3.11 Substructure Types

There are thus five possible types of substructures which can be used in this program. They are:

1) An initial mesh-generated substructure in which all the data to generate the mesh is input. When substructure 3 or 4 is the first data generated substructure then all the data for the bridge also needs to be input.
2) An update of some previously specified value of a mesh-generated substructure.
3) A substructure with a slightly skewed mesh. This substructure may or may not have supports. If it has supports then the supports are possible either at diaphragm locations or at the interior of the substructure.

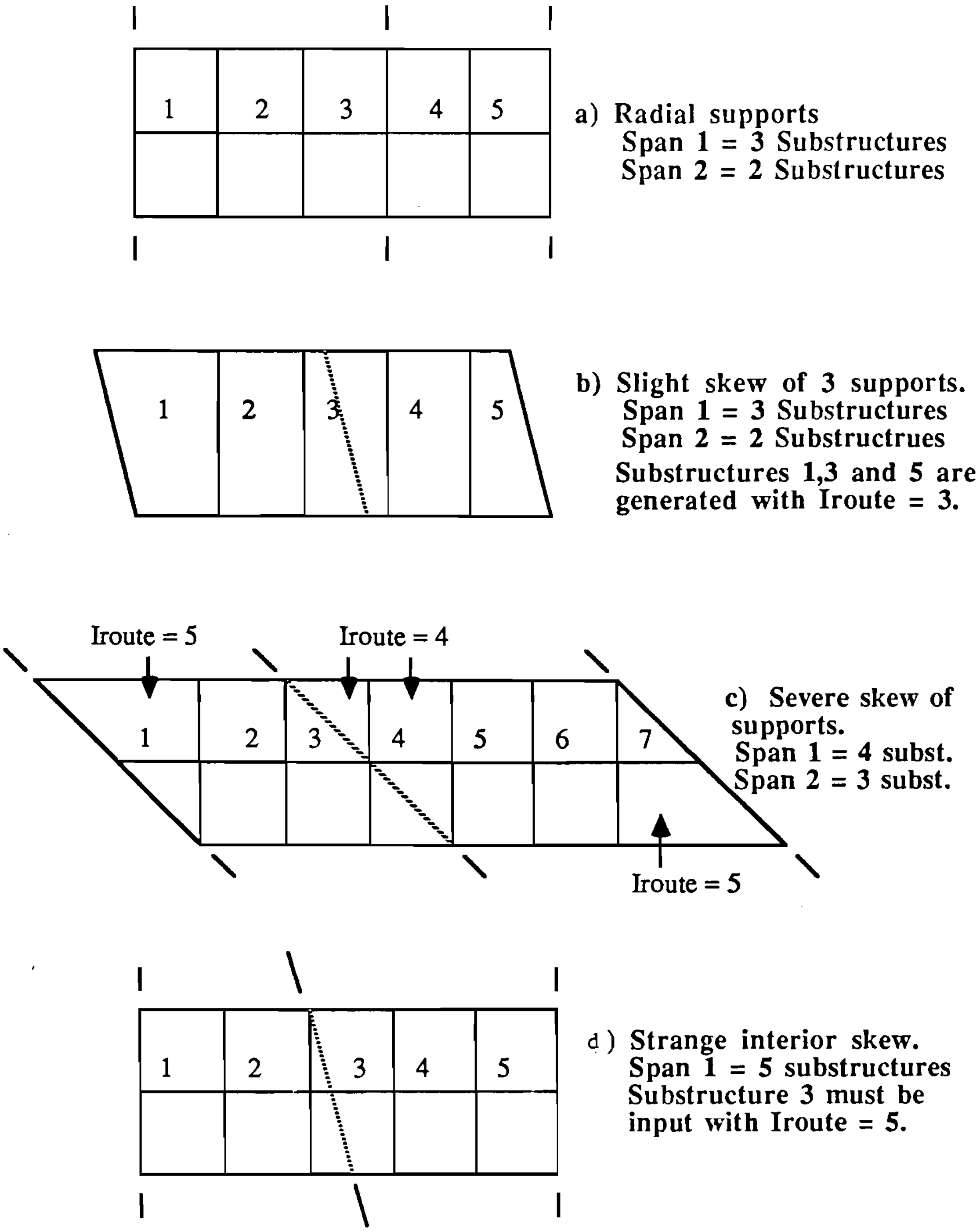


Figure 2.8 Number of substructure per span.

4) A substructure with the ability to handle severely skewed supports. This substructure may have supports at both diaphragm and interior node locations but it is only good for interior supports of the bridge.

5) A custom substructure that does not lend itself to any of the data generated substructures.

2.4 INPUT DOCUMENTATION

This section describes the input required for GENPUZ, PUZF83 AND RECPUZ. These programs combine to form KURV87. The variable corresponding to the five types of substructures is called IROUTE. The documentation will show how to prepare the data files for all of these types of substructures. Further explanations are found in sections 2.5, 2.6 and 2.7. Appendix C includes the input in tabular form for easy reference. The labels in paranthesis are the same ones used in the program and help clarify explanations in other parts of this documentaion. In the description of the input data the following abbreviations apply:

A = Alphanumeric
F = Floating point number (must be typed with a decimal point).
I = Integer value (must be packed to the right of the field).

2.4.1 **Initial Data for Program**

A) CONTROL CARDS

LINE	COLS.	TYPE	
1	1 - 80	A	Labeling information for this computer run.
2	1 - 80	A	Labeling information for this computer run.
3	1 - 5	I	Number of different types of substructures. 196 Maximum (Nogp)
	6 - 10	I	Plotting option. (Ipl) = 1 , to execute plots in XY and YZ plane for each substructure. = 2 , to also execute plots of concrete slab stresses by nodes.
	11 - 15	I	Output print option. (Idef) = 0 , for minimum output display. This outputs only stresses and the minimum echoing of input data. = 1 , for additional output display. This outputs deflections by master nodes and more echoing of check prints. = 2 , for maximum output display. This outputs deflections by both element and master node with more check prints. For large problems, there can be a lot of elements and this output will be quite voluminous.
	16 - 20	I	Load case of this computer run. Also used as erection stage. (Jrec)
	21 - 25	I	Number of steel erection stages to be input. (Irec) This number will be different than the current erection stage if concrete stages are to be superimposed on the steel stages.

LINE	COLS.	TYPE	
	26 - 30	I	Summation of stresses with previous load case. (Ircsum) = 0 , no stress summation. = 1 , if the stresses of this run are to be added to the last run which Ircsum equaled one. This is used to add dead load to live load and for summation of erection stage stresses.
	31 - 35	I	Number of prescribed support displacements or spring constants to be input. (Nospg) Support settlements and springsapply to the vertical direction only.
	36 - 40	I	Automatic mesh size determination. (Kgen) = 0 , number of mesh divisions to be input by user. = 1 , program internally generates size of mesh.
	41 - 45	I	Composite or noncomposite girder. (Komp) This choice makes the rigid link a truss element and sets all concrete slab stresses equal to zero. = 0 , Girder is composite. = 1 , Girder is noncomposite.

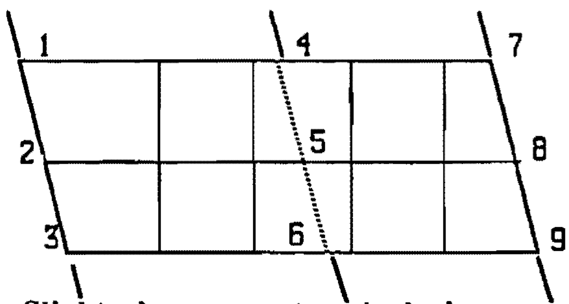
B) SUPPORT CONDITIONS

The support conditions are specified by the number of spans and the number of substructures per span. For further information see Section 2.5.2 and Figure 2.8.

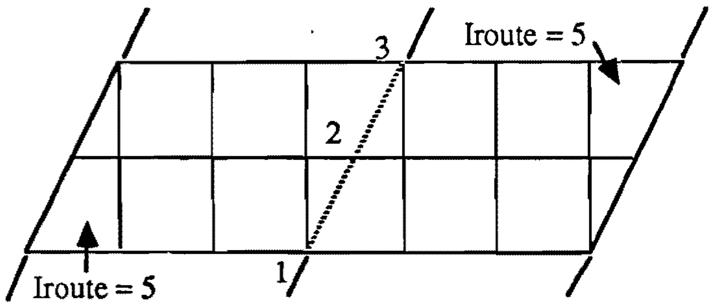
LINE	COLS.	TYPE	
1	1 - 5	I	Number of spans. 10 maximum (Nspan)
	6 - 10	I	Number of substructures until the support is pinned. (Nfix) = 0 , if bottom of girders at the first diaphragm are pinned. >= 0 , if bottom of girders at the first diaphragm are on rollers. <= 0 , if bottom of girders at the first diaphragm are not supported.
	11 - 15	I	Number of substructures for first span. (Nsubbc) For substructures with skewed supports, count the number of substructures in a span until the next substructure is not supported. An example is shown in Figure 2.8.
	16 - 20	I	Number of substructures for the second span. Input the number of substructures for each span in "I5" format until all spans are input.

C) PRESCRIBED SUPPORT CONDITIONS

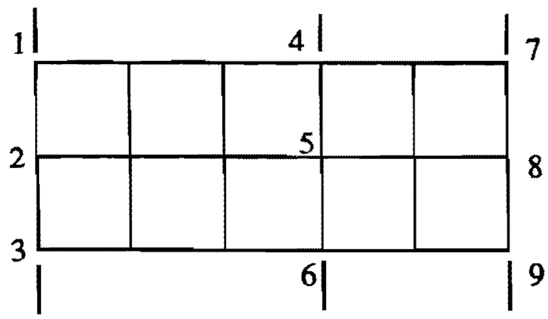
The number of lines for input should equal the number of support displacements or spring stiffnesses specified in the control cards (Nospg). The support restraints must be input in the order that they are encountered when numbered from left to right. Thus, they shall be input in increasing order relating to the master nodes. When the support is in the interior of a substructure then the input should follow all the input to the left of the interior support. A couple of examples of the proper order of input are shown in Figure 2.9. Also, if one girder at a support has the input for restraints then all the girders at that support must have boundary conditions input.



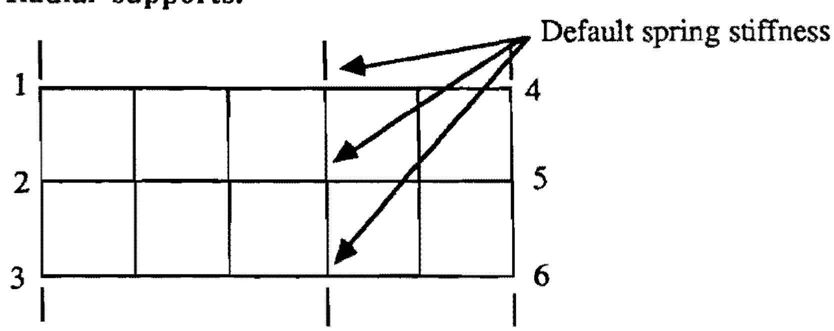
a) Slight skew, counter-clockwise



b) Severe skew at all supports. End support conditions input with section 2.4.6 parts K and N.



c) Radial supports.



d) Radial supports with only end supports prescribed.

Figure 2.9 Input order for prescribed support conditions.

LINE	COLS.	TYPE	
1	1 - 5	I	Number of substructures until a support. (Jspg) This number should correspond to the total number of substructures going from left to right.
	6 - 10	I	Displacement or spring index. (Kspg) = 2 , for input corresponding to a prescribed displacement. = 3 , for input corresponding to a spring stiffness.
	11 - 20	F	Displacement or spring value. (Vspg) = displacement specified. Positive upward. - inches - = spring stiffness specified. - kips / inch -

D) ERECTION PROCEDURE (one line for each stage)

LINE	COLS.	TYPE	
1	1 - 10	F	Starting girder for erection stage. (Kgrst)
	11 - 20	F	Ending girder for erection stage. (Kgren)
	21 - 30	F	Starting substructure for erection stage.(Ksbst)
	31 - 40	F	Ending substructure for erection stage. (Ksben) If just the diaphragms are added between two girders then the first two entries must be input as the negative of the girder numbers. The starting substructure must start with zero at the beginning of a bridge. If only one substructure is added, then the last two entries must be the same value. See the example of a four-girder bridge as shown in Figure 2.10.

2.4.2 IROUTE = 1

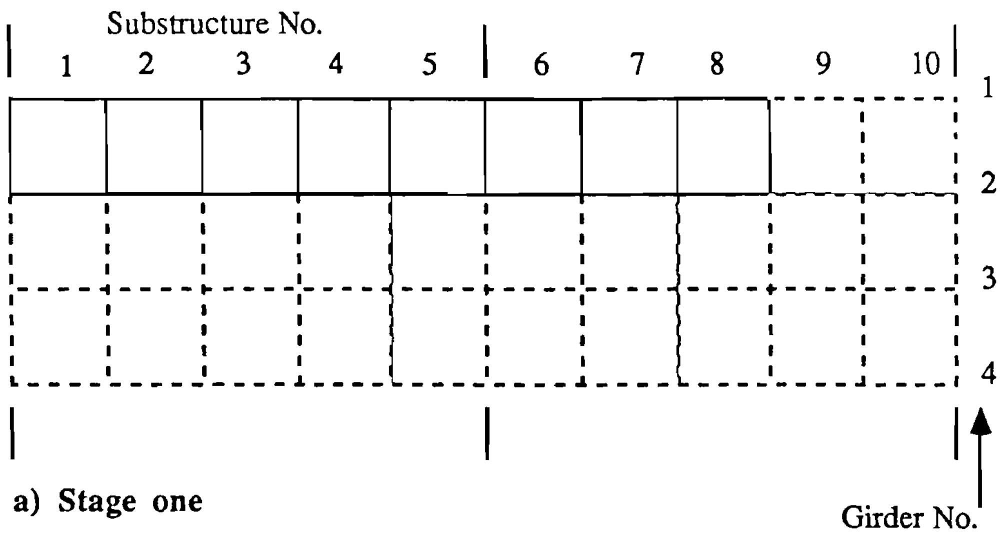
This choice generates the first data-generated substructure based upon the initial bridge data. This choice should be done only once. Subsequent changes should be done with IROUTE = 2 so that values that do not change from one substructure to the next do not have to be input again. If the intial mesh is for a skewed support, i.e., IROUTE = 3, or = 4, then this input must come first and the input required for either IROUTE = 3 or IROUTE = 4 should immediately follow.

A) SUBSTRUCTURE INDEX

LINE	COLS.	TYPE	
1	1 - 5	I	= 1 , Type of substructure. (Iroute)

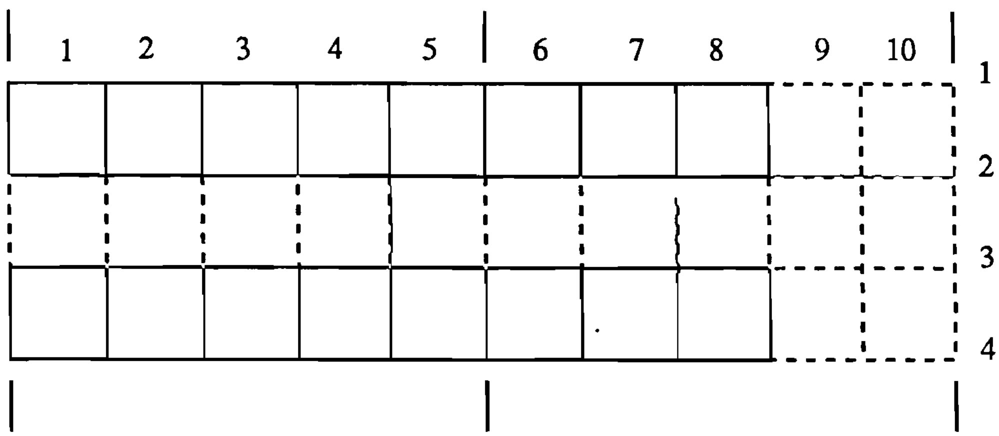
B) BRIDGE GEOMETRY

LINE	COLS.	TYPE	
1	1 - 5	I	Number of times this same substructure is repeated consecutively. 16 Maximum. (Nv)
	6 - 10	I	Number of mesh divisions between girders. 10 Maximum. (Idivt)



a) **Stage one**

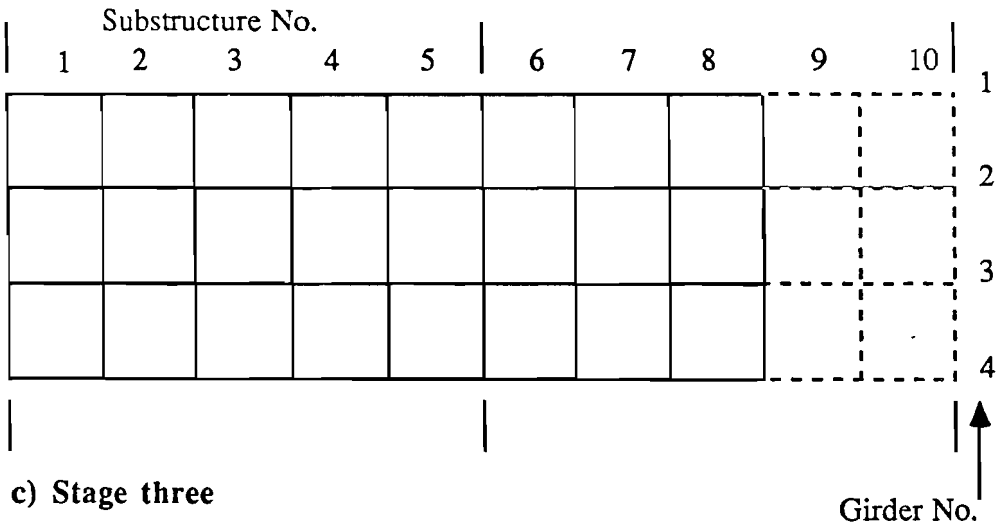
Stage no.	Starting Sub.	Ending Sub.	Starting Girder	Ending Girder
1	0	8	1	2



b) **Stage two**

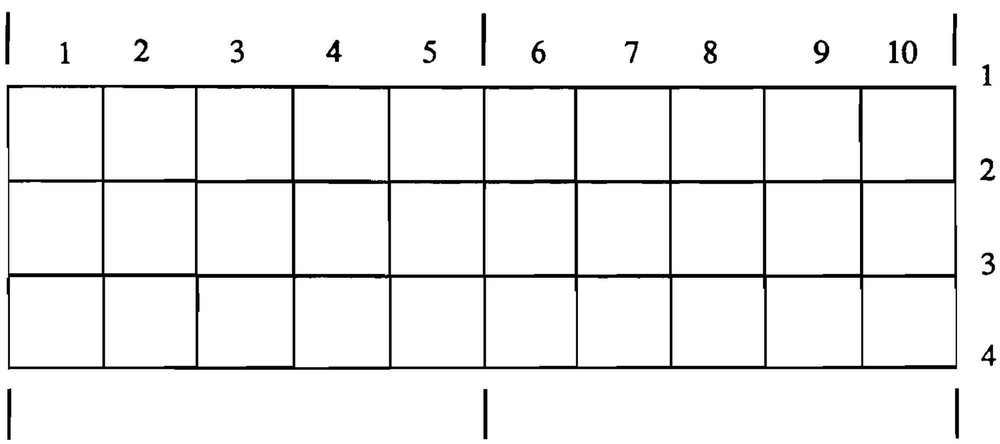
Stage no.	Starting Sub.	Ending Sub.	Starting Girder	Ending Girder
1	0	8	1	2
2	0	8	3	4

Figure 2.10a,b Erection stage example



c) Stage three

Stage no.	Starting Sub.	Ending Sub.	Starting Girder	Ending Girder
1	0	8	1	2
2	0	8	3	4
3	0	8	-2	-3



d) Stage four

Stage no.	Starting Sub.	Ending Sub.	Starting Girder	Ending Girder
1	0	8	1	2
2	0	8	3	4
3	0	8	-2	-3
4	9	10	1	4

Figure 2.10c,d Erection stage example

LINE	COLS.	TYPE	
	11 - 15	I	Number of mesh divisions between diaphragms. 10 Maximum. (Idivl) Idivt and Idivl are not needed if mesh size is automatically determined.
	16 - 20	I	Number of girders. 10 maximum. (Ngir)
2	1 - 10	F	Reference radius of bridge. -feet- See Fig. 2.4b. (Rref) = 0, For a straight substructure. >= 0, For a substructure that curves right. <= 0, For a substructure that curves left.
	11 - 20	F	Thickness of concrete slab. -in- (Tc)
	21 - 30	F	Inside overhang of concrete slab. -ft- (Ovin)
	31 - 40	F	Outside overhang of concrete slab. -ft- (Ovout)
	41 - 50	F	Reference line spacing of diaphragms. Reference line arc length between diaphragms for curved substructures. -ft- (Sdia)

C) GIRDER INFORMATION

LINE	COLS.	TYPE	
1	1 - 10	F	Distance from reference line to girders. (Rgir) Input all distances of girders in ascending order of girder numbers in "F10" field length. Use more than one line if required. -ft-
	" - "	"	" " " '
	71 - 80	F	Distance from reference line to girder.
2	1 - 10	F	Web depth. -in- (Girdwd) Not including flange thicknesses.
	11 - 20	F	Web thickness. -in- (Tw)
	21 - 30	F	Average top flange thickness for entire structure. -in- (Tftavg)
	31 - 40	F	Average bottom flange thickenss for entire structure. - in- (Tfbavg)
	41 - 50	F	Distance from top of the flange to bottom of the concrete slab. Assumed constant. -in- (Soff)
			Input the following for each girder.
3	1 - 10	F	Thickness of top flange. -in- (Tft)
	11 - 20	F	Width of top flange. -in- (Bft)
	21 - 30	F	Thickness of bottom flange. -in- (Tfb)
	31 - 40	F	Width of bottom flange. -in- (Bfb)

D) DIAPHRAGM INFORMATION

LINE	COLS.	TYPE	
1	1 - 5	I	Diaphragm bracing configuration for beginning of substructure. (Ibrace(1)) = 1, For X-bracing with top horizontal member.

LINE	COLS.	TYPE	
			= 2, For X-bracing without top horizontal member.
			= 3, For K-bracing.
	6 - 10	I	Diaphragm bracing configuration for end of substructure. As above. (Ibrace(2))
	11- 15	I	Number of different types of diaphragm bracing members. (Ndifbr)
	16 - 20	I	Number of bottom horizontal braces. (Ibrbot) All assumed to have the same properties.
2	1 - 10	F	Half the area of the diagonal diaphragm brace -in2- (Adia)
	11- 20	F	Half the moment of inertia for the brace about the horizontal axis. -in4- (Diy)
	21 - 30	F	Half the moment of inertia for the brace about the vertical axis. -in4- (Diz)
	31 - 40	F	Half the torsional moment of inertia. -in4- (Torj)
	41 - 45	I	Description index for locating the properties of the diaphragms. (Ibr) See Figure 2.11 for further information. The number corresponds to the following description : = 1, For diagonal member of beginning brace. = 2, For bottom member of beginning brace. = 3, For top member of beginning brace. = 4, For diagonal member of ending brace. = 5, For bottom member of ending brace. = 6, For top member of ending brace.
	46 - 50	I	Description index for second brace member
	51 - 55	I	Description index for third brace member
	56 - 60	I	Description index for fourth brace member
	61 - 65	I	Description index for fifth brace member
	66 - 70	I	Description index for sixth brace member.

E) BOTTOM HORIZONTAL LATERAL BRACING INFORMATION

LINE	COLS.	TYPE	
1	1 - 10	F	Full area of diaphragm for bottom horizontal bracing. -in2- (Adia(7))
	11 - 20	F	Full moment of inertia for bottom horizontal bracing about the horizontal axis. -in4- (Diy(7))
	21 - 30	F	Full moment of inertia for bottom horizontal bracing about the vertical axis. -in4- (Diz(7))
	31 - 40	F	Full torsional moment of inertia for bottom horizontal bracing. -in4- (Torj(7))
2	1 - 5	I	Girder number in which the brace starts. (Ngsbot)
	6 - 10	I	Girder number in which the brace ends. (Ngebot) This must be input for each bottom brace. The start of the substructure is the cross-section in which the node numbering begins. See Figure 2.12 for further information.

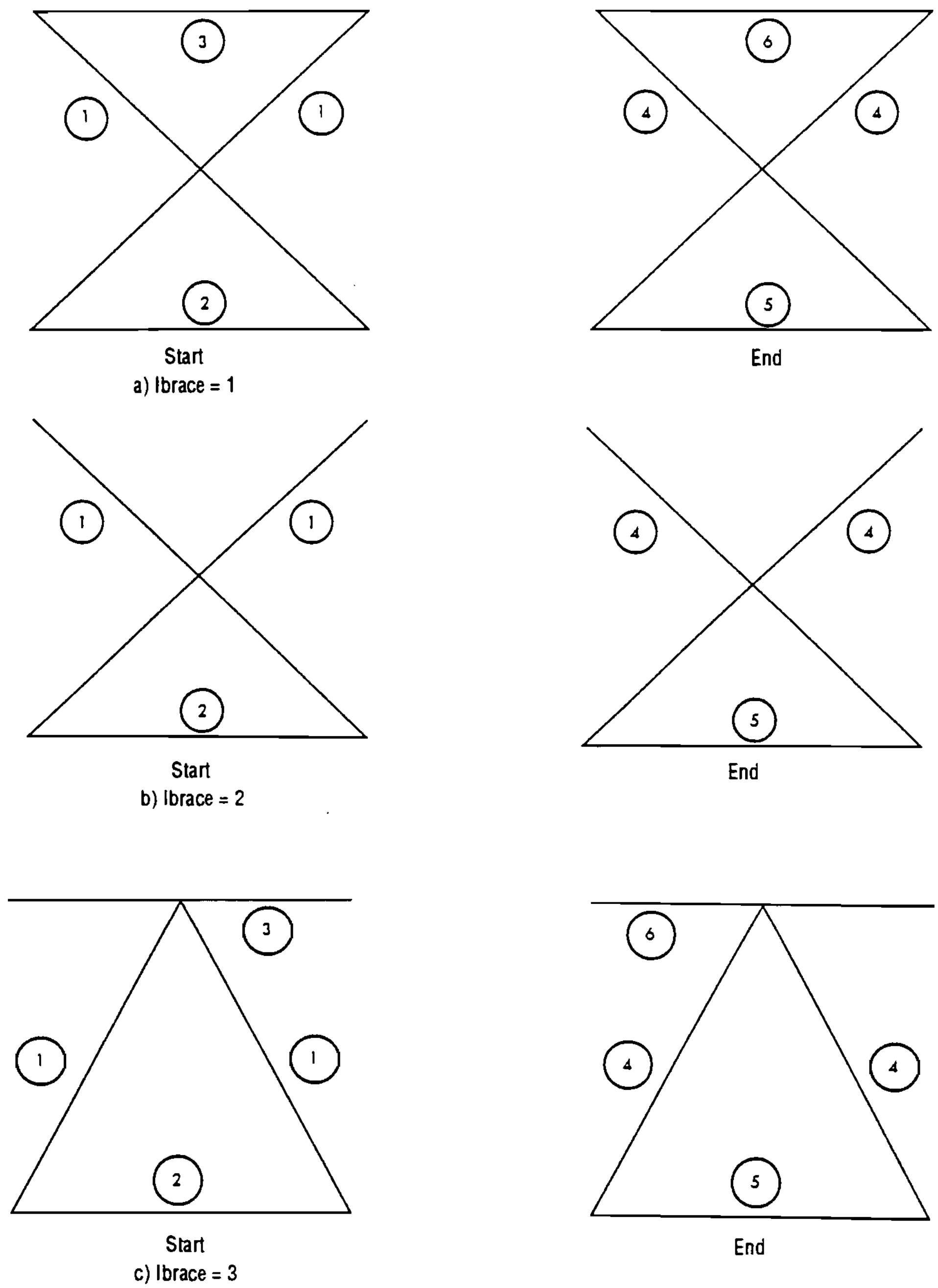


Figure 2.11 Diaphragm members

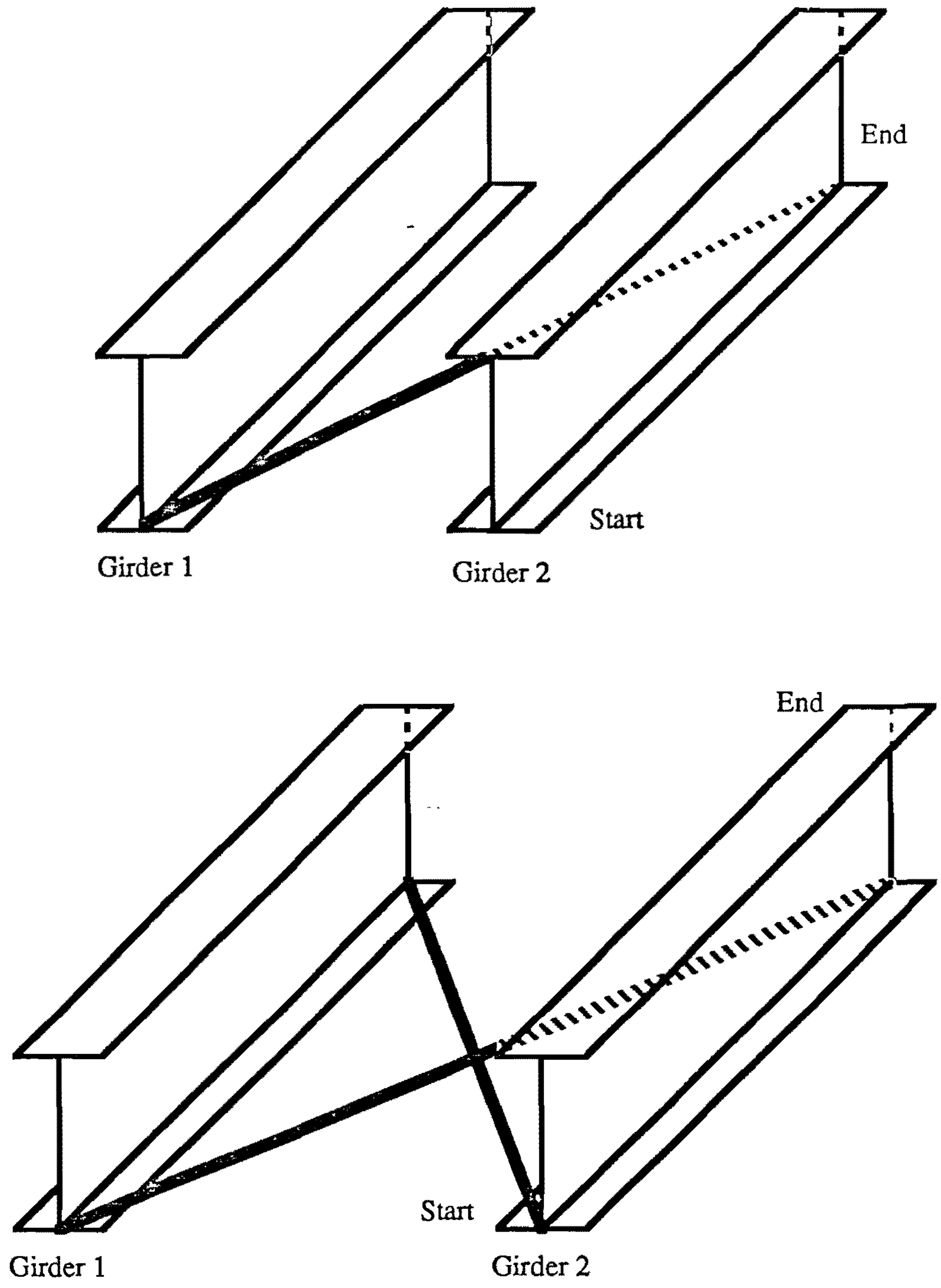


Figure 2.12 Horizontal lateral bracing

F) MATERIAL PROPERTIES

LINE	COLS.	TYPE	
1	1 - 10	F	Modulus of elasticity for concrete in the x-axis direction. -ksi- (Ec(1))
	11 - 20	F	Modulus of elasticity for concrete in the y-axis direction. -ksi- (Ec(2))
	21 - 30	F	Poisson's ratio for concrete for a stress in the x-axis direction. (Poisc)
	31 - 40	F	Denstity of concrete. -kcf- (Densc)
2	1 - 10	F	Modulus of elasticity for steel. Assumed the same in both directions -ksi- (Est)
	11 - 20	F	Poissons's ratio for steel. (Poisst)
	21 - 30	F	Density of steel. -kcf- (Densst)

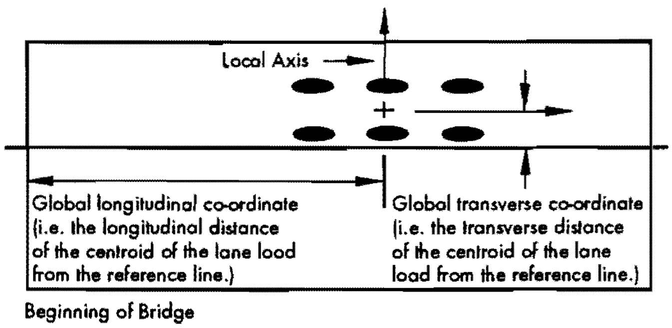
G) SLAB LOAD INFORMATION

LINE	COLS.	TYPE	
			Negative is downward.
1	1 - 10	F	Curb load of outside overhang. -k / ft- (Curbld(1))
	11 - 20	F	Curb load of inside overhang. -k / ft- (Curbld(2))
	21 - 30	F	Superimposed surface load on concrete slab. -ksf- (Surfld)
	31 - 40	F	Live load index. (Zlivld) = 0.0, For no live load. = 1.0, For truck loading. = 2.0, For lane loading.

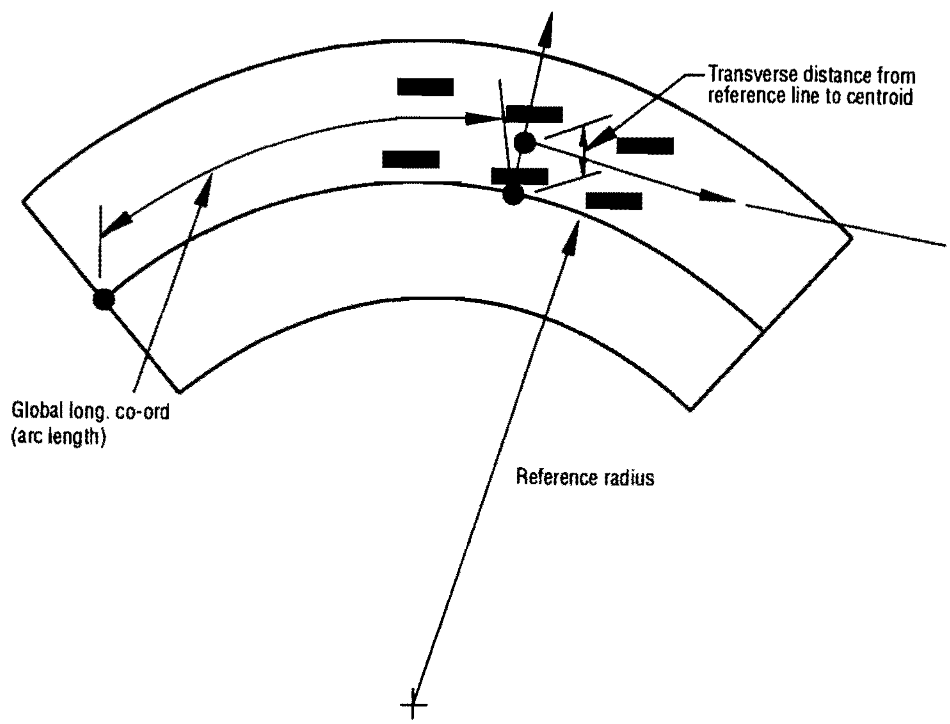
H) TRUCK LOAD INFORMATION

See Figures 2.13 and 2.14 for further description.

LINE	COLS.	TYPE	
1	1 - 5	I	Number of Trucks. (Ntruck)
2	1 - 8	A	Truck name.
	9 - 10	I	Repetition index. (Ibid) This value is used if the truck input directly preceding this one is the exact same. = 0, If this is a new truck geometry. = 1, If the geometry of this truck was input immediately preceding this truck.
	11 - 20	F	Load impact factor. (Dlf)
	21 - 30	F	Longitudinal location of centroid of truck from the reference line of the bridge. -ft- (Xpos)
	31 - 40	F	Transverse location of centroid of truck from the reference line of the bridge. -ft- (Ypos)

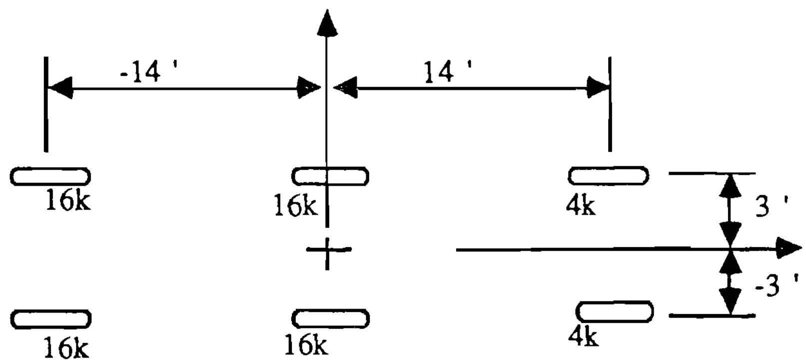


a) Truck placement on a straight bridge.



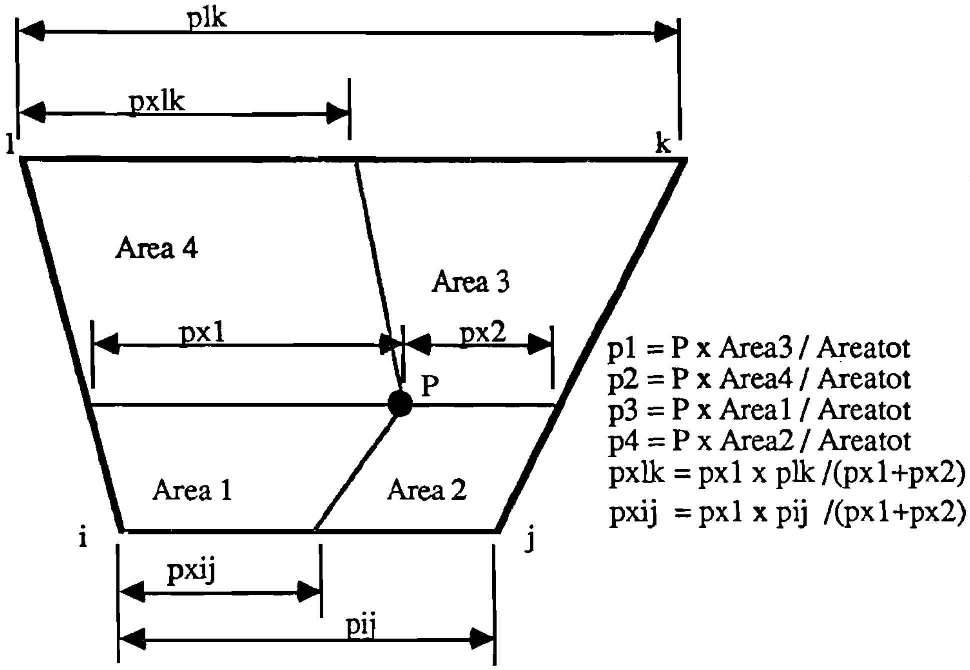
b) Truck placement on a curved bridge

Figure 2.13 Global truck positioning



a) **Truck example**

Load (kips)	Local transverse location (ft)	Local longitudinal location (ft)
4	3.0	14.0
4	- 3.0	14.0
16	3.0	0.0
16	-3.0	0.0
16	3.0	-14.0
16	-3.0	-14.0



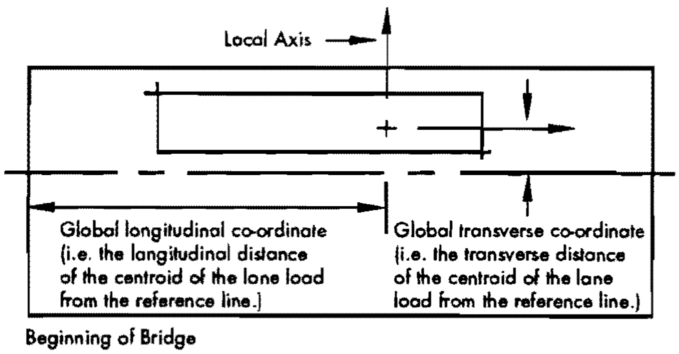
b) **Wheel load allocation to nodes**

Figure 2.14 Truck loads

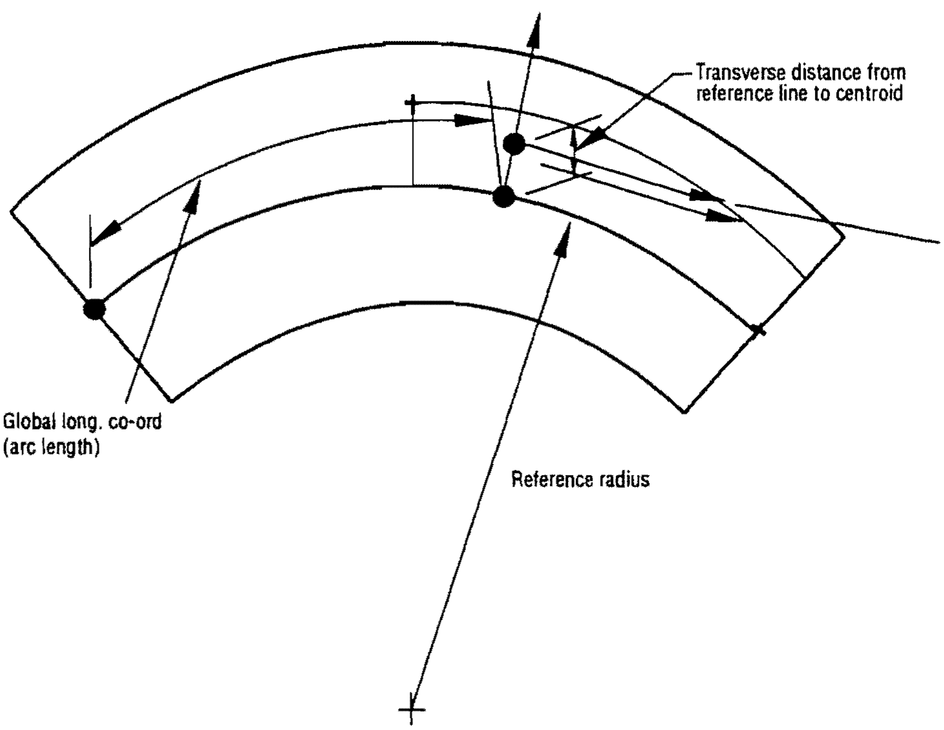
LINE	COLS.	TYPE	
	41 - 50	F	Radius of curve to the reference line of the bridge at the position of the centroid of the truck. -ft- (Radtr)
	52 - 55	A	Direction of truck. (Idir) = Forw, For a truck going from left to right. = Back, For a truck going from right to left.
			The next lines are needed if Ibid = 0. Positive is downward.
3	1 - 5	I	Number of wheels. (Nwheel)
4	1 - 10	F	Wheel load. -kips- (Trld)
	11 - 20	F	Local transverse location of wheel. -ft- (Centtr)
	21 - 30	F	Local longitudinal location of wheel. -ft- (Centlg)

I) LANE LOAD INFORMATION

LINE	COLS.	TYPE	
1	1 - 5	I	Number of lane loads. 10 Maximum. (Nlanld)
2	1 - 8	A	Name of lane load.
	11 - 20	F	Load impact factor. (Dlf)
	21 - 30	F	Longitudinal distance of the centroid of the lane load from the reference line of the bridge. (Xpos) -ft-
	31 - 40	F	Transverse distance of the centroid of the lane load from the reference line of the bridge. (Ypos) -ft-
	41 - 50	F	Radius of curve to the reference line of bridge during length of lane load. (Radln) -ft- If the curvature of the bridge changes in the lane load then two lane loads must be input corresponding to the respective radii.
3	1 - 5	I	Number of concentrated loads. (Nconld)
	6 - 15	F	Uniform lane load. This should be the same for all lane loads. Positive up -ksf- (Wdist)
4	1 - 10	F	Concentrated load. Negative down. (Trld) -kips-
	11 - 20	F	Local transverse location of load. (Centtr) - feet-
	21 - 30	F	Local longitudinal location of load. (Centlg) -feet-
5	1 - 10	F	Transverse location of upper left corner of the lane load. -ft-
	11 - 20	F	Longitudinal location of upper left corner of the lane load. -ft-
	21 - 30	F	Transverse location of lower right corner of the lane load. -ft-
	31 - 40	F	Longitudinal location of lower right corner of the lane load. -ft-

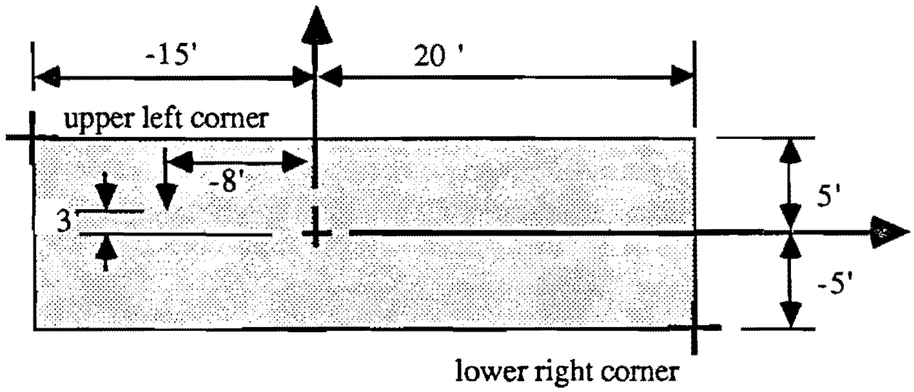


a) Lane load placement on a straight bridge.



b) Lane load placement on a curved bridge.

Figure 2.15 Global lane load positioning



1 concentrated load = 18 k

concentrated load coordinates in feet

local trans	local long
-8.0	3.0

distributed load = 0.064 ksf

corner coordinates in feet

local trans	local long.
- 15.0	5.0
20.0	-5.0

Figure 2.16 Lane load local coordinates

J) ADDITIONAL LOADS

Loads are input by elements. Each line of data is superimposed on loads that may exist for that member. End this section by placing a zero in column one. Signs of additional loads should follow element axis as shown in Fig. 2.3a, b and c.

LINE	COLS.	TYPE	
1	1 - 5	I	DOF of forces for this element (i.e., 1 - 6) (Idof)
	6 - 10	I	Substructure where this element is located. (Jsubld)
	11 - 15	I	Element number. (Kelno)
	21 - 30	F	Force at node I of this element. (P(1))
	31 - 40	F	Force at node J of this element. (P(2))
	41 - 50	F	Force at node K of this element. (P(3))
	51 - 60	F	Force at node L of this element. (P(4))

2.4.3 IROUTE = 2

This option is used to update a previously specified parameter. The parameters that can be changed are shown in the following list. Each parameter is assigned a number (Nalt). To change the value of a parameter, simply specify the parameter to be changed and the corresponding new value.

A) SUBSTRUCTURE INDEX

LINE	COLS.	TYPE	
1	1 - 5	I	= 2 , Type of substructure. (Iroute)

B) NUMBER OF VALUES TO BE CHANGED

LINE	COLS.	TYPE	
1	1 - 5	I	Number of values to be changed. (Noval)

C) UPDATE OF VALUES

LINE	COLS.	TYPE	
1	1 - 5	I	Parameter number specifying the parameter to be changed. (Nalt)
	6 - 15	F	New value of parameter. (Value) Input the number as the same type as input for IROUTE = 1. i.e. integer or real.

Nalt = 1 , Number of repeated substructures. (Nv)
= 2 , Number of divisions along the girder. (Idivl)
= 3 , Radius of reference line of bridge. -ft- (Rref)
= 4 , Spacing of diaphragms. -ft- (Sdia)
= 5 , Thickness of the web. -in- (Girdwd)
= 6 , Diaphragm bracing configuration and properties.

Additional input is required and the second space in line two should be left blank.

The following input is required for new diaphragms.

LINE	COLS.	TYPE	
1	1 - 5	I	Diaphragm bracing configuration for the beginning of the substructure. (Ibrace(1)) = 1, For X-bracing with top horizontal member. = 2, For X-bracing without top horizontal member. = 3, For K-bracing.
	6 - 10	I	Diaphragm bracing configuration for end of substructure. As above. (Ibrace(2))
	11 - 15	I	Number of different types of diaphragm bracing members. (Ndifbr)

The following should be input for each different brace.

LINE	COLS.	TYPE	
1	1 - 10	F	Half the area of the diaphragm. -in^2- (Adia)
	11 - 20	F	Half the moment of inertia for the brace about the horizontal axis. -in^4- (Diy)
	21 - 30	F	Half the moment of inertia for the brace about thevertical axis. -in^4- (Diz)
	31 - 40	F	Half the torsional moment of inertia. -in4- (Torj)
	41 - 45	I	Description index for locating the properties of the diaphragms. (Ibr)

The number corresponds to the following description :

= 1, For diagonal member of beginning brace.
= 2, For bottom member of beginning brace.
= 3, For top member of beginning brace.
= 4, For diagonal member of ending brace.
= 5, For bottom member of ending brace.
= 6, For top member of ending brace.

	46 - 50	I	Description index for second brace member
	51 - 55	I	Description index for third brace member
	56 - 60	I	Description index for fourth brace member
	61 - 65	I	Description index for fifth brace member
	66 - 70	I	Description index for sixth brace member.

Nalt = 7 , Bottom horizontal bracing. New values for the dimensions of the braces should be input as the following:

LINE	COLS.	TYPE	
1	1 - 10	F	Full area of diaphragm for bottom horizontal bracing. -in^2- (Adia)
	11 - 20	F	Full moment of inertia for bottom horizontal bracing about the horizontal axis. -in^4- (Diy)
	21 - 30	F	Full moment of inertia for bottom horizontal bracing about the vertical axis. -in^4- (Diz)

LINE	COLS.	TYPE	
	31 - 40	F	Full torsional moment of inertia for bottom horizontal bracing. (Torj)
		Nalt	= 8 , Number of bottom braces in a cross-section. This number can be input as zero and no bottom braces will be added.
			= 9 , Bottom bracing geometry. The following lines must be input for each bottom brace. The second value in this row should be zero.
1	1 - 5	I	Girder number in which the brace starts. (Ngsbot)
	6 - 10	I	Girder number in which the brace ends. (Ngebot) This must be input for each bottom brace. The start of the substructure is the cross-section where the node numbering begins.
		Nalt	= 10 , Flange dimensions. The following must be input for each girder. The second value in this row should be zero.
1	1 - 10	F	Thickness of top flange. -in- (Tft)
	11 - 20	F	Width of top flange. -in- (Bft)
	21 - 30	F	Thickness of bottom flange. -in- (Tfb)
	31 - 40	F	Width of bottom flange. -in- (Bfb)
		Nalt	= 11, Modulus of elasticity of concrete in the x-axis direction. -ksi- (Ec(1)))
			= 12 , Modulus of elasticity of concrete in the y-zxis direction. -ksi- (Ec(2))
			= 13, Poisson's ratio for concrete due to stress in the x-axis direction. (Poisc)
			= 14, Density of concrete. -kcf- (Densc)
			= 15, Modulus of elasticity of steel. -ksi- (Est)
			= 16, Poisson's ratio for steel. (Poisst)
			= 17, Density of steel. -kcf- (Densst)
			= 18, Curb load of outside overhang. -k/ft- (Curbld(1))
			= 19, Curb load of inside overhang. -k/ft-(Curbld(2))
			= 20, Surface slab load. -ksf- (Surfld)

There are several values of Nalt that may be entered on this line. For values of Nalt of 1-5, 8, and 11-20, only a single entry for the corresponding value (columns 6-15) is required, thus requiring a single line of input to change a value. If all values of Nalt are of this type, for example, then Noval lines of input are required. For other values of Nalt (i.e., 6, 7, 9, and 10), more than one line is required to make a change. In this case, Nalt is entered as for the other cases in columns 1-5 and then values must be entered on additional lines that follow. For Nalt of 6, two additional lines are required, while for Nalt of 7, 9, and 10, one additional line is required. When additional lines are required, the second entry (columns 6-15) on the line containing Nalt should be left blank. In what follows, the various values of Nalt that can be entered are described and the corresponding values that must be entered are given. After the Noval changes are made, additional loads may be entered, as described in Part D that follows. The entry of additional loads is terminated by placing a zero in column one and thus does not require an Nalt value.

D) ADDITIONAL LOADS

Loads are input by elements. Each line of data is superimposed on loads that may exist for that member.End this section by placing a zero in column one. Signs of additional loads should follow element axis as shown in Fig. 2.3a, b and c.

LINE	COLS.	TYPE	
1	1 - 5	I	DOF of forces for this element (i.e., 1 - 6) (Idof)
	6 - 10	I	Substructure where this element is located. (Jsubld)
	11 - 15	I	Element number. (Kelno)
	21 - 30	F	Force at node I of this element. (P(1))
	31 - 40	F	Force at node J of this element. (P(2))
	41 - 50	F	Force at node K of this element. (P(3))
	51 - 60	F	Force at node L of this element. (P(4))

2.4.4 **IROUTE = 3**

This choice of substructure is used for a slight skew of a support. The distinciton is made between a slight skew and a severe skew as shown in Figure 2.8. It is up to the judgment of the user but the category used in this program is that a slight skew is one which the finite element mesh is slightly skewed to accomodate the supports. A severe skew is one in which the mesh is not distorted. A general quantitative definition is that a slight skew is less than about 20 degrees.

This type of substructure is a very convenient one to represent a slightly skewed mesh with or without supports but it requires some modeling of finite elements. Either the end diaphragm or interior nodes can be skewed and supported. The distance along the reference line and skew angle of the transverse row of nodes is input by the user. In this way, the user custom fits the mesh to a slight skew. If the interior nodes are supported then there are no diaphragms over the support. The diaphragms follow the sides of the substructure. This has its advantages in that radial diaphragms are possible with slightly skewed supports. The input for IROUTE = 1 or IROUTE = 2 up to the point of additional loads must immediately precede the following lines.

A) SUBSTRUCTURE INDEX

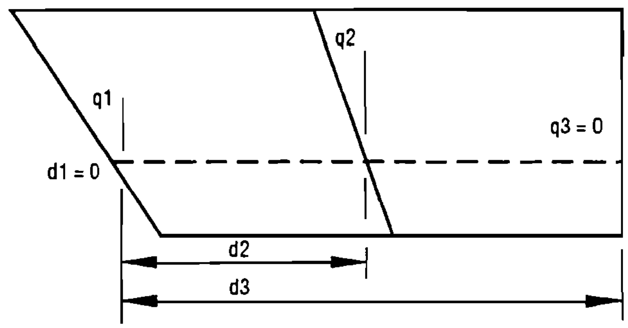
LINE	COLS.	TYPE	
1	1 - 5	I	= 3 , Type of substructure. (Iroute)

B) INPUT DATA FOR EITHER SECTIONS 2.4.2 OR 2.4.3

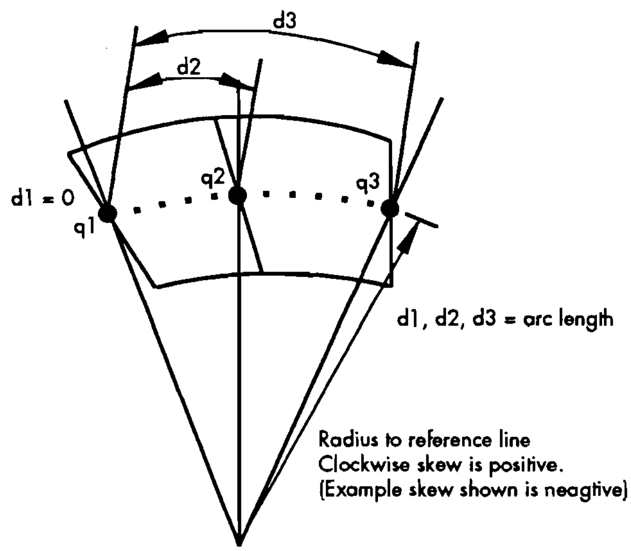
Any additional loads still must be input at the end of this section.

C) SLIGHT SKEW SUPPORT PLACEMENT

LINE	COLS.	TYPE	
1	1 - 5	I	Number of nodes along the bottom of the girder until it is supported. (Nsbc) Assumed the same for all girders.



a) Straight substructure Clockwise skew is positive.
(Example skew shown is negative)



b) Curved Substructure

Figure 2.17 Substructures for Iroute = 3

D) SLIGHT SKEW MESH GEOMETRY

The following is input for each transverse rows of nodes at the bottom flange locations.Caution should be shown if the mesh size was automatically determined. For further information see Figure 2.17.

LINE	COLS.	TYPE	
1	1 - 10	F	Reference line distance until transverse row of nodes.-ft- (Dsks)
	11 - 20	F	Skew angle of interior row of nodes. Positive is measured as clockwise. The angle is measured from the radius through the node. -degrees- (Dske)

E) ADDITIONAL LOADS

Loads are input by elements. Each line of data is superimposed on loads that may exist for that member. End this section by placing a zero in column one. Signs of additional loads should follow element axis as shown in Fig. 2.3a, b and c.

LINE	COLS.	TYPE	
1	1 - 5	I	DOF of forces for this element (i.e., 1 - 6) (Idof)
	6 - 10	I	Substructure where this element is located. (Jsubld)
	11 - 15	I	Element number. (Kelno)
	21 - 30	F	Force at node I of this element. (P(1))
	31 - 40	F	Force at node J of this element. (P(2))
	41 - 50	F	Force at node K of this element. (P(3))
	51 - 60	F	Force at node L of this element. (P(4))

2.4.5 **IROUTE = 4**

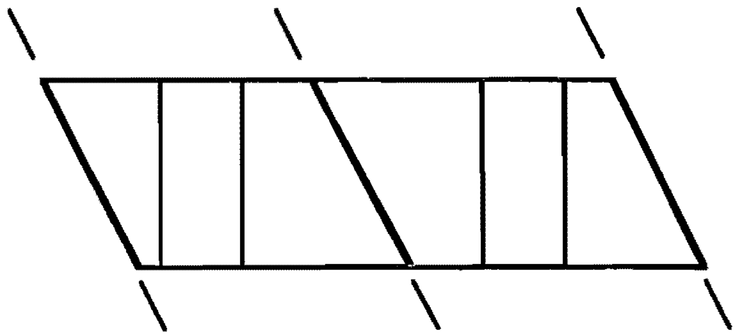
This choice of substructure is used for severly skewed supports. Unlike the slight skew of supports, the mesh for this substructure is not distorted. The assumptions are made that the diaphragms are intersected at support locations over the girders. This assumption allows for many configurations for a skewed support as shown in Figures 2.8, 2.9 and 2.19. Depending on the arrangement of the diaphragms, the supports can be located at the ends of the substructure or at interior nodes as well. The assumption is still made that a substructure is defined as the part between two diaphragms. When there are many diaphragms between the first and last support, this type of substructure must be used until all girders are supported. As for IROUTE = 3, the data for either IROUTE = 1 or IROUTE = 2 up to the data on additional loads must immediately precede the following input.

A) SUBSTRUCTURE INDEX

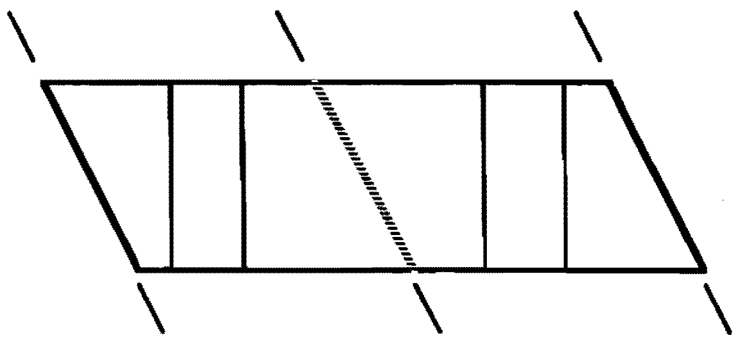
LINE	COLS.	TYPE	
1	1 - 5	I	= 4 , Type of substructure. (Iroute)

B) INPUT DATA FOR EITHER SECTIONS 2.4.2 OR 2.4.3

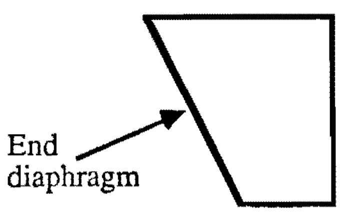
Any additional loads still must be input at the end of this section.



a) Slight skew bridge with non-radial diaphragms
and diaphragm over interior support

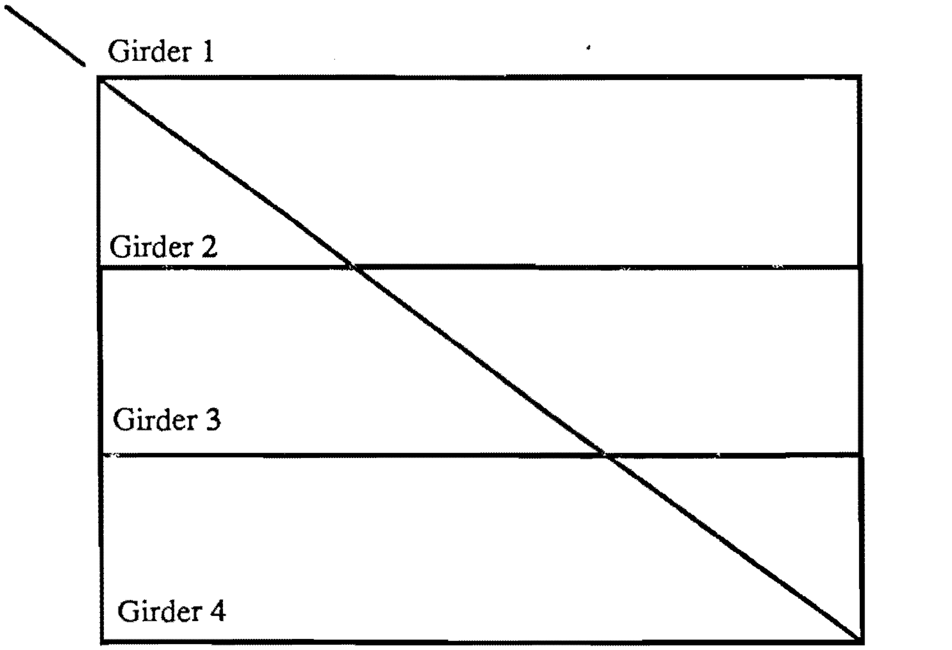


b) Slight skew bridge with interior radial diaphragms
and no diaphragm over interior support

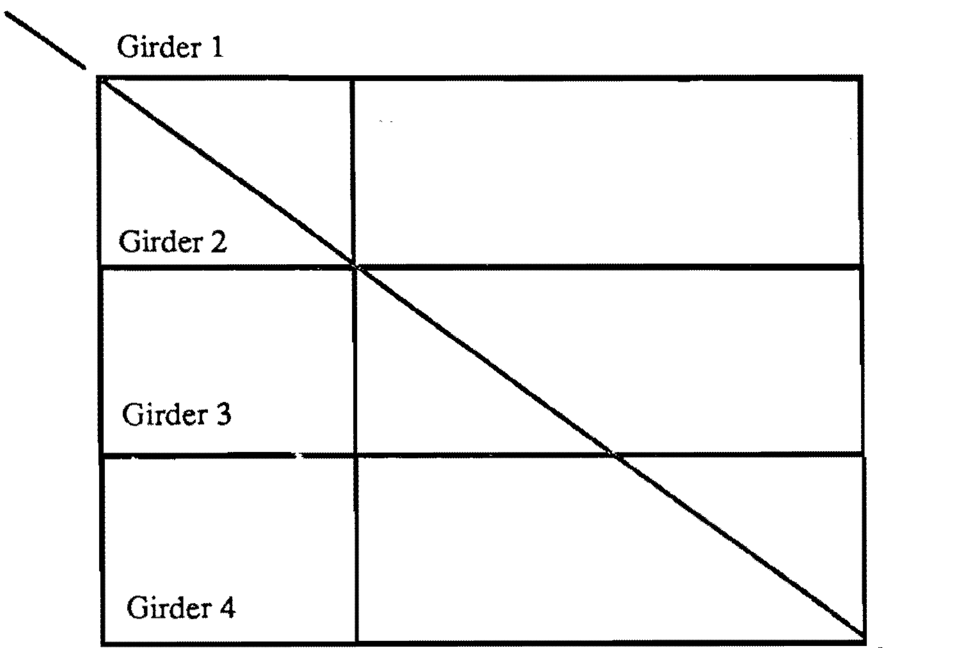


c) End support slightly skewed
end diaphragms not radial

Figure 2.18 Examples of Iroute = 3



a) One substructure for severely skewed support



b) Two substructures for severely skewed supports

Figure 2.19 Example for Iroute = 4

C) SEVERE SKEW SUPPORT PLACEMENT

LINE	COLS.	TYPE	
1	1 - 5	I	Girder number which is supported along the starting diaphragm.(Nge)
	6 - 10	I	Girder number which is supported along the ending diaphragm. (Ngs)

D) ADDITIONAL LOADS

Loads are input by elements. Each line of data is superimposed on loads that may exist for that member. End this section by placing a zero in column one. Signs of additional loads should follow element axis as shown in Fig. 2.3a, b and c.

LINE	COLS.	TYPE	
1	1 - 5	I	DOF of forces for this element (i.e., 1 - 6) (Idof)
	6 - 10	I	Substructure where this element is located. (Jsubld)
	11 - 15	I	Element number. (Kelno)
	21 - 30	F	Force at node I of this element. (P(1))
	31 - 40	F	Force at node J of this element. (P(2))
	41 - 50	F	Force at node K of this element. (P(3))
	51 - 60	F	Force at node L of this element. (P(4))

2.4.6 IROUTE = 5

This substructure is used when the geometry of the bridge does not lend itself to one of the previously specified substructures. This substructure is then placed after the previous substructure and all renumbering of the master nodes is done automatically. Care must be taken in numbering the nodes. Any nodes which border on a substructure that is generated must be numbered consecutively from lowest to highest. Also, all nodes specified must have elements connected to them. All live loads due to trucks or lanes must be input for this substructure.

A) SUBSTRUCTURE INDEX

LINE	COLS.	TYPE	
1	1 - 5	I	= 5 , Type of substructure. (Iroute)

B) CONTROL CARDS FOR THIS SUBSTRUCTURE

LINE	COLS.	TYPE	
1	1 - 5	I	Number of elements for this substructure. 200 maximum. (Noel)
	6 -10	I	Number of points for this substructure. 162 maximum. (Nopt)
	11 -15	I	Number of times this substructure is repeated consecutively. 16 Maximum. (Nv)
	16 - 25	F	Reference radius. -ft- See Figures 2.4, 2.20, and 2.21.

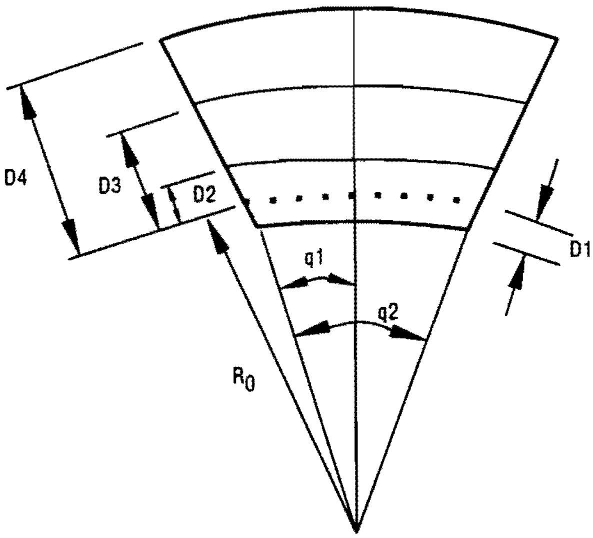


Figure 2.20 Cylindrical coordinates

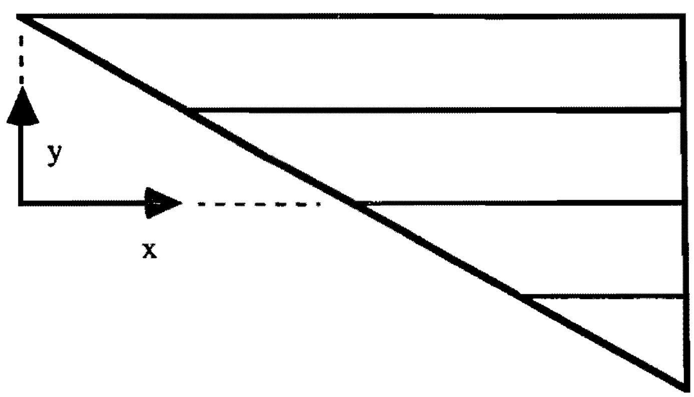


Figure 2.21 Cartesian coordinates.

C) ELEMENT NODE NUMBERS FOR THIS SUBSTRUCTURE

One line per element is required for this input. Element node numbers should be numbered counter-clockwise as shown in Figures 2.3a, 2.3b and 2.3c. For elements with less than four nodes, leave the remaining node numbers blank.

LINE	COLS.	TYPE	
1	1 - 5	I	Element nodal point I
	6 - 10	I	Element nodal point J
	11 - 15	I	Element nodal point K
	16 - 20	I	Element nodal point L

D) CYLINDRICAL COORDINATES

Used if the Reference radius input above is larger than zero. One line per point is required as described below. If the reference radius is input as zero, skip this section. See Figures 2.4b and 2.20 for further information.

LINE	COLS.	TYPE	
1	1 - 5	I	Point number.
	11 - 20	F	Distance, D , form Reference circle. -ft-
	21 - 30	F	Angle in degrees, A , from Y-axis.
	31 - 40	F	Distance , Z , from X-Y plane.

E) CARTESIAN COORDINATES

Used if the Reference radius is input as zero. One line per point is required as described below. See Figures 2.4a and 2.21 for further information.

LINE	COLS.	TYPE	
1	1 - 5	I	Point number.
	11 - 20	F	Cartesian X- coordinate. -ft-
	21 - 30	F	Cartesian Y- coordinate. -ft-
	31 - 40	F	Cartesian Z- coordinate. -ft-
	41 - 45	F	Component in X- direction of unit vector *S1* *
	46 - 50	F	Component in Y- direction of unit vector *S1* **
	51 - 55	F	Component in Z- direction of unit vector *S1*. **
	56 - 60	F	Component in X- direction of unit vector *S2*. **
	61 - 65	F	Component in Y- direction of unit vector *S2*. *
	66 - 70	F	Component in Z- direction of unit vector *S2* **

* is usually input as 1.0

** is usually input as 0.0

NOTE: *S1* and *S2* are surface coordinates and are described in detail in CFHR Report 23 - 3F and have the same meaning here.

F) ELEMENT TYPES AND MATERIAL PROPERTIES

The plate element can be either a triangle or a quadrilateral with the correct input for nodes and triangles. The distinction is made between a flange and a beam element for ease of input and output only. The flange element is assumed rectangular and the inertias are calculated from the thickness and width. The output for the flange elements are in terms of stresses and the output for the beam elements are in terms of forces. One line per element is required as described below. Use the same order of elements numbered in section 2.4.6, part C.

Web Elements

LINE	COLS.	TYPE	
1	1 - 5	I	Number of nodes per element.
	6 - 10	I	Number of sub triangles of quadrilateral.
	11 - 15	I	= 0, if CLST is used for membrane stiffness.
			= 1, if QM5 is used for membrane stiffness.
	16 - 20	I	= 0, type of plate element.
	21 - 30	F	Modulus of elasticity in x-axis direction. -ksi-
	31 - 40	F	Modulus of elasticity in y-axis direction. -ksi-
	41 - 50	F	Thickness of element. -in-
	51 -60	F	Poisson's ratio for stress in x-axis direction.
	61 - 70	F	Element unit weight. -kcf-
			This should be input as negative for dead loads.

Slab Elements

LINE	COLS.	TYPE	
1	1 - 5	I	Number of nodes per element.
	6 - 10	I	Number of sub triangles of quadrilateral.
	11 - 15	I	= 0, if CLST is used for membrane stiffness.
			= 1, if QM5 is used for membrane stiffness.
	16 - 20	I	= 1, type of plate element.
	21 - 30	F	Modulus of elasticity in x-axis direction. -ksi-
	31 - 40	F	Modulus of elasticity in y-axis direction. -ksi-
	41 - 50	F	Thickness of element. -in-
	51 -60	F	Poisson's ratio for stress in x-axis direction.
	61 - 70	F	Element unit weight. -kcf-
			This should be input as negative for dead loads.

Beam Elements

LINE	COLS.	TYPE	
1	1 - 5	I	= 2, Number of nodes per element.
	16 - 20	I	= 0, Type of beam element.
	21 - 30	F	Modulus of elasticity. -ksi-

LINE	COLS.	TYPE	
	31 - 40	F	Cross-sectional area. in2-
	41 - 50	F	Shear Modulus multiplied by torsional moment of inertia. -k in2- (GJ)
	51 - 60	F	Element unit weight. - k / ft -This should be input as negative for dead loads.
	61 - 70	F	Inertia -Y. -in4-
	71 - 80	F	Inertia -Z. -in4-

Flange Elements

LINE	COLS.	TYPE	
1	1 - 5	I	= 2, Number of nodes per element.
	16 - 20	I	= 1, Type of beam element.
	21 - 30	F	Modulus of elasticity. -ksi-
	31 - 40	F	Cross-sectional area. -in2-
	41 - 50	F	Shear Modulus multiplied by torsional moment of inertia. -k in2- (GJ)
	51 - 60	F	Element unit weight.- k / ft - This should be input as negative for dead loads.
	61 - 70	F	Thickness of flange. -in-
	71 - 80	F	Width of flange. -in-

Rigid Link Elements

LINE	COLS.	TYPE	
1	1 -5	I	= 2, Number of nodes per element.
	16 - 20	I	= 2, Type of beam element. All the remaining properties are internally set in the program. The rigid link connects the top flange to the concrete slab and functions as a shear stud.

G) NUMBER OF ELEMENTS BELONGING TO EACH SUBSTRUCTURE

LINE	COLS.	TYPE	
1	1 - 5	I	Number of elements for substructure 1.
	6 -10	I	Number of master nodes for substructure 2.
	" "		
	" "		
	76 - 80	I	Number of master nodes for substructure 16. The first value should be the number of elements and all additional values should be the <u>negative</u> of the number of master nodes for the element. See section 2.5.16 and examples in Chapter 4 for further information.

H) CONTROL CARDS FOR THIS SUBSTRUCTURE

LINE	COLS.	TYPE	
1	1 - 5	I	Number of master nodes for this substructure.

LINE	COLS.	TYPE	
	6 - 10	I	Number of nodes having boundary conditions excluding master nodes.
	11 - 15	I	Number of nodes having boundary conditions excluding master nodes which either prescribed displacements or spring stiffnesses are input.
	16 - 20	I	= 1, Master nodes are renumbered for this substructure.

I) SUBSTRUCTURE MASTER NODES

These are the substructure nodes which will be renumbered to master nodes. For data-generated substructures, these nodes are at diaphragm location. Substructure master nodes must be input in increasing order.

LINE	COLS.	TYPE	
1	1 - 5	I	First master node for this substructure.
	6 - 10	I	Second master node for this substructure.
	" "		
	" "		
	76 - 80	I	Sixteenth master node for this substructure. Use more lines if needed.

J) NODES HAVING BOUNDARY CONDITIONS EXCLUDING MASTER NODES

One line is required for each node with boundary conditions.

LINE	COLS.	TYPE	
1	1 - 5	I	Node number.
	6 - 10	I	Boundary condition code* for x- dir. translation.
	11 - 15	I	Boundary condition code* for y- dir. translation.
	16 - 20	I	Boundary condition code* for z- dir. translation.
	21 - 25	I	Boundary condition code* for x- dir. - rotation.
	26 - 30	I	Boundary condition code* for y- dir. - rotation.
	31 - 35	I	Boundary condition code* for z- dir. - rotation.

*, B.C. code = 0 for unrestrained.
= 1 for restrained.
= 2 non-zero displacement to be specified.
= 3 elastic spring to be specified.

K) SPECIFIED BOUNDARY CONDITIONS FOR NON-MASTER NODES

One line is required for each node with at least one boundary condition code greater than one.

LINE	COLS.	TYPE	
1	1 - 10	F	Action for X direction translation.
	11 - 20	F	Action for Y direction translation.

LINE	COLS.	TYPE	
	21 - 20	F	Action for Z direction translation.
	31 - 40	F	Action for X direction rotation.
	41 - 50	F	Action for Y direction rotation.
	51 - 60	F	Action for Z direction rotation.

L) INPUT REQUIRED FOR DATA GENERATOR PROGRAM

LINE	COLS.	TYPE	
1	1 - 5	I	Number of nodes for a data-generated cross-section at a diaphragm location. (Nxnd)
	6 - 10	I	Number of master nodes with boundary conditions for this substructure. (Nbc5)
	11 - 15	I	Number of master nodes with prescribed displacements or support springs. (Lpgo5)
	16 - 25	F	Reference line length for substructure. -ft- Used in global placement of truck and lane loads. (Dcent5)

M) BOUNDARY CONDITIONS FOR MASTER NODES FOR THIS SUBSTRUCTURE

LINE	COLS.	TYPE	
1	1 - 5	I	Master node number. This node number is the local node which will be renumbered to the correct master node.
	6 - 10	I	Boundary condition code* for x- dir. translation.
	11 - 15	I	Boundary condition code* for y- dir. translation.
	16 - 20	I	Boundary condition code* for z- dir. translation.
	21 - 25	I	Boundary condition code* for x- dir. - rotation.
	26 - 30	I	Boundary condition code* for y- dir. - rotation.
	31 - 35	I	Boundary condition code* for z- dir. - rotation.

*, B.C. code = 0 for unrestrained.
= 1 for restrained.
= 2 non-zero displacement to be specified.
= 3 elastic spring to be specified.

N) SPECIFIED BOUNDARY CONDITIONS FOR MASTER NODES

One line is required for each node with at least one boundary condition code greater than one.

LINE	COLS.	TYPE	
1	1 - 10	F	Action for X direction translation.
	11 - 20	F	Action for Y direction translation.
	21 - 20	F	Action for Z direction translation.
	31 - 40	F	Action for X direction rotation.
	41 - 50	F	Action for Y direction rotation.
	51 - 60	F	Action for Z direction rotation.

O) ADDITIONAL LOADS

Loads are input by elements. Each line of data is superimposed on loads that may exist for that member. End this section by placing a zero in column one. Signs of additional loads should follow element axis as shown in Fig. 2.3a, b and c.

LINE	COLS.	TYPE	
1	1 - 5	I	DOF of forces for this element (i.e., 1 - 6) (Idof)
	6 - 10	I	Substructure where this element is located. (Jsubld)
	11 - 15	I	Element number. (Kelno)
	21 - 30	F	Force at node I of this element. (P(1))
	31 - 40	F	Force at node J of this element. (P(2))
	41 - 50	F	Force at node K of this element. (P(3))
	51 - 60	F	Force at node L of this element. (P(4))

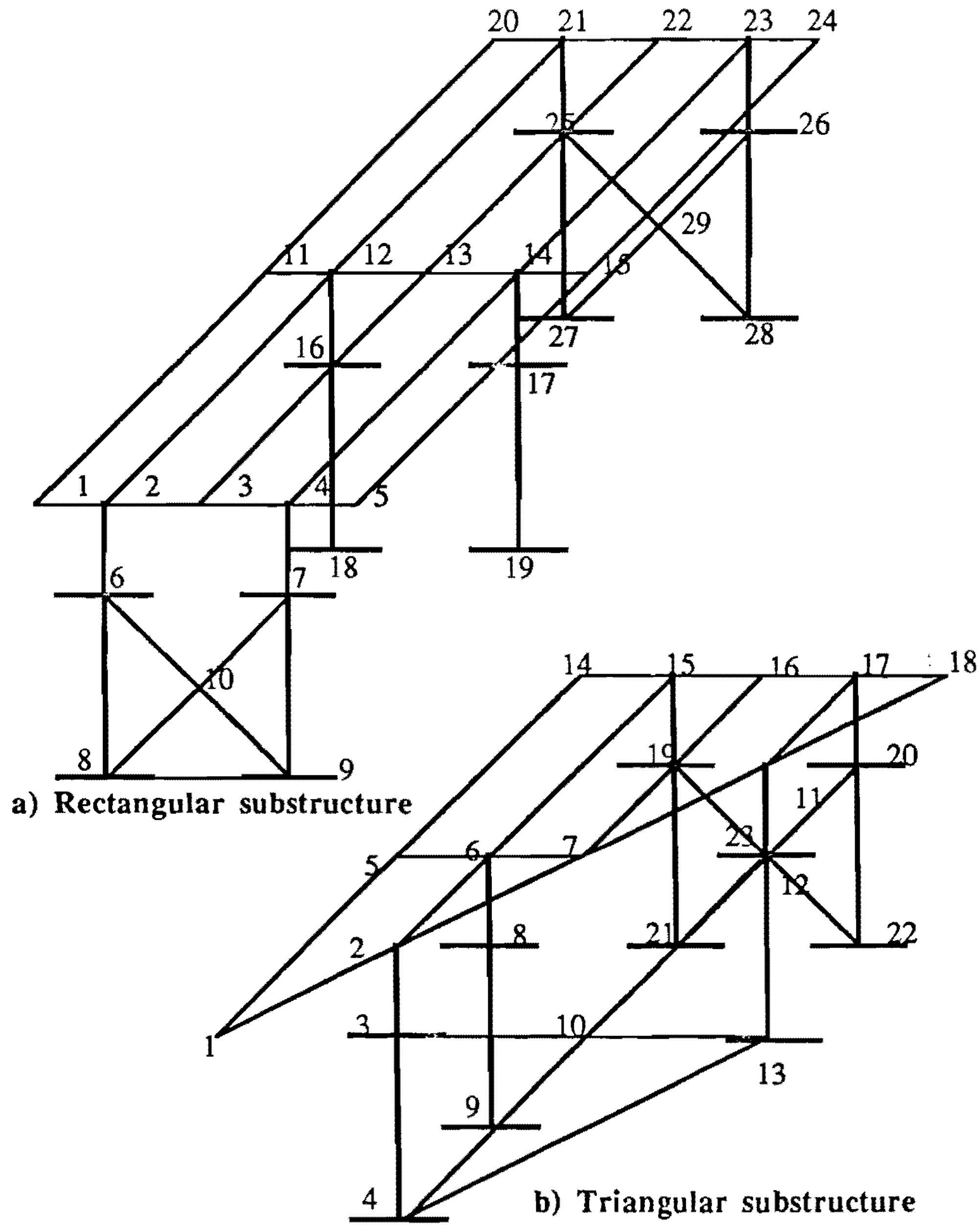


Figure 2.22 Examples of Iroute = 5

2.5 ADDITIONAL REMARKS

The following commentary should be used along with the documentation to aid in the preparation of the input data. The examples in Chapter 4 can also help in understanding the documentation.

2.5.1 Control cards

The data generator automatically adds the half-diaphragms on the end of the bridge if the end substructures are not input by hand. (Iroute = 5). The number of type of substructures (Nogp) is automatically incremented and two new substructures are defined. The counting of the number of substructures for the support conditions and the erection process must not take these substructures into account. The number of substructures between supports and for erection purposes should consider only the type of substructures that are between diaphragms inclusive. For further clarification see Figure 2.8 and examples in chapter four.

The plotting routine (Ipl) will plot all the elements as viewed in both the XY and YZ planes. The node numbering will be only for the first diaphragm cross-section in the YZ plane and the slab nodes in the XY plane. This was done to avoid congestion on the plot. The plots of the concrete slab stresses by slab node are also available if IPL equals 2. This is useful for a visual description of the slab stresses as well as the fact that the slab stresses are only printed out at the midpoint of the slab elements.

The check prints (Idef) option outputs information that is mostly relative to the data generator and related programs. The most important option is how deflections should be printed.: i.e., Idef = 0,1 or 2. If Idef = 1 then deflections are printed out by master node. In most all cases this is at diaphragm locations. If Idef = 2 then deflections are also printed out by element. For large jobs there are a lot of elements and the output of deflections will entail a lot more output. Many times the deflections by master nodes may be sufficient. If there are any supports for local nodes as opposed to master nodes, then Idef needs to be set equal to 2 to determine these reaactions.

The size of the mesh can be automatically generated. This should be used with caution since the program may not generate a good mesh in all cases. Since the web depth is fixed, the mesh generator determines the number of divisions along the reference line that will most likely give a square web element. Knowing the longitudinal size of the elements, the mesh generator determines the number of transverse divisions so that the concrete slab elements are most nearly square. The mesh generator is done only once for the bridge so, for best results, the spacing of the diaphragms should not change too much longitudinally across the bridge. Also for a better mesh size, the reference line should be located between the first and last girder of the bridge. It may be recommended that the automatic mesh size generator be used in preliminary designs and the user should determine the mesh size for the final computer runs.

2.5.2 Support Conditions

A couple of examples of the number of substructures per span are shown in Figure 2.8. A cantilever is counted as a span. For supports done with IROUTE = 5, do not count that support as dividing a span. If all supports are input with IROUTE = 5, Then input a 1 for Nspan and a number greater than the total number of substructures for Nsubbc. The number of substructures per span are used as a counting procedure to place the supports in the data generator. When the supports are not generated then this special input is needed.

For data-generated substructures, the boundary conditions are internally set in the program. Only one support is restrained in the horizontal planes. All other supports are supported in the vertical direction only. The default support stiffness is 1.0 x 10^{50} kips / inch. Other vertical support stiffnesses and initial displacements can be input for each support. In this way, the effect of support settlements, rotations and different spring stiffnesses can easily be analyzed. Support rotations can be analyzed assuming a linear displacement for vertical displacements for the girders. The choice of boundary conditions for Iroute = 5 is up to the user. Spring stiffnesses and displacements can also be input for all these supports.

2.5.3 Erection process

The erection program uses superpostion to add up stresses for the construction stages. The old stresses are written onto tapes and the option is given to add new stresses to the old stresses (Ircsum). The assumption was made that during construction, the steel girders being placed already have the diaphragms and the bottom bracing attached between them. Diaphragms and bottom bracing can be attached between two girders in later stages by using the negative number of the girders to which they are being attached. The diaphragms and the bottom bracing are assumed to be attached during the same stage however.

The erection of the bridge is actually done by analyzing the same bridge geometry but modifying the material properties. Initially, the steel modulus is set equal to a small number and the density of the steel is set equal to zero for girders not yet erected. At the erection of the girder, both the density and the elastic modulus are set equal to the proper value. All subsequent erection stages have the correct modulus but the denstiy of that girder is zero. Thus the theory of superposition of loads is ensured. The placement of the concrete must be done by hand in the same manner. This was done so that full flexibility can be accomplished in analyzing the construction process. The user can set the elastic modulus of concrete equal to different values as the concrete cures and new pours are made on the bridge. However, the load of the concrete should only be included in one stage. The erection process prints the sum of the stresses along with the new stage stresses. The program does not use second order analysis by connecting a new girder to the deformed shape of a previously erected girder. The load is placed on the bridge after the erection is finished. It is thus assumed that a continuous connection occurs before the load is placed. The user should be cautioned that the crictical point of loading may occur when the connection is pinned and pinning a connection is not possible with this program.

The same procedure for adding erection stresses can be used to sum the stresses due to different loadings with the only difference being that no erection stages are input (Irec = 0). In this way, dead load and live load stresses can be summed by the computer. Load factors for the live load can be input and load factors for the dead load can be taken account of with the proper densities.

2.5.4 Bridge Geometry

If the mesh size generator is not used, then the number of divisions between girders and diaphragms is up to the user. Care should be taken in keeping the aspect ratio of the web element around 2.0. The aspect ratio of the overhang element can probably be around 4.0 since it has much less structural importance. The program calculates the aspect ratios of all four node plate elements and prints out the maximum and the average aspect ratios. The aspect ratio for any element probably should not exceed 5.0 and the average aspect ratio of all the plate elements probably should not exceed 4.0. Since there are no nodes between the flanges for the web element, the aspect ratio might be most critical for this element. The number of divisions should be decided upon considering both aspect ratio and computer solution time.

The reference radius is arbitrarily input by the user. A positive radius is defined as one which curves to the right as one goes from left to right on the bridge. A negative radius curves to the left. A straight substructure is defined as having a zero radius even though the radius is actually infinite. The spacing of the diaphragms are defined along this reference line. The thickness of the concrete is assumed constant for the bridge. Wearing surface should be input as a surface load. The outer overhang is the one next to girder one as shown in Figure 2.4b.

2.5.5 Girder

The location of the girders is done with respect to the reference line. Girder one is input first and each girder is input in increasing order. Girder one is defined as the girder having the largest radius. For straight substructures, girder one is on the left as one goes from left to right on the bridge. The web depth is assumed constant for all girders at all locations. The web thicknesses can be changed at different substructures but the web is assumed within a substructure. Average top and bottom flange thicknesses are input to set the location of the flanges for the structure and does not have anything to do with structural element thicknesses or properties. The properties of the beam elements are lumped at these locations as shown in Figure 2.6. This allows much easier input and much better advantage can be taken of repeated substructures.

2.5.6 Diaphragm

Three types of diaphragms are provided for by the data generator as shown in Figure 2.11. The bracing configuration can be different at each end of the substructure by changing the bracing index. A substructure can have K-bracing at the start and X-bracing at the end of the substructure. Only the number of different types of bracing members have to be input. The properties of each member are then input. Half the area, inertias and torsional moment of inertias are used so that the sum of these properties will be correct when the next substructure is placed adjoining it. A slight drawback to this system is the erection sequence when the next diaphragms are not added. In this case, the stiffness of the brace at the free end is only half of its full value. A description index is used to determine the location of the properties of the diaphragm members. The diagonals are 1 and 4, bottom horizontals are 2 and 5 and top horizontals are 3 and 6 as shown in Figure 2.11.

2.5.7 Bottom Horizontal Braces

The bottom braces are connected in between the bottom flanges at diaphragm locations. These braces are all assumed to be the same type of member. The full properties are input since these braces are not adjoining another substructure. The braces can be can be arranged in X-bracing or single bracing may be different between any two girders as shown in Figure 2.12. If the bracing configuration changes from one substructure to the next , then a new substructure type needs to be defined. The user could make the choice of modeling the bracing as X-bracing and then only one substructure type needs to be defined.

2.5.8 Material Properties

The material properties for concrete and steel apply to all the respective elements in the substructure. All concrete elements in the substructure have the same moduli, poisson's ratios and density. The web, flange, and beam elements also have the same properties. The properties of the rigid link are internally set. The densities of the elements must be input as a positive value for gravity loads. The program then switches the sign to match the global sign convention.

The slab can be modeled with orthotropic elements if different elastic moduli are used for the two directions. Since the x-direction parallels the side ij in quadrilateral elements and this side is mostly tangent to the radius, the elastic modulus in the x-direction corresponds closely to longitudinal bending. A different elastic modulus can be input in a perpendicular direction to side ij. One poisson ratio is input corresponding to the transverse displacement due to a stress in the x-direction. The poisson ratio in the other direction is internally calcluated based on elasticity equations. Also, the shear modulus is internally calculated based on approximations for an orthotropic material (5). The equation of the shear modulus for an orthotropic material was taken as the following:

$$\mathbf{G} = \frac{\left(E_1 \times E_2\right)^{\frac{1}{2}} - V_{21} \times E_2}{\left(1 - V_{21} \times V_{12}\right) \times 2}$$

The most common application of orthotropic plates is over negative moment regions. Also, orthotropic plates can be used for the slab load on the bare girder for dead load case one. When the longitudinal concrete is considered ineffective, as in negative moment regions, the slab load is still carried transversely to the girders as a one-way slab. Thus, the elastic modulus in the x-direction can be set equal to a small number and the modulus in the transverse direction can be set equal to the correct value. Since the poisson's ratio loses meaning in one-way action as with the case with very different elastic moduli, it is suggested the the poisson's ratio be set equal to zero for these cases.

2.5.9 Slab Loads

Different curb loads are possible for the inside and outside curbs. The curb loads are smeared on the respective overhang elements as an increase in density of the concrete slab. The curb load should include all loading expected on the overhang such as curbs, sidewalks, railings or pedestrians. The surface load is applied throughout the concrete elements as an increase in density. No patterned surface load is possible for one substructure, but it is possible to change the surface load on a new substructure type. Negative loads are downward.

2.5.10 Truck Loads

The input for the truck loading is very convenient. The global centroid location of the truck is specified along with an impact factor, name and direction. The location of the wheels are referenced from the centroid of the truck as shown in Figures 2.13 and 2.14. If the same truck is placed at a different location in the bridge, then only the global location needs to be specified again. The same local wheel locations are used if the variable IBID is set equal to one. It is thus advantageous to place all alike trucks in order so that the least amount of input is required.

The reference line of the bridge at the truck reference point should be the same number as the reference radius for that substructure. This number can be positive, zero or negative. The truck is assumed to be straight going around a curve so that the wheel will not be exactly the same distance off the reference arc. If the radius changes in the middle of the truck, then two trucks must be input with the input for each wheel load for the respective radii. This does not apply to the end of the bridge in which any wheels off the end of the bridge are ignored. The program also ignores wheels off the side of the bridge. In this case, the program prints out a warning statement and the number of wheels off the side of the bridge.

If a truck is placed on a substructure that is input by hand (Iroute=5), then these wheel loads must also be input by hand. The program determines the load to each node with the areas of each quadrilateral as shown in Figure 2.14b. This loading is applied in a way general enough to be used for skewed meshes which are in a curve. Because upward loads are positive, *all* truckloads must be entered as negative.

2.5.11 **Lane loads**

The lane loading is input in much the same way as the truck loads. The lanes are assumed to follow the radius at the centroid. If the radius changes in between the lane load, then two lane loads must be input. The concentrated loads are applied at a single point much the same way as a wheel load. The user must define an equivalent concentrated load if a concentrated live load is desired. Only the upper left and lower right corners are input for the lane loads. The program divides the distributed loads into many concentrated loads and places these loads on the respective elements in much the same way the truck loads are placed. For lane loads over skewed meshes or end supports the lane load should be place over the entire area desired and the program will disregard any lane loads not over elements. Negative lane loads are downward.

2.5.12 **Additional Loads**

Any loading can be superimposed on any element by using this input. The degree of freedom for the forces correspond to the local axis of orientation of the element. This is not the global degree of freedom of the substructure. Thus, if there is a wind load on the web then the loads would be in the z - direction on the element and DOF = 3. If there is more than one loading DOF on the element then a line for each DOF must be input. As an example, if there are vertical loads and two moments then three lines of input are needed.

The user inputs the substructure where the element is located. The substructure number corresponds to the repeated substructure number. If there is a load on the third substructure out of five repeated substructures then the substructure where this element is located is three. This is independent of where these substructures are in relation to the bridge.

2.5.13 **Iroute = 2**

This section is used so that only the changes from previous substructures need to be input. There is a number corresponding to each parameter that can be changed. Thus, to change a value, input the number corresponding to the parameter to be changed and the new value.

Many geometric parameters can not be changed since a constant cross-section is assumed for node locations. An exception to this is the diaphragm configuration. It is possible to put K-bracing on one end of the substructure and X-bracing on the other end. If this is desired then a new substructure needs to be defined for the subsequent substructure. The reason is that the X-bracing and the K-bracing will not match when these two substructures are adjoined together.

The automatic determination of the mesh size is only done once for the bridge. If the spacing of the diaphragms change significantly then the number of divisions between diaphragms may need to be changed. When any material properties are changed, all alike elements are affected. Additional loads are also possible per section 2.5.2 part J.

2.5.14 Iroute = 3

The slight skew substructure can be used with or without supports. If there are no supports then the number of transverse rows until the girders are supported (Nsbc) should be input as zero. A few examples of the use of a slight skew substructure are shown in Figure 2.18. The most common use would be a slight skew at an end support in which the girders at the end are supported. Also, a common use would be at an interior support in which the diaphragms are either radial or skewed with the support. It is possible to have a skewed mesh for a curved substructure. In this case the degree of skew is measured from the radial line at the reference line as shown in Figure 2.17. All girders are assumed to be supported between diaphragms. If this is not the case, then either Iroute=4 or Iroute= 5 must be used. Additional loads are also possible on slight skew of substructures.

2.5.15 Iroute = 4

A severe skew has been assumed to go from one girder to another at diaphragm locations as shown in Figure 2.19. This may be quite a severe restriction but it was done in order to ensure a proper generated mesh. This assumption still allows quite a bit of flexibility as shown in Figures 2.19. If there are diaphragms between the first and last girder supports then Iroute = 4 must be called again for this second substructure. This type of substructure must be repeatedly used until all girders are supported. See the examples in Chapter 4 for further information. Additional loads are also possible for this substructure.

2.5.16 Iroute = 5

This type of substructure is used when the geometry of the bridge does not lend itself to one of the previously specified substructures. Two examples may be the best to present this section as shown in Figures 2.20 and 2.21. The first example is a curved substructure and the second is a triangular substructure. Cylindrical coordinates are used for Figure 2.20 and cartesian coordinates are used for the straight triangular substructure. The node numbering should be consecutive when bordering a data-generated substructure as in Figure 2.22. The nodal coordinates must match exactly in the x-axis and z-axis when it borders on a data-generated substructure. The user could run a trial data-generated substructure to determine these coordinates.

Five different elements can be input. The web and slab elements can be either quadrilaterals or triangles. If the element is a triangle it has three nodes and one subtriangle. If it is a quadrilateral, it has four nodes and four subtriangles as shown in Figure 2.3. The distinction is made between a web and slab elements for ease of input and output only. The program also offers two membrane stiffness elements. The QM5 should be used for quadrilateral elements when the mesh is mostly regular. The CLST should be used if the element is severely distorted or is a triangular element. In most cases the QM5 is preferred for quadrilateral elements and the CLST for triangular elements.

The distinction is also made between a rigid link, beam element and flange element for ease of input and output only. The beam elements are used for the diaphragms and the output is in forces. The diaphragm can be made a truss if the inertias and torsional rigidity are set equal to a small number or zero. The flange elements are assumed to be rectangular with the physical properties internally determined. The output for the flange elements is in stresses. The rigid link has the properties internally set in the program and the output for them is in forces.

The input in part G for the number of elements belonging to each substructure is most important for repeated substructures. If there is only one substructure then only the number of elements for the substructure must be input. The master nodes are global nodes and they describe how the substructures fit together. The data-generated substructure uses the nodes at diaphragm location for master nodes. The master nodes in Figure 2.22a could be 1-10 and 20-29. The master nodes in Figure 2.22b could be 1-4, 7, 10-13, and 14-23. For repeated substructures, the second through the rest of the entries in line six (see Appendix, Sec. 2.4.6, Part G) should be the negative number of the number of master nodes. For example, if there where three repeated substructures as in Figure 2.22a then the input would be 34, -20, -20 since there are 34 elements and 20 master nodes. The substructure in Figure 2.22b can not be repeated.

The control cards describe in more detail the master nodes for the program. The nodes that have boundary conditions that are not master nodes are used if there is an interior support. If the substructure in Figure 2.22a is supported in the middle at nodes 18 and 19 then the number of nodes having boundary conditions that are not master nodes is two since nodes 18 and 19 are not master nodes. The boundary conditions for each non-master node must be input in part J. The number of nodes which are not master nodes that have prescribed boundary conditions (Lpgi5) must equal the number of nodes which have boundary condition codes greater than one in part J.

Additional input is required for the data-generated program. The number of nodes for a data-generated cross-section is assumed constant and it would be 10 for both Figure 2.22a and 2.22b since they both have 10 nodes at a diaphragm location bordering a data generated substructure. The number of master nodes with boundary conditions must also be input along with the restraints. The number of master nodes with prescribed bondary conditions (Lpgo5) must also equal the number of nodes which have boundary condition codes greater than one in part M. The reference line length should be the arc length along the reference line. This is used for global placement of the truck and lane loads. All live loads must be input in the section for additional loads. This applies to both truck and lane loads. Thus, if a truck is half on and half off a substructure as in Figure 2.22 then the wheel loads must be resolved to nodal loads on the respective elements. Also, only the wheel loads on the data generated substructure must be input. Again, all loads should have the signs corresponding to the element coordinate system.

2.6 REMARKS IN GENERAL

This program actually was developed in three parts with the most of the work being done on pre- and post- processors. Many things had to be adapted to the existing program. One of the things that was used was master nodes. The master nodes in a general sense describe how the substructures fit together. In addition to those master nodes, any nodes can be master nodes such as the nodes on the ends of the bridge. The master nodes must be known for a substructure since the output of deflections is first output by master nodes and then by elements. The way the program determines which local node is a master node is through renumbering. The program automatically renumbers the respective local nodes to master nodes and outputs the renumbering scheme.

Many assumptions as to the size of problems to be solved were made. If problems are larger than the stated maximums then some guidelines to changing dimension statements are given in Appendix A.

2.7 SUGGESTIONS FOR THE APPROACH TO THE SOLUTION

The approach to this program was to apply easy input to a rather complex problem while still remaining flexible enough to handle many bridge types. Along this line, general guidelines in modeling are described below.

1) Determine a preliminary design with approximate methods such as the V-load, similiar bridges etc..
2) Determine the number of substructures for each span. A substructure is defined as the part of the bridge between two diaphragms as shown in Figure 2.2 .
3) Determine the number of different types of substructures. This is done by looking at changes in flange dimensions, radii , diaphragm members and configuration, diaphragm spacing and material properties of concrete and steel etc..
4) Of these types of substructures, determine which can be generated by the data generator. It is to the users advantage that as many substructures as possible be generated by the data generator. Since this is the case, approximations in modeling might prove extremely helpful. Four of the major approximations are in the flange dimensions, radius changes, concrete modulus and diaphragm spacing. The flanges are assumed to change at diaphragm locations. This is almost always not the case. The user must decide whether to continue or change a flange from one diaphragm to the next. An alternative is to change the diaphragm spacing to coincide with change in the flanges. The radius is also assumed to change at diaphragm locations. Since there is generally not a sharp change in the radius, this assumption is not seen to be too critical.

The place where the concrete is ineffective, such as in negative moment regions, must also coincide with a diaphragm. No reinforcement bars are included in the slab. Over negative supports the concrete modulus can be set equal to the following equation:

$$E' = (\text{Area of rebar} \times E\ \text{concrete}\ 2)/(\text{Area of concrete} \times E\ \text{steel})$$

The user can run a fully composite bridge and determine the area of concrete in tension. The user can then set the concrete modulus in the appropriate places.

5) Determine how many alike substructures are repeated consecutively. It is also to the user's advantage to repeat as many substructures consecutively for ease of input. The same assumptions that apply in item 4 (above) can apply here.
6) Determine the refinement of the slab mesh. As a first approximation, one can take the spacing of the diaphragms and divided it by the web depth. The number of longitudinal divisions is this whole number. The number of transverse divisions between girders is the girder spacing divided by the longitudinal division increment. The idea is to get a more or less square mesh. Since the web depth is fixed, the slab elements size should be reasonably close to the web depth. The difference should be less than about 2.5 to 1.0 and preferably less than 1.5 to 1.0.
7) Determine the physical parameters for each substructure type as needed per the documentation.
8) Draw a sketch of the bridge layout and substructure types.
9) If some substructures need to be input by hand then layout the nodal coordinates and the element types. Determine the data needed per documentation for section 2.4.6.

10) Determine live loads. The location of the live loads for maximum effect could be roughly determined with approximate methods such as influence lines or by the V-load method of Ref 14.
11) Run the dead load case with the longitudinal elastic modulus of concrete equal to a small number. This simulatates the bare steel while still getting the correct slab load to the girders. If it is desired to sum subsequent load cases set Jrec and Ircsum equal to one.
12) Run the live load case with the appropriate concrete elastic modulus. If it is desired to sum the live load case with the previous dead load case then set Jrec equal to two and Ircsum equal to one.

The solution time and cost will increase with an increase in the total number of elements. For preliminary analysis, a coarser mesh and more approximations can be made. As the solution is finalized, it may be necessary to refine the mesh.

CHAPTER 3. OUTPUT INTERPRETATION

3.1 INTRODUCTION

The output is described in this chapter from the input given in Chapter 2 This program was originally developed as three separate programs GENPUZ, PUZF83 and RECPUZ. There are many check prints to ensure that the correct information is transferred between programs. In addition, there are many parameters that are internally generated which are also check printed.

Two parameters that control the amount of output are IPL and IDEF. IPL equals one if plots of the XY and YZ cross-sections are desired. IPL equals two if a plot of concrete slab stresses in additon to the cross-section plots are desired. IDEF is used to echo the degree of output. If IDEF equals zero then only the minimum of output is shown. This basically includes echo prints of input data and element stresses. No deflections are output. If IDEF equals one then deflections by master node and elements are output along with echo prints of input data. If IDEF equals two then deflections and the maximum number of check prints are output.

3.2 GENPUZ

The GENPUZ program reads in all the data and determines the element stiffnesses and load arrays. It does this completely for each type of substructure.

3.2.1 GENPUZ Control Cards

The first output is two lines of labeling information for this run. This is followed with an echo print of the data for the control cards in part A, line 3. The number of groups before half-diaphragms are added refers to the number of groups or types of substructures input. This need not be the final number since the program internally generates the half-diaphragms on the end of the bridge if required. NDOF at each joint has been internally set equal to six. IGLB and IROT are variables used in PUZF83 and are internally set. The plotting option (IPL) and output print option (IDEF) have been described above. The superpostion of stresses are used in the construction stage or in different load cases if the stresses from the current computer run are to be superimposed on the previous computer run. The rest of the check print should be the same as the data for the control cards in Section 2.4.1, part A.

3.2.2 GENPUZ Erection Output

This section is output if the erection stage (Jrec) is greater than zero. Each steel stage number, along with the starting and ending girder, and the starting and ending substructure, is output. This should be exactly as input in Chapter 2.

3.2.3 GENPUZ Input Echo for Iroute = 1 to 4

The first output is for the substructure type. (Iroute) The support conditions are only output once and correspond to the input in Chapter 2. The substructure type value is renumbered if the half-diaphragms are added on the end of the bridge. Thus, if the half-diaphragms are added on the end of the bridge, then the first substructure type is the half-diaphragm. The second substructure type is the original number one substructure. The bridge geometry, girder, and flange information should all be output the same as the input for Chapter 2.

The diaphragm member and the bracing index numbers should correspond to Figures 2.8a, b, and c. The inertia-y and inertia-z should correspond to the local member axis as shown in Figure 2.3c.

The material properties and the slab loads should also correspond to the input for Chapter 2. The curb load should be for the correct inside or outside overhang. The density and slab loads should be positive if downward.

The truck location and direction is output for each truck centroid. The local distance from this centroid is then output for each wheel. If IDEF equals one then the generated global location of the concentrated loads are output. The load should be positive if downward. The truck is assumed straight going around a curve so that the relative transverse distances and longitudinal distances are not the same in the global coordinates. The global transverse location is the distance from the reference line if the substructure is straight or the radius of the curve at the load if the substructure is curved. The global longitudinal location is the length along the reference line as shown in Figure 2.10.

The lane load location is output for each centroid. The local distances are then output relative to the centroid. If IDEF equals one then the global locations of the concentrated loads are output. The global transverse location is the distance from the reference line if the substructure is straight or the radius of the curve at the load if the substructure is curved. The global longitudinal location is the length along the reference line. The upper left and lower right corners are also ouput for the distributed load. A negative load is downward.

3.2.4 Nodal Coordinates

The nodal coordinates are output in either cylindrical or cartesian coordinates depending on the substructure geometry. If IDEF equals two, then three values are output that are important to the program. These numbers are counters for the number of nodes at specific points at a cross-section. They are used in the bookkeeping process to determine element nodes, master nodes and support conditions.

3.2.5 Element Material Properties

The program generates the element material properties along with the node numbers. Each type of element is output along with the calculated material properties. The beam elements should have the same values as input in Chapter II except the density which is in units of kips per inch. The density of the web and slab elements are in units of kips per inch cubed. The elastic moduli for the web is assumed to be the same in both directions while different elastic moduli for the slab can be input . Only one poisson ratio is printed out in the interest of space but the poisson ratio in the y-axis direction is calculated based on elasticity equations. The area, GJ, Iy and Iz of the flange elements are all calculated based on a rectangular plate with the axis as shown if Figure 2.3.c. The density is in units of kips per inch. The stud properties have been internally set with no density.

3.2.6 Element Node Numbers

The element node numbers should correspond with the nodal coordinates, the element type, and the element material properties. The number of elements, nodes, and repeated substructures are then output as a check.

3.2.7 Aspect Ratio

The aspect ratio of the plate elements are calculated for the user's information only. Error statements are printed if the maximum aspect ratio exceeds five or the average aspect ratio exceeds four. These values are set to ensure that the finite elements give valid answers. If the aspect ratio is too large, then the user must redefine a mesh.

3.2.8 Master Node Information

The master nodes for a data-generated substructure have been defined as the nodes at diaphragm cross-sections. Thus the master nodes correspond to the beginning and ending of the substructure. The number of master nodes should be twice the number of nodes for a cross-section with diaphragms. The number of nodes having boundary conditions, excluding master nodes, may be greater than zero for substructure types for Iroute = 3 and 4. These nodes that have boundary conditions may be in the interior of the substructure and are not master nodes. If IDEF equals two then a check print is made if the nodes are renumbered. If interior nodes have boundary conditions then the local node and the boundary conditions are printed.

The substructure local nodes are renumbered to master nodes. The master nodes are used by the program to put the substructures together into a bridge. The user should be aware that master nodes are used and that the deflections are printed out by master nodes as well as element nodes.

3.2.9 Superimposed Loads

All superimposed loads due to truck, lane or any additional loads are printed out. The substructure number corresponds to the repeated substructure number and not the number of substructures in the bridge. The DOF and the local node number corresponds to the local element axis. Local node 1 corresponds to node i for example. The load direction also corresponds to the local axis as shown in Figure 2.3a, b, and c. All slab loads are negative downward.

3.2.10 GENPUZ Input Echo for Iroute = 5

Much of the output is the same for a substructructure input by hand and a data-generated one. There is, of course, more input required and this is echoed in the program for data verification.

The number of elements belonging to the substructure are output if IDEF equals two. This data is part of a larger array which is output at the completion of GENPUZ. The number of master nodes for this substructure and the number of nodes having boundary conditions excluding master nodes should be as input in part H for Iroute = 5. The substructure nodes chosen to be master nodes should be renumbered and the new master nodes should be shown. All the rest of the input as per parts I through O for Iroute = 5 should be output exactly as input.

3.2.11 Substructure Assemblage Information

The substructure assemblage information echos input that is required for the main solver PUZF83. The first five items output if IDEF = 2 are preset values in the program corresponding to the maximum size of arrays possible for solution. The number of master nodes with boundary conditions should be for the entire bridge. If IDEF = 2 then an echo of the print options is given. The first five values are check prints for PUZF83 and they should be preset equal to zero. The value of IPE should be the same as IDEF. The maximum number of master nodes, the total number of elements, and the total number of substructures correspond to the entire bridge.

The number of elements per substructure follows the description if part G for Iroute = 5. If the number is positive then this should be the number of elements for that substructure. If the number is negative then the number should be the number of master nodes for the substructure. The boundary restraints are internally set and should correspond to the entire bridge.

3.3 PUZF83

PUZF83 is the main solver of the program and determines the nodal displacements. The first data listed for PUZF83 should be the same as the substructure assemblage information at the end of GENPUZ. The data are written on a tape and this is a check print to make sure that the data were passed correctly.

3.3.1 Echo Print of Master Nodes

The next output if IDEF equals two is a check print on the number of substructures. This is then followed by the number of elements per substructure (Isub) as described in GENPUZ. The rest of the output for this section is verification that data was read correctly by PUZF83. The main thing that the user should check is that the nodes are renumbered to the correct master nodes and the boundary conditions are correct.

3.3.2 Storage Requirements

The storage required is shown for the user's information on solution size. The user should look at these numbers and make sure that they are less than the values set in section 3.3. Also, the user can get an idea of the size of the problem that is trying to be solved.

3.3.3 Computation Effort in Reduce

This section is printed out if IDEF equals two. This section shows how efficiently the buffer size is used in processing the solution.

3.3.4 Deflections by Master Nodes

The Deflections are output by master node if IDEF is greater than or equal to one. The deflections are printed out starting with the last substructure and working backward. The deflections are printed out in increasing order starting with the lowest master node number for that substructure. The renumbering information in section 3.2.8 should be used for ease of interpretation. The reactions can be determined with the boundary condition information in section 3.2.9. The default spring constant of 1.0×10^{50} was used at all supports where prescribed displacents or springs were not input. Thus, to get the reactions, find the vertical deflection at the master node which is supported and multiply by the spring constant. The reaction is then in kips.

3.3.5 Check Print for Isub and Jsub

This check print is done if IDEF equals two. Isub should be the same as in section 3.3.1. The positive numbers of Jsub should be the number of repeated substructures for each substructure type. The negative numbers should be the number of elements for the substructure. The output for Jsub is in reverse order of the substructures.

3.3.6 Deflection by Element

The deflections by element are output if IDEF is greater than or equal to two. The deflections are output by all alike elements for each repeated substructure. For example, it is easy to tell the variation of the deflection of the bottom flange for one substructure type as that substructure is repeated across the span since the deflections are printed out all in order. The output across the row are printed out the same as for the master nodes, i.e. x, y, z, rot-x, rot-y, and rot-z. The first line of deflections correspond to node i, the second line corresponds to node j and so on. The axis of orientation at the node is in the direction of the surface coordinates for that element.

3.3.7 Solution Time

The solution time should be used by the user in much the same way as the storage required. The user can determine how to cut down on the solution time by keeping track of the time in various modeling schemes.

3.4 RECPUZ

Recpuz determines the stresses in the elements. The first output should be the number of groups of substructures. If this is a construction sequence, and stresses are to be superimposed, then the disk file number of the current stresses and the disk file used for superimposed stresses are printed out. These disk files should alternate with each load stage. The user has the option of plotting the concrete slab stresses at each node if IPL was set equal to 2.

3.4.1 Stresses

The output of the stresses are broken up into five groups for ease of interpretation. The output of stresses and forces for all the elements are with respect to the local member axis as shown in Figure 2.3a, b, and c. The forces are printed out for the rigid link and the diaphragm beam elements. The user can resolve these forces into stresses with the respective element properties. The stresses for the flange elements are printed out based on a rectangular section. The sigma x -YY and sigma x -ZZ are the strong axis and weak axis bending stresses relative to the center of the flange as shown in Figure 3.1. The torque is only relative to the nodes of the member and not to the entire girder as a whole. The stresses at the midpoint of the web and slab elements are output for the top and bottom of the element. The user is cautioned that these stresses are at the midpoint and not at each node. The slab stresses at each node can be output if Idef equals two. These stresses are the same as the ones plotted.

The summation of the load case stresses or forces are printed out directly below the stresses or forces of the current erection stage. This is indicated with the word "sum" next to the element number.

The plot of stresses are the average stresses at each node for each substructure. The stresses at the borders of two substructures may not be exactly the same due to this averaging. It is up to the user to average these stresses.

3.5 OUTPUT SUMMARY

The bridge geometry, loadings, and material properties are determined and output in the first part of the program. Deflections by master node and by element are then output for the bridge, if desired. The stresses by element are then given along with summation of stresses from the different load stages. Plots of each substructure type in the XY and YZ cross-section are available along with plots of the average slab stresses. All the output is given in a space that is able to be trimmed to a neat 8-1/2 by 11 paper size.

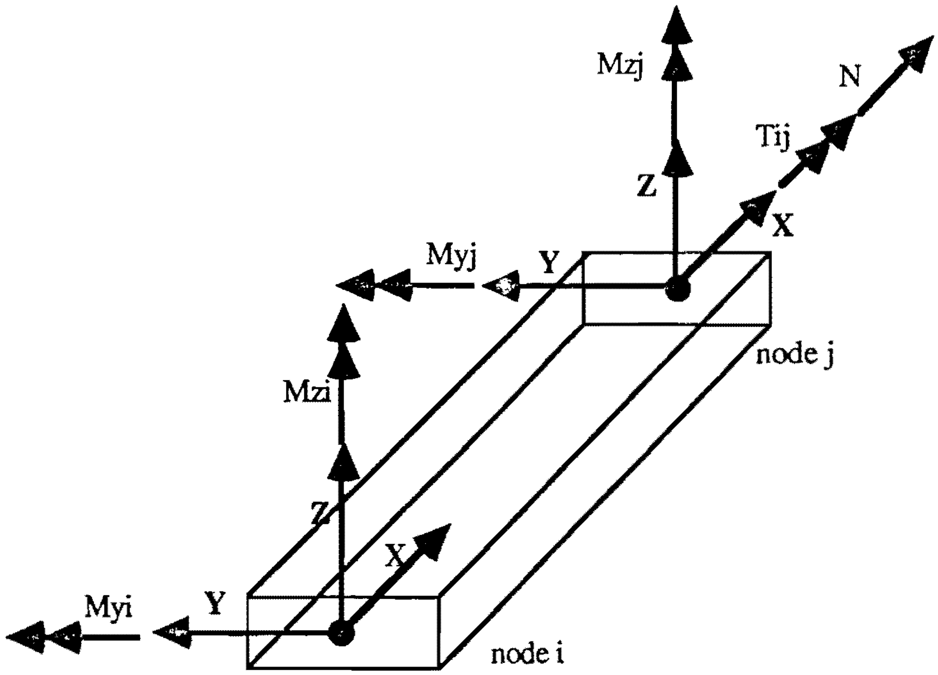


Figure 3.1a Beam element axes orientation

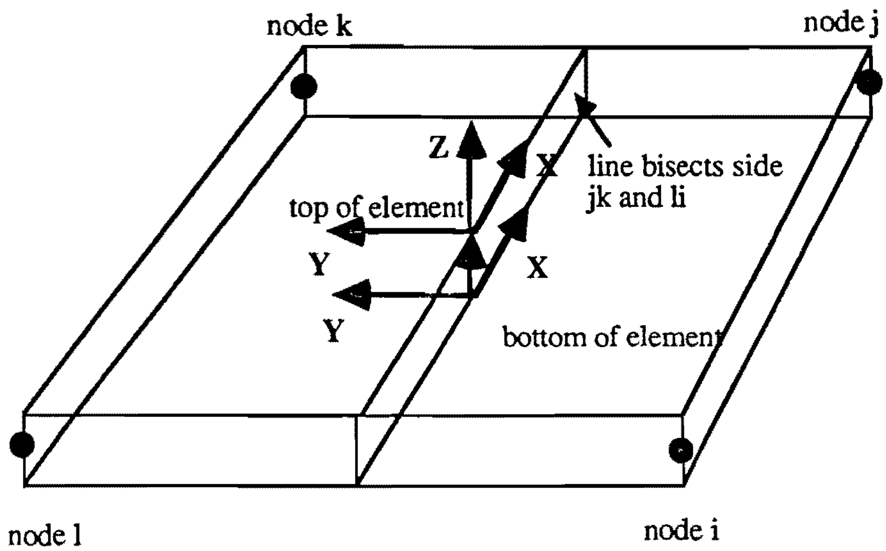


Figure 3.1b Plate element axis orientation

CHAPTER 4. EXAMPLE PROBLEMS

4.1 INTRODUCTION

Six example problems will be described in this chapter to demonstrate both the input procedure and the versatility of the program. They are:

1) Simple span, two-girder curved bridge with a radius of 500 feet.
   This example will be shown for both a dead load noncomposite case and a truck load case.
2) Three-span, four-girder curved bridge with a radius of 1,145.9 feet.
3) Two-span, three-girder straight bridge with severely skewed interior supports and radial end supports.
4) Two-span, three-girder straight bridge with all supports severely skewed.
5) Two-span, four-girder curved bridge with all supports slightly skewed. The radius and diaphragm spacing also changes for the bridge. This example will be shown for a dead load noncomposite case, a two truckload case and a two lane load case.
6) Two-span, four-girder curved bridge with reverse curvature and all supports slightly skewed.This example will be shown for a dead load noncomposite case, a two-truck load case and a two-lane load case.

4.2 EXAMPLE 1

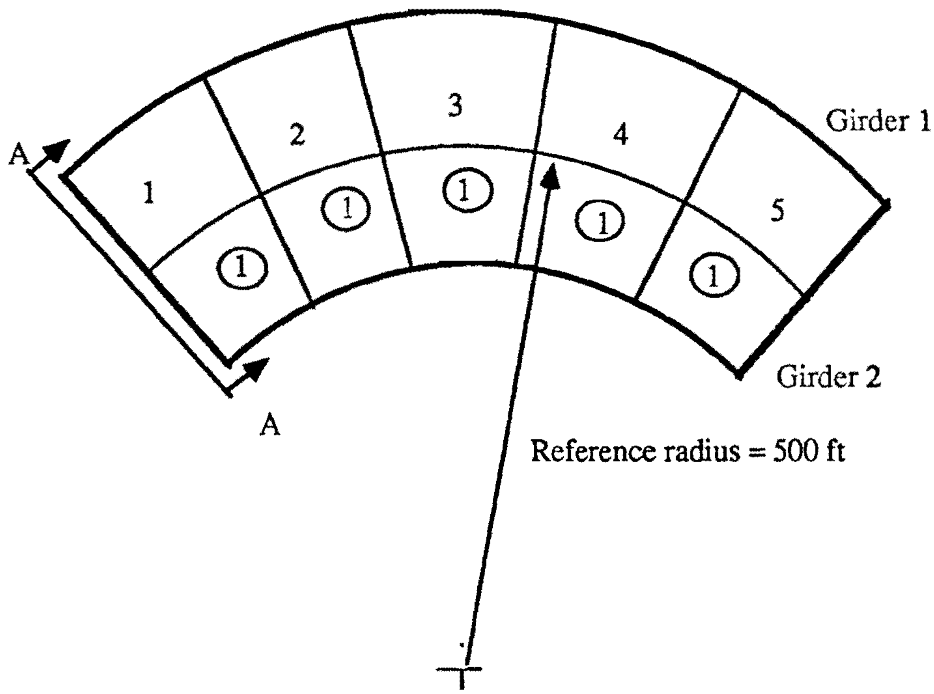
The first example is a simple span, two girder bridge with a radius of 500 feet. This problem is probably one of the easiest ones for this program and it is also part of the example problems used to test the V-load and finite element programs. Cross-section and plan views are shown in Figure 4.1.

Two load cases are to be run for example 1.

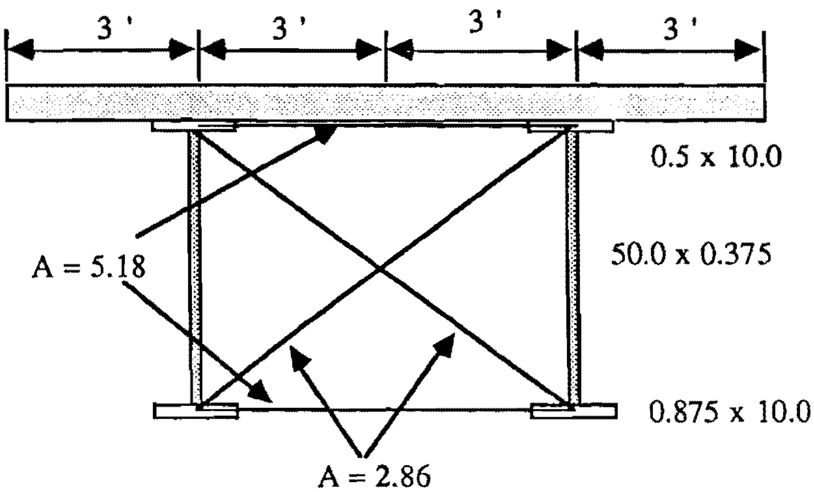
Run dead load of concrete and steel for a noncomposite geometry.

Run live load of an HS20 - 44 truck going forward with the middle wheels at 54 feet from the left end of the road as shown in Figure 4.2. The following is given for the example:

Centerline span length = 100 feet.
Centerline radius = 500 feet.
Diaphragm spacing = 20 feet.
Concrete slab thickness = 7.5 inches.
Slab overhang = 3.0 feet each side.
Girder spacing = 6 feet.
Top flange = 0.5 x 10.0 inches.
Bottom flange = 0.875 x 10.0 inches.
Web = 50.0 x 0.375 inches.
X- bracing with top horizontal member.
Area of diagonal diaphragm braces = 2.860 in2.
Area of horizontal diaphragm braces = 5.18 in2.
Elastic modulus of steel = 29,000 ksi.

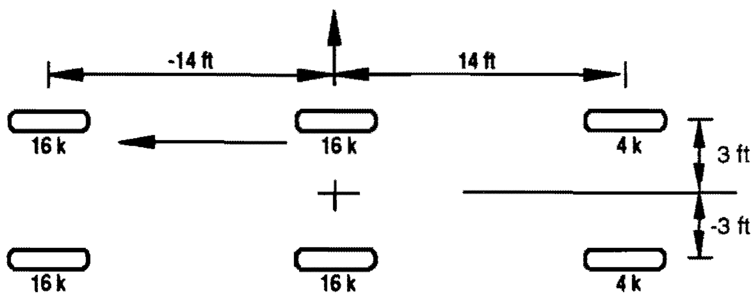


a) Plan view of Example 1

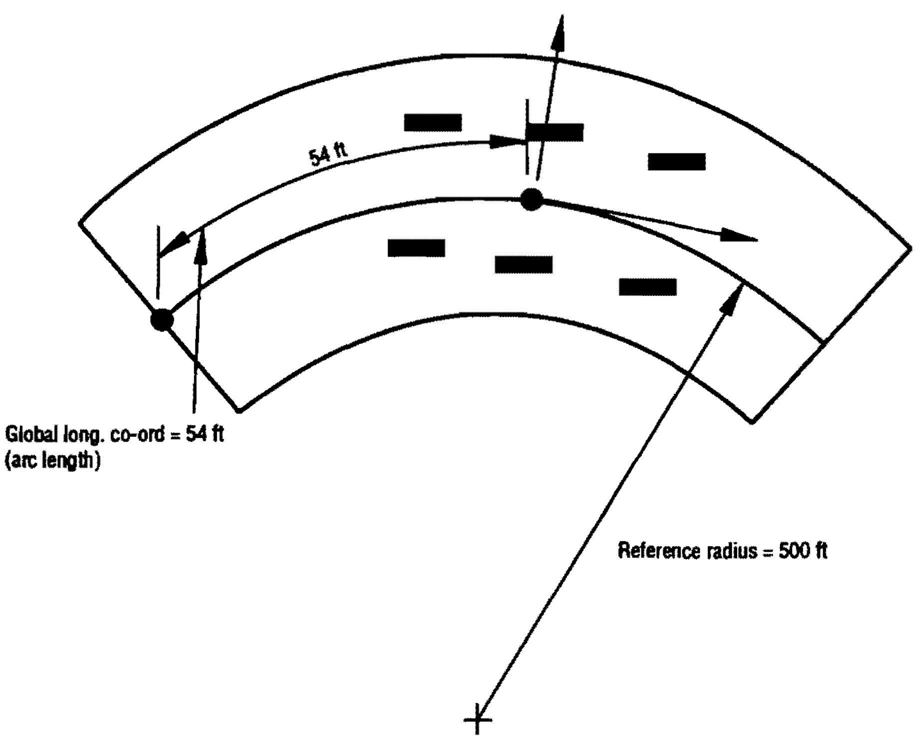


b) Section A-A of Example 1

Figure 4.1 Example 1



a) Truck example



b) Truck placement for Example 1

Figure 4.2 Live load for Example 1

Elastic modulus of concrete = 3,625 ksi.
Density of Concrete = 0.150 k / ft^3.
Density of Steel = 0.490 k / ft^3.
Poisson ratio for concrete = 0.17.
Poisson ratio for steel = 0.30.

4.2.1 Dead Load for Example 1

The following is a guide for preparing the data for example 1. A listing of the input is given in Appendix B. The first two lines are description lines for this program run. For the control cards, the number of groups equals one since the bridge has a constant cross-section. This number will automatically be increased to three when the half-diaphragm substructures are added at each end of the bridge. IDEF should be set equal to 1 if a print-out of deflections by master nodes is desired. The next thing that is considered is that the run is for the dead load and these stresses will be superimposed on the live load stresses. Since this is the first load case, JREC should equal 1. Also, since the live load stresses are going to be added to the dead load stresses in a future run, IRCSUM should equal 1. IRCSUM tells the program to store the stresses on a disk file and to read any old stresses. Load factors for the dead load can be taken into account with the material densities but for this simple case the live and dead load factors equal 1.0 .

Since there is one 100-foot span, and the diaphragm spacing equals 20 feet, then there are five substructures for the span. The horizontal support restraints can be applied to either the first or last support. In this case, the horizontal restraints are applied at the first support and NFIX equals zero. The rest of the data is easily input. Remember, the diaphragm properties must be divided by two since they are adjacent to other substructures. Also, since this is a noncomposite section, KOMP should equal 1 and the concrete modulus must be set equal to a small number like 1.0 ksi. KOMP sets the concrete slab stresses equal to zero and makes the rigid link a truss element so the studs are less effective.

4.2.2 Truck Load For Example 1

The live load must be run after the dead load run. To run the live load, set JREC equal to 2. To make the bridge composite set KOMP equal to zero and set the concrete modulus equal to the proper value. Also, the density of the steel and concrete needs to be set equal to zero so that the proper stresses are superimosed. To signify that a truck is to be input, set ZLIVLD equal to 1.0. Input the correct geometry of the truck as shown in Figure 4.2. The location of the truck centroid is arbitrary but is usually easier if it is placed at the truck centerline.

The live load is run and the sum of the stresses is printed out below the stresses due to the truck only. If any error occurs, increase IDEF equal to two and check the global placement of the concentrated loads. Remember, the truck is assumed to be straight when going around a curve.

4.3 EXAMPLE 2

The bridge for example 2 is shown in Figure 4.3 and Figure 4.4 and represents an example of a highway bridge sent from the Houston Bridge Division. This three span, four girder bridge example was used because two other computer programs have been run on the bridge and the results could be verified. Also, the bridge is quite simple in geometry but still represents a typical design case. A listing of the input data is given

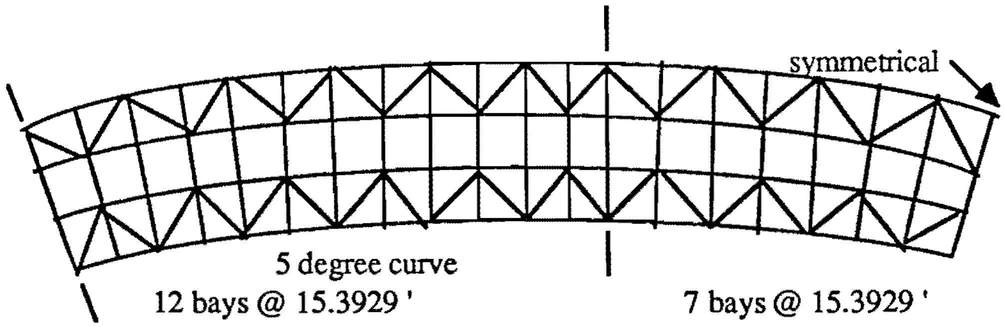


Figure 4.3 Plan view of Example 2

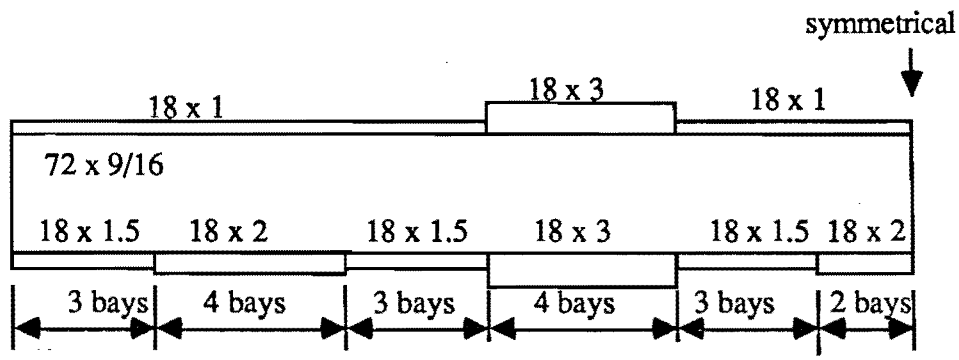


Figure 4.3b Elevation view of typical girder for Example 2

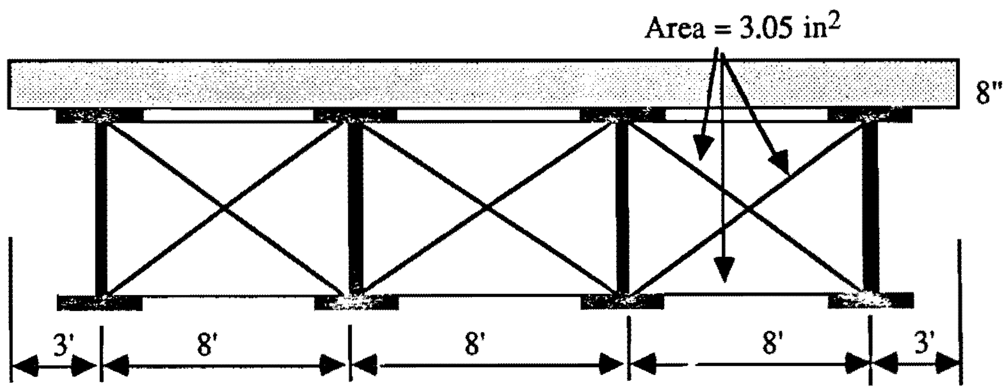


Figure 4.4 Cross-section of Example 2

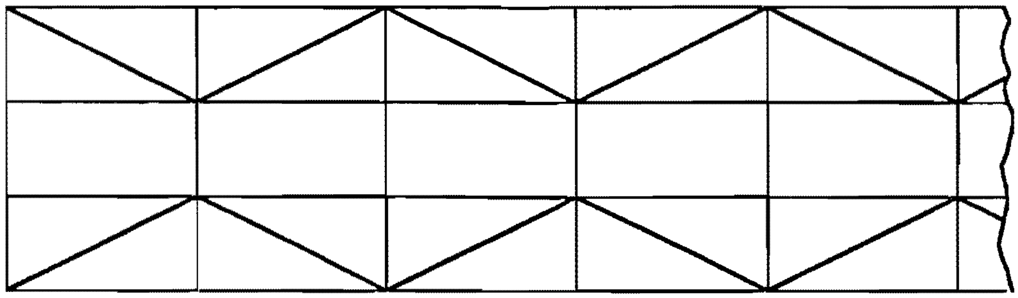
in Appendix B. Run the dead load due to concrete and steel for a noncomposite geometry since the procedure for live load placement is similar to example 1. The following is given for Example 2:

Centerline span lengths = 185.15, 210.0 and 185.15 feet.
Centerline radius = 1,145.9156 feet. (5 degree curve)
Diaphragm spacing = 15.3929 feet.
Concrete slab thickness = 8.0 inches.
Slab overhang = 3.0 feet each side.
Girder spacing = 8 feet.
Area of diagonal braces of diaphragms = 3.05 in^2.
Area of horizontal flange braces = 3.05 in^2.
Area of diagonal bottom lateral brace = 2.81 in^2
Elastic modulus of steel = 29,000 ksi.
Elastic modulus of concrete = 1.0 ksi. (noncomposite)
Density of Concrete = 0.150 k / ft^3.
Density of Steel = 0.490 k / ft^3.
Poisson ratio for concrete = 0.17.
Poisson ratio for steel = 0.30.

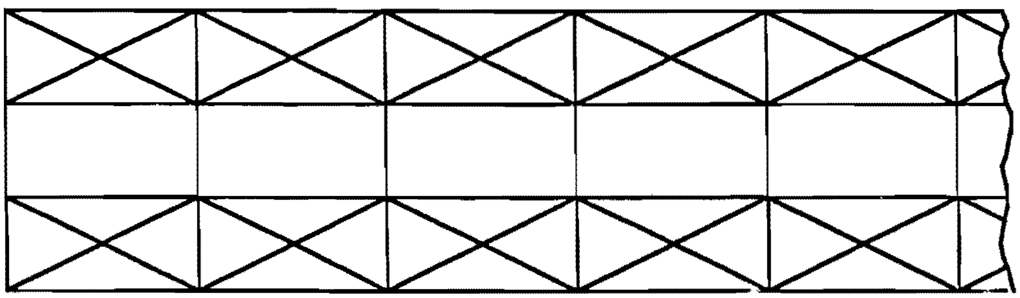
4.3.1 DEAD LOAD FOR EXAMPLE 2

This example is not much different than the first example except for two things. First, the flange dimensions change. This means a new substructure needs to be defined whenever the flanges change. Since the flanges change 11 times, there are 11 groups of substructures. Since the flange dimension changes, an average top and bottom flange thickness needs to be input for the vertical placement of the nodes. It is up to the user to decide what an appropriate average should be. For this example, the average top and bottom flange thicknesses were set equal to zero so the nodes would be placed on top and bottom of the 6 foot web. The second difference is the presence of bottom flange diagonal braces. The program does not easily handle braces which switch back and forth since a new substructure needs to be defined everytime this happens. An easy way to model it would be to put x-bracing in each bay with half the area as shown in Figure 4.5b. In this way the bracing configuration needs to be input only once and the stiffness and the dead weight is almost the same. This method should not be used for fatigue loading since the stresses in the braces are probably not the ones the model describes. Another way to model them is shown in Figure 4.5c in which the bracing direction changes when a new substructure needs to be defined anyway. The bracing can then go in one direction and then switch near the inflection point.

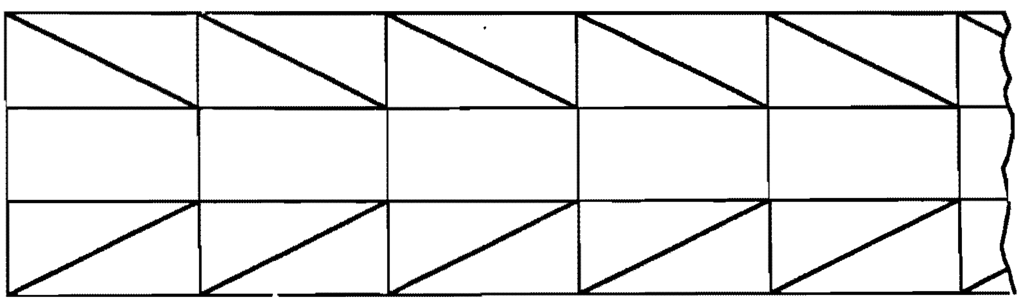
When the flanges change, the substructure geometry needs to be updated and the data from section 2.4.3 needs to be used. The two values that need to be changed are the number of repeated substructures (NV) and the new flange dimensions. After each substructure is input, a zero is placed in column one to signify no additional loads are placed on this substructure.



a) Designed braces for most economical results
(A new substructure is defined for each bay)



b) Alternative X-bracing
(Each bay has the same bracing configuration)



c) Alternative bracing scheme
(Change bracing configuration when flanges change
or at inflection points)

Figure 4.5 Examples of bottom horizontal bracing

4.4 EXAMPLE 3

Example 3 is a two-span, three-girder bridge with a severe skew at the interior support and radial end supports. This example will show how to deal with a severe interior skew as well as specified support spring stiffnesses. The bridge is shown in Figure 4.6 and a listing of the input data is given in Appendix B. The bridge should be run for dead load due to concrete and steel for noncomposite geometry. The following is given for example 3:

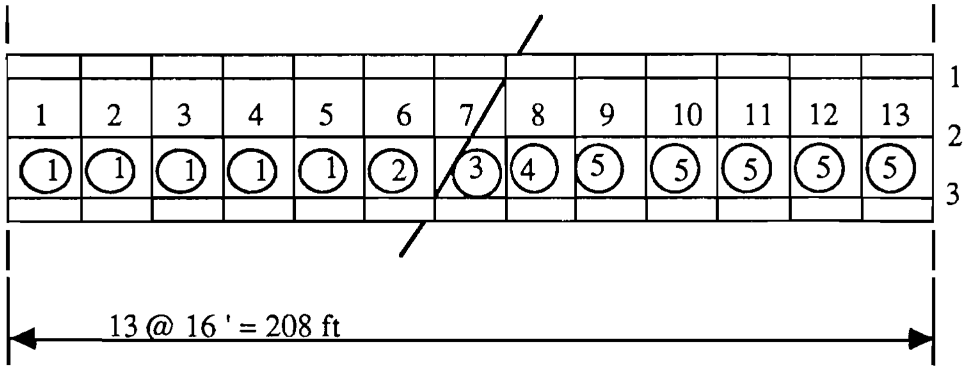
Centerline span lengths = 104 and 104 feet.
Centerline radius = 0.0. (straight)
Diaphragm spacing = 16.0 feet.
Concrete slab thickness = 7.5 inches.
Slab overhang = 3.0 feet each side.
Girder spacing = 8 feet 10 inches.
X- bracing without top horizontal member.
Area of diagonal diaphragm braces = 2.860 in^2.
Area of horizontal diaphragm braces = 5.18 in^2.
Elastic modulus of steel = 29,000 ksi.
Elastic modulus of concrete = 1.0 ksi. (noncomposite)
Density of Concrete = 0.150 k / ft^3.
Density of Steel = 0.490 k / ft^3.
Poisson ratio for concrete = 0.17.
Poisson ratio for steel = 0.30.
Support stiffness for girder 1 = 1.0 x 10^{30} kips/inch.
Support stiffness for girder 2 = 2.0 x 10^{30} kips/inch.
Support stiffness for girder 3 = 3.0 x 10^{30} kips/inch.
Interior support horizontally restrained.

4.4.1 Dead Load for Example 3

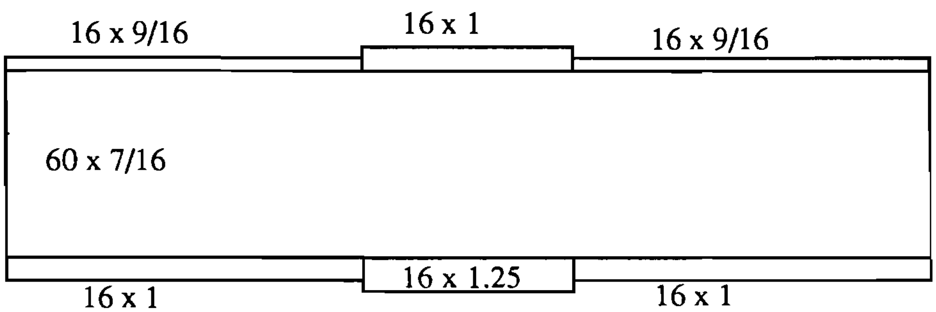
This example is straightforward except for the support springs and the skew of the interior support. As described in Chapter 2, the order of the spring input must correspond to the order in which the supports are numbered. Different values for the girder spring stiffnesses were used to better show the proper order of input. The listing of the input including the proper order of the spring stiffnesses is given in Appendix B.

The interior skew is handled with the input from Section 2.4.5. The additional input required for substructure 7 is only the starting and ending girder number. In this case the support starts at girder 3 and ends at girder 1 as shown in Figure 4.6. The support at girder 2 is on the interior of the substructure so that there needs to be two divisions between diaphragms so that a node is over girder 2. The number of substructures for the first span is 7 and 6 for the second span. This is in keeping with the definition of the number of substructures per span given in Chapter 2. Since the interior support is horizontally restrained, NFIX equals 7.

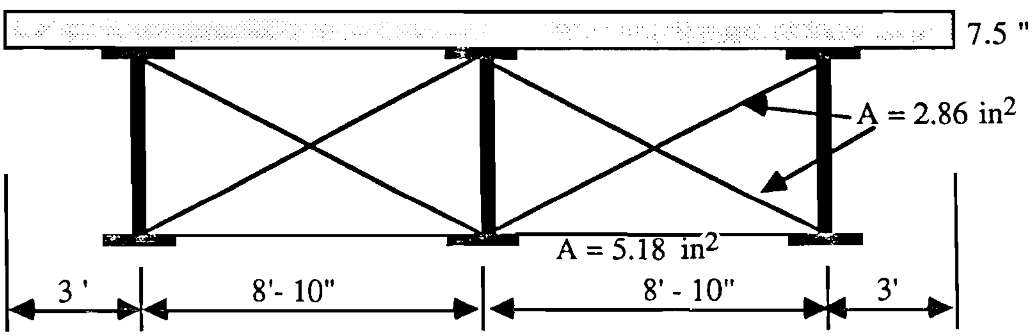
The user must decide how to best model the change in flange dimensions for the interior skew. Even when the supports are skewed, the flange properties must still change at a diaphragm location. This is quite a severe restriction near negative supports since the flanges most likely do not change at diaphragm locations. When there are skewed supports and flange changes, the user can place the flange changes at diaphragm



a) Plan view for Example 3



b) Typical girder elevation for Example 3



c) Cross - section for Example 3

Figure 4.6 Example 3

locations or put dummy diaphragms (zero physical properties) at places where the flanges change. For girders 1 and 3 this is not a problem since the flange dimensions can change one bay away from each support. For girder 2 the flanges must change between substructures 6 and 8 or substructures 5 and 9. For this example it was decided to change the flange dimensions for substructures 6 and 8. Thus, the flanges for the negative moment region of girder 2 do not extend as far as girders 1 and 3.

4.5 EXAMPLE 4

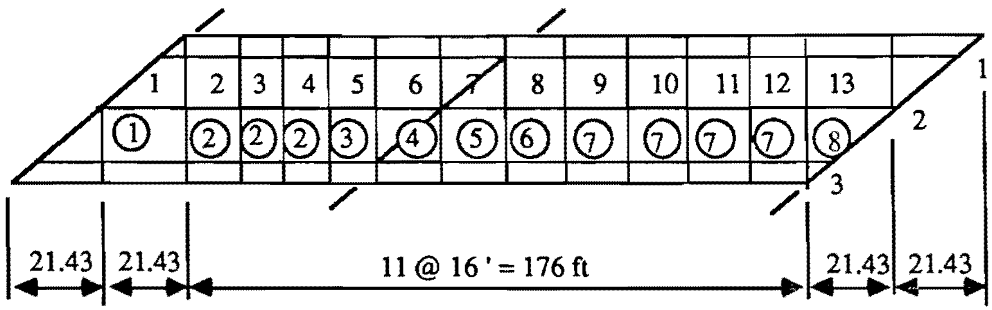
Example 4 is similar to example 3 except the end supports are severely skewed and the interior support is skewed across two bays instead of one. A listing of the input is shown in Appendix B. The bridge should be run for dead load due to concrete and steel for noncomposite geometry. The following is given for example 4:

Centerline span lengths = 111.434 and 111.434 feet.
Centerline radius = 0.0. (straight)
Diaphragm spacing = 21.434, 16.0 feet.
Concrete slab thickness = 7.5 inches.
Slab overhang = 3.0 feet each side.
Girder spacing = 8 feet 10 inches.
X- bracing without top horizontal member.
Area of diagonal diaphragm braces = 2.860 in2.
Area of horizontal diaphragm braces = 5.18 in2.
Elastic modulus of steel = 29,000 ksi.
Elastic modulus of concrete = 1.0 ksi. (noncomposite)
Density of Concrete = 0.150 k / ft3.
Density of Steel = 0.490 k / ft3.
Poisson ratio for concrete = 0.17.
Poisson ratio for steel = 0.30.
Support stiffness for girder 1 = 1.0×10^{30} kips/inch.
Support stiffness for girder 2 = 2.0×10^{30} kips/inch.
Support stiffness for girder 3 = 3.0×10^{30} kips/inch.
Interior support horizontally restrained.

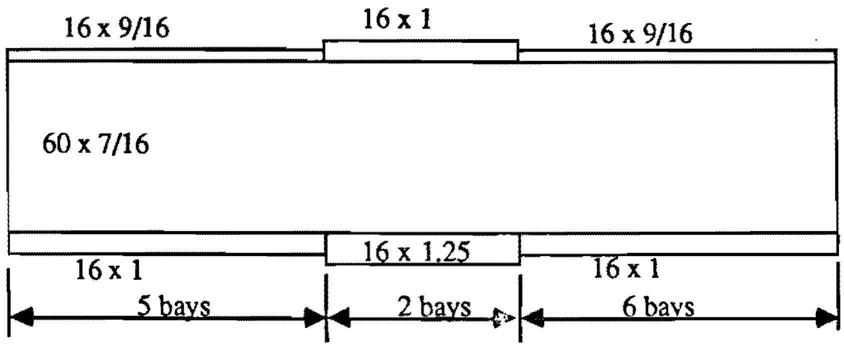
4.5.1 Dead Load for Example 4

The data generator can easily handle the interior support but it can not generate the triangular substructures 1 and 13. For these special substructures the data from Section 2.4.6 needs to be input. Pictures of substructures 1 and 13 are shown in Figures 4.8 and 4.9. The input required for these sections is much longer than others and the support springs have to be input for these end substructures. Also, the diaphragm properties at the end of the bridge are no longer handled automatically in the program. Thus, the diaphragm properties must have the full value at the end of the bridge and half the value when bordering a substructure.

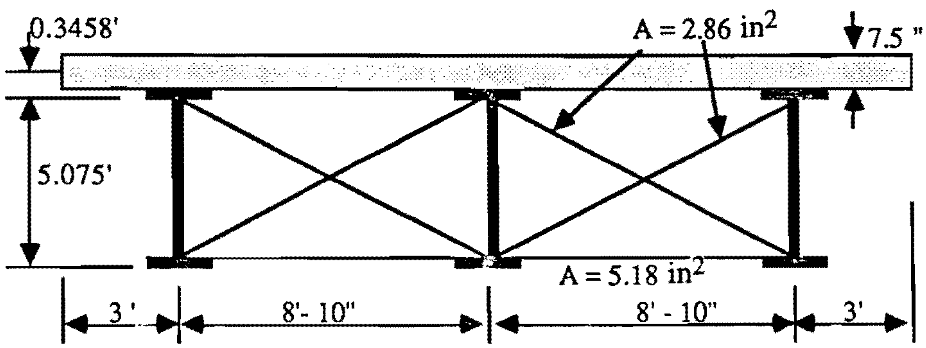
The user must decide how to best model the concrete at the skewed support region. The parallelograms which cantilever out will give very large displacements if the elastic modulus is set equal to a small number. The use of the orthotropic properties might be especially useful in modeling the concrete of



a) Example 4 plan view



b) Typical girder elevation



c) Cross - section for Example 4

Figure 4.7 Example 4

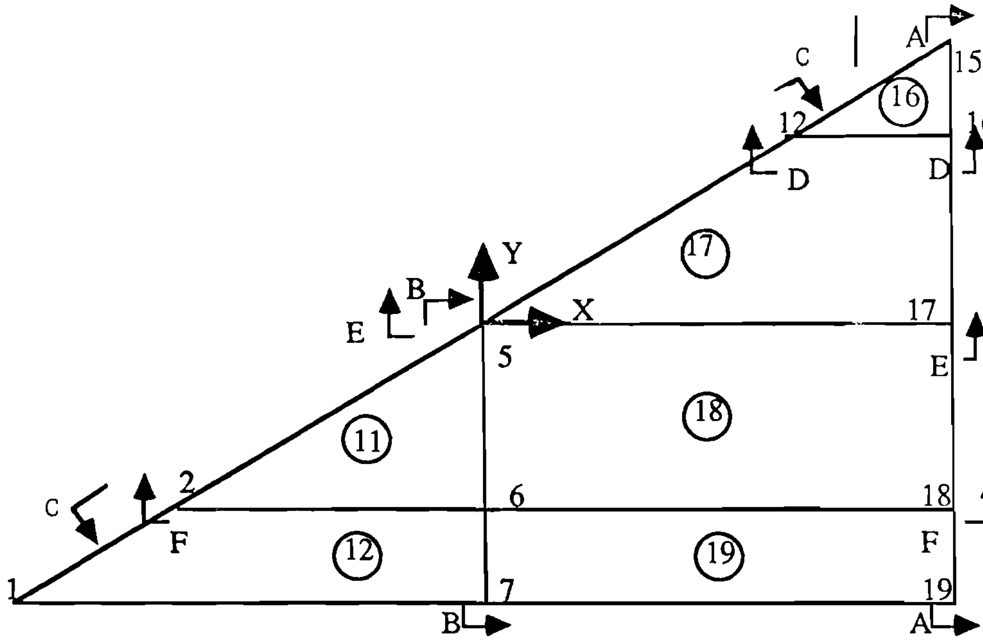
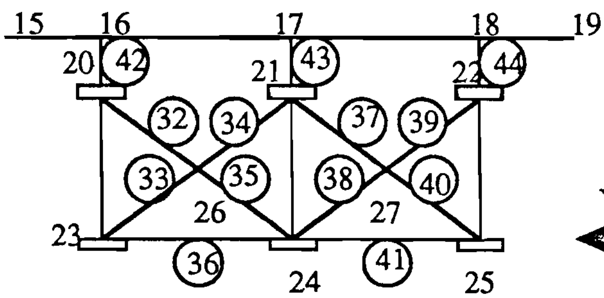
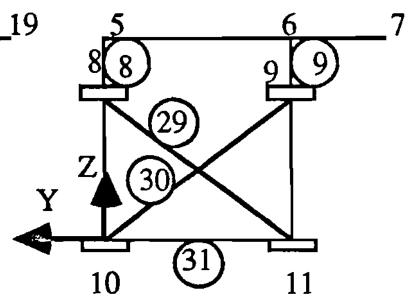


Figure 4.8, Plan view of slab Substructure 1 of Example 4



Section A-A



Section B-B

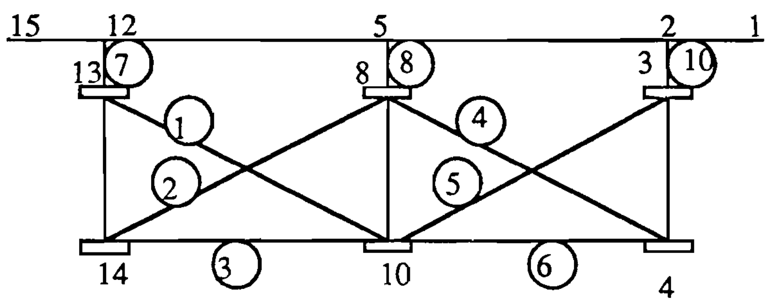


Figure 4.8, Section C-C

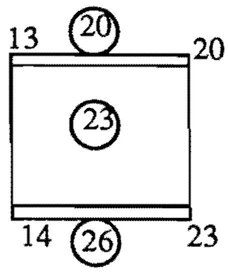
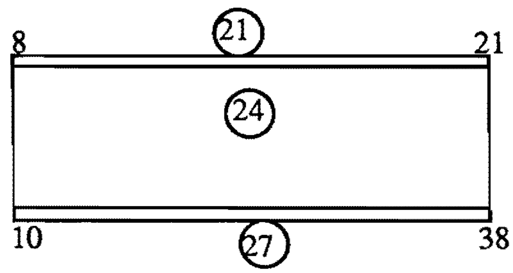


Figure 4.8, Section D-D



Section E-E

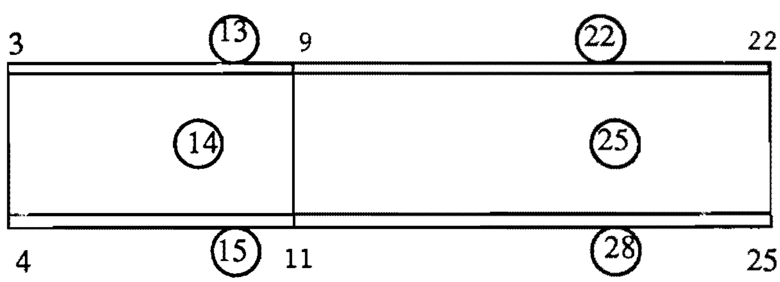


Figure 4.8, Section F-F

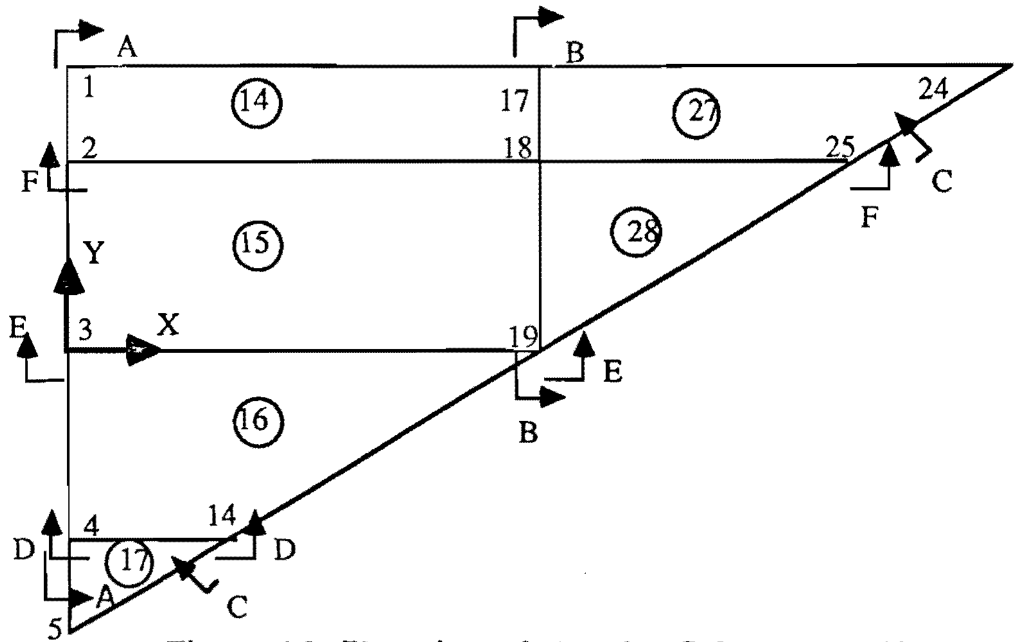
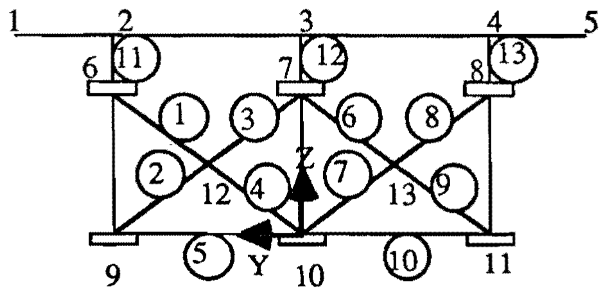
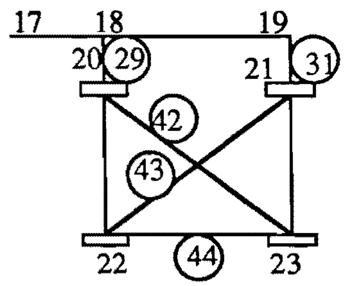


Figure 4.9, Plan view of slab for Substructure 13 of Example 4



Section A-A



Section B-B

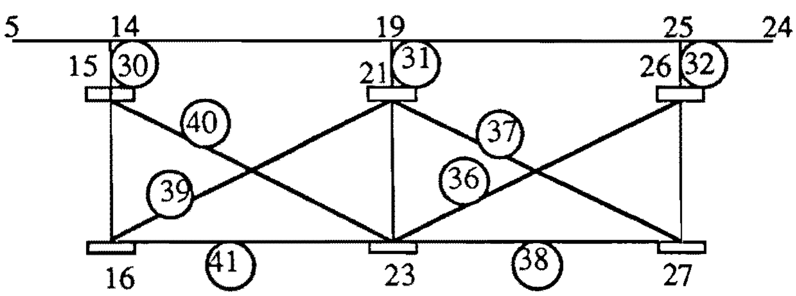


Figure 4.9, Section C-C

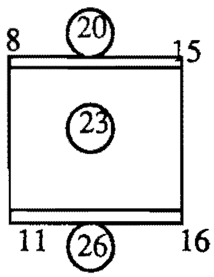
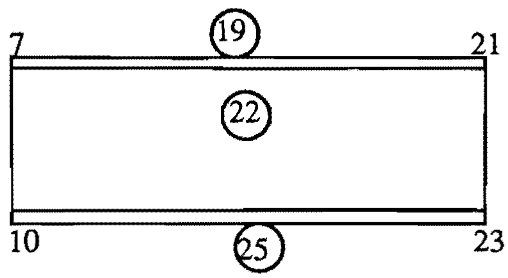


Figure 4.9, Section D-D



Section E-E

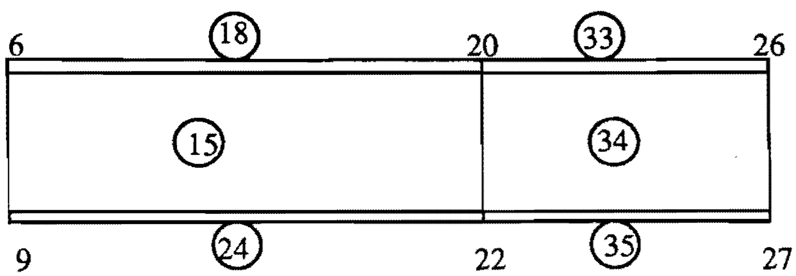


Figure 4.9, Section F-F

substructures 1 and 13 as one way slabs. The triangular elements must use the CLST stiffness and 0 needs to be input for these elements. The quadrilateral elements still use the QM5 stiffness and 1 is input.

The number of substructures per span does not change from example 3 since the data generator counts the number of substructures per span in the same way. One support still needs to be horizontally restrained and the user can input this at the end supports or it can automatically be done by the data generator. Since the interior support is horizontally restrained, NFIX = 7 and it is automatically done.

There is little less of a problem with the flange dimension changing since the flange changes can all be the same length. Even when the supports are skewed, the flange properties must still change at a diaphragm location. This is quite a severe restriction near negative moment supports since the flanges most likely do not change at diaphragm locations. When there are skewed supports and flange changes, the user can place the flange changes at diaphragm locations or put dummy diaphragms (zero physical properties) at places where the flanges change. The diaphragm spacing has to be increased at the end supports since all the supports are parallel. When a custom substructure is adjacent to a data-generated substructure then the nodes much all match up between substructures.

If a truck was placed on the end substructures then these concentrated loads must be input by hand. If the interior support did not line up in such a nice way then substructures 6 and 7 would have to be input according to Section 2.4.6 just like substructures 1 and 13.

4.6 EXAMPLE 5

Example 5 is a two-span, four-girder bridge with slightly skewed supports at all locations as shown in Figures 4.10, 4.11 and 4.12 This example shows the versatility of the program since, in addition to the skewed supports, the flanges, the diaphragm spacing and the radius all change for the bridge. Run the dead load due to concrete and steel load for noncomposite geometry. Also run two HS20-44 trucks with 14-foot axle spacing as shown in Figure 4.13b. Load case 3 is two HS20-44 lane loads placed for maximum positive moment for span one as shown in Figure 4.13a. A listing of the input is given in Appendix B. The following is given for example 5:

Centerline span lengths = 110.0 and 110.0 feet.
Centerline radius for bays 1 - 8 = 500.0 feet.
Centerline radius for bays 9 - 13 = 1,000.0 feet.
Diaphragm spacing for bays 2-6 and 8-12 = 16.0 feet.
Diaphragm spacing for bays 1,7 and 13 = 20.0 feet.
Concrete slab thickness = 7.5 inches.
Slab overhang = 3.0 feet each side.
Soffit = 2.0 inches.
Girder spacing = 8 feet 10 inches.
K- bracing for diaphragms.
Area of diagonal diaphragm braces = 2.860 in^2.
Area of horizontal diaphragm braces = 5.18 in^2.
Elastic modulus of steel = 29,000 ksi.
Elastic modulus of concrete = 3,625 ksi.
Density of Concrete = 0.150 k / ft^3.

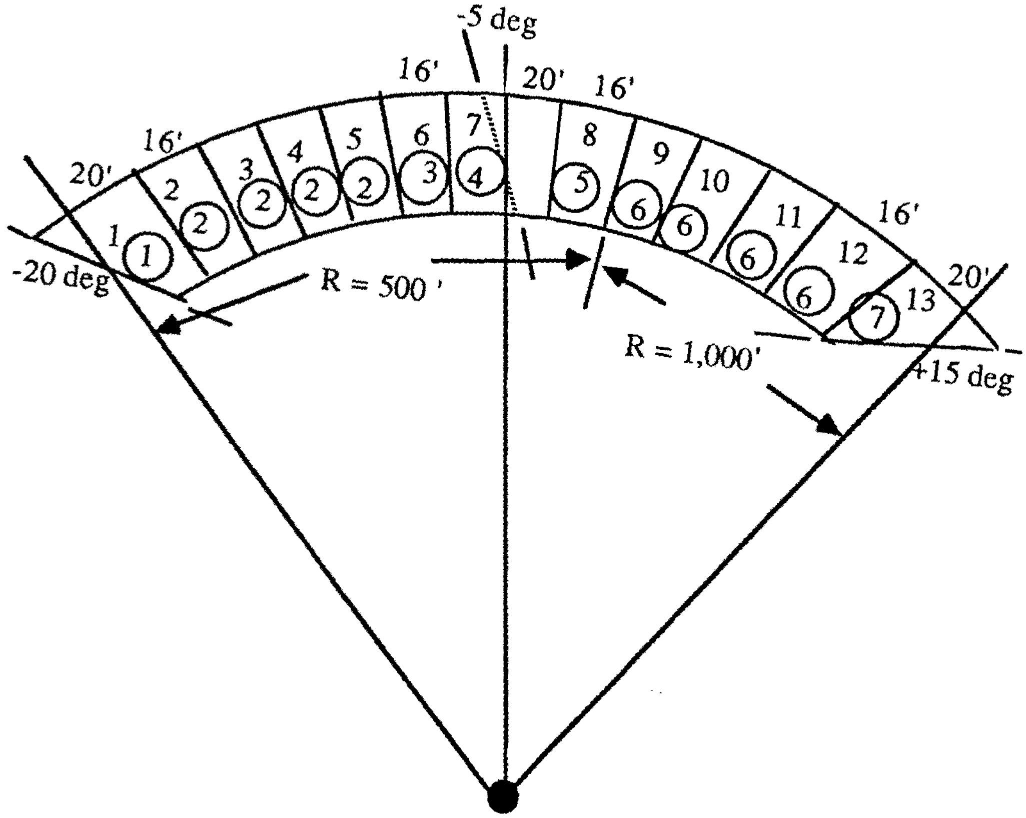


Figure 4.10 Plan view of Example 5

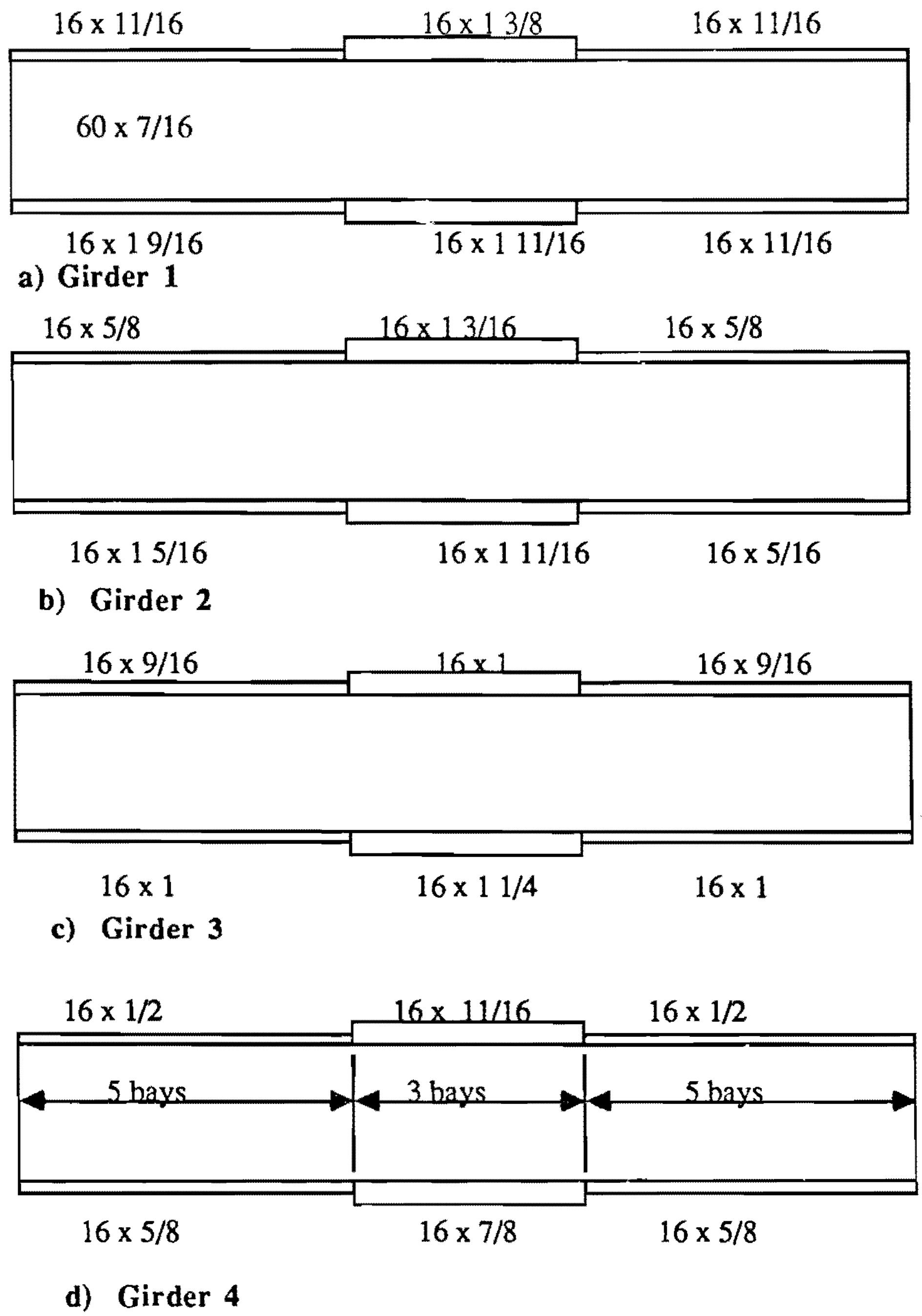


Figure 4.11 Girder elevation for Example 5

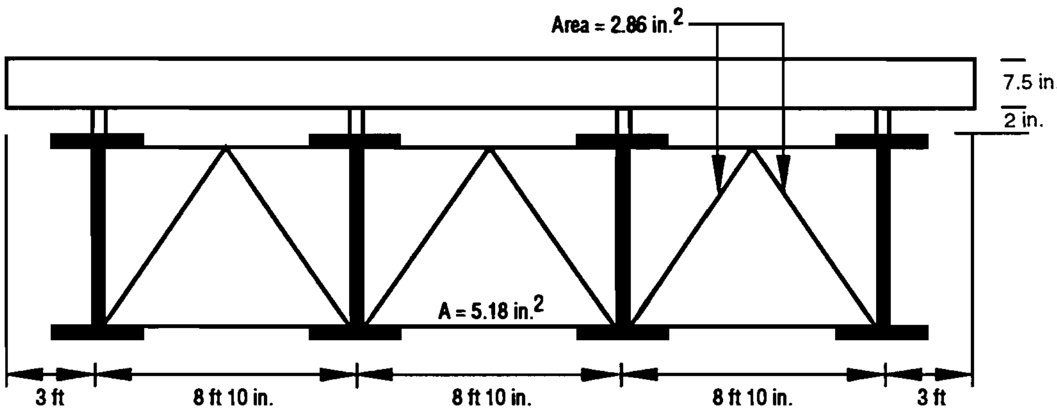


Figure 4.12 Cross-section of Example 5

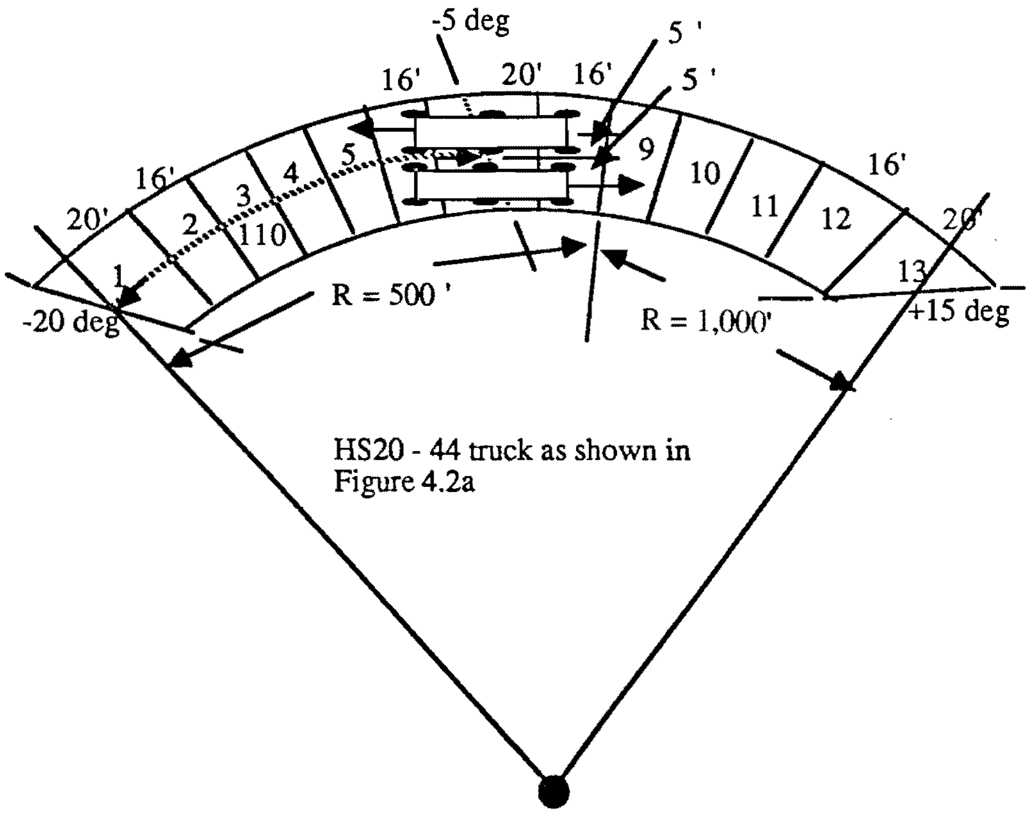
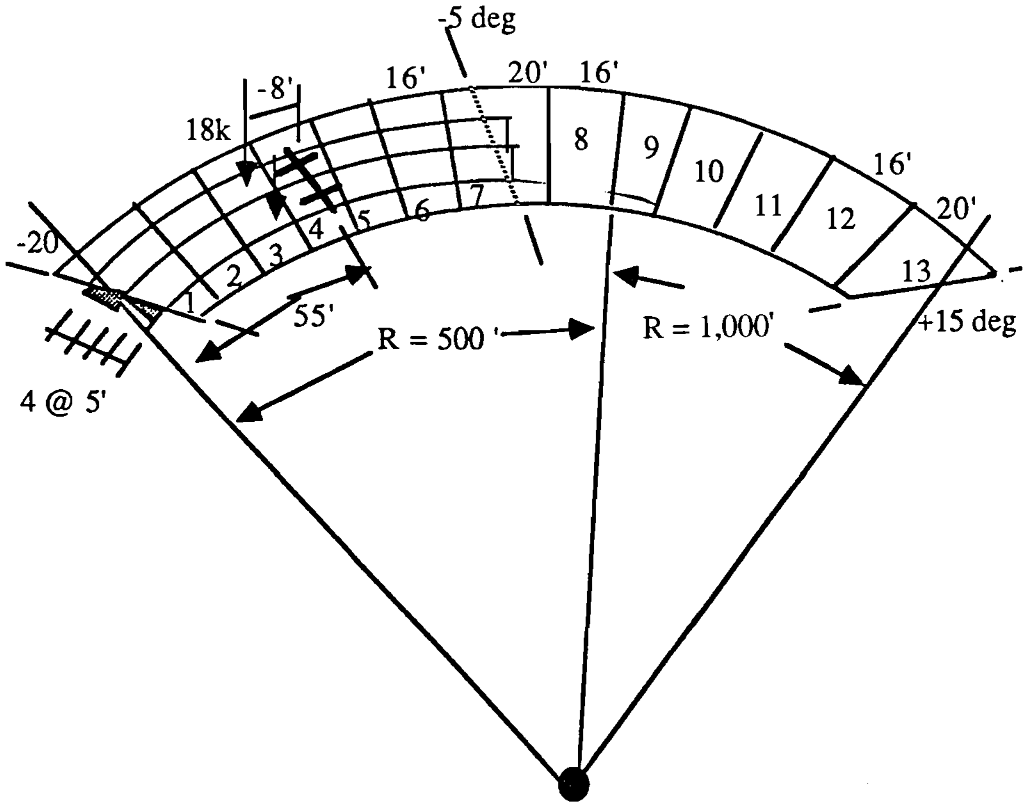


Figure 4.13a Truck placement for Example 5



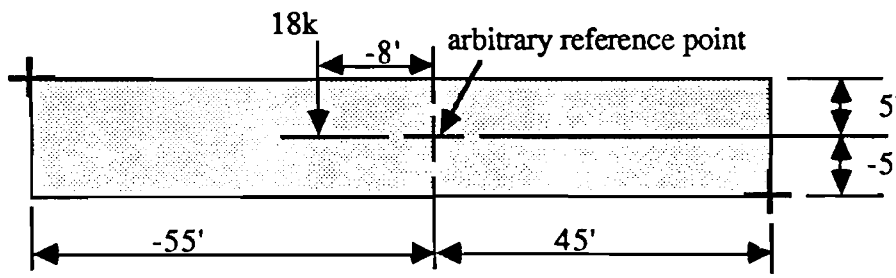


Figure 4.13b Lane load placement for Example 5

Density of Steel = 0.490 k / ft^3.
Poisson ratio for concrete = 0.17.
Poisson ratio for steel = 0.30.

4.6.1 Dead Load for Example 5

The data preparation for this bridge follows the other examples except for the skewed substructures. There are seven groups of substructures as shown by the circled numbers in Figure 4.10. The bridge has two spans and the second support can assumed to be horizontally restrained. From the definition of the number of substructures per span, there are 7 substructures for span 1 and 6 for span 2. NFIX should be equal to 7 to horizontally restrain the interior support.

Substructures 1, 7 and 13 have slightly skewed supports and require additional input of Section 2.4.4. The first thing to decide is the refinement of the mesh. Since the web depth is 5 feet, it seems reasonable to have two divisions between diaphragms and one between girders. The mesh of the skewed substructures must be decided by the user. The easiest decision is to use a linear change for the skew angle between the ends of the substructure. A linear distance along the centerline can be used as the location of the interior row of nodes. Shown in Figure 4.14 are examples of ways to divide up the mesh for substructures 1, 7, and 13.

The soffit is the distance from the top of the flange to the bottom of the concrete. The soffit and the average top and flange thicknesses determine the vertical placement of the nodes. The user must determine the average flange thicknesses for the entire bridge and determine an appropriate soffit dimension. The average top flange thickness was set equal to 0.8 inches and the average bottom flange thickness was set equal to 1.0 inches for this example.

For substructures 1, 7, and 13 prepare the data as in previous examples and add the information on the skew support placement and the skew of the mesh.

4.6.2 Truck Load For Example 5

The placement of the trucks is shown in Figure 4.13a and the truck is the same one as shown in Figure 4.2a. The program determines the truck load on the correct element even if the element has a skewed geometry. If the truck spans the place where the radius changes then two trucks must be input corresponding to the respective radii. Any wheel off the end of the bridge is ignored by the program so the truck placement can be partially off the end of the bridge without modifying the local truck geometry.

Since the two trucks are the same, the local geometry of the truck only has to be input once. The global placement of the two trucks are input and IBID is set equal to 1 so the local wheel locations and loads do not have to be input again.

Once the worst load case for the truck placement is determined these stresses can be summed with the dead load stresses if JREC = 2 and IRCSUM = 1. The truck placement in this example is not the worst load case but was used to show how to place more than one truck on a skewed mesh.

For live load, it must be decided how to model the concrete over the negative moment region. The program allows the elastic modulus to be set equal to different values in the transverse and longitudinal directions. To make the concrete ineffective in the longitudinal direction, the modulus in the x-direction should equal a small value. The concrete can be considered effective or ineffective in the transverse direction. Since a truck is over the negative support, the concrete modulus in the transverse direction was set equal to the proper value. The end result is that the slab functions as a one-way slab over the negative support. For

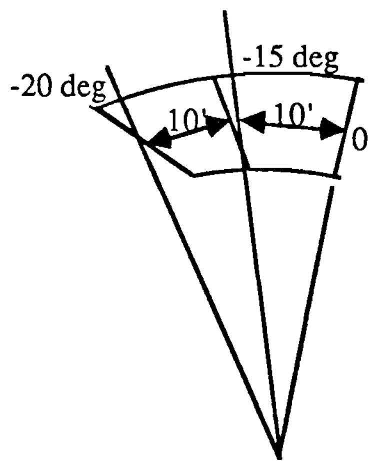


Figure 4.14a Substructure 1 for Example 5

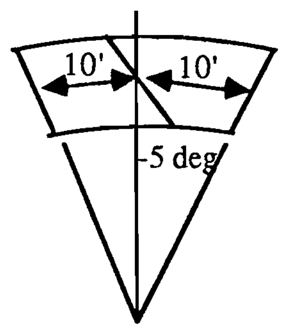


Figure 4.14b Substructure 7 for Example 5

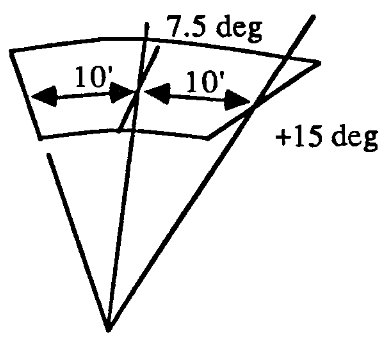


Figure 4.14c Substructure 13 for Example 5

simplicity's sake, the concrete for substructures 6, 7, and 8 was considered ineffective in the longitudinal direction.

4.6.3 Lane Load For Example 5

The lane load is place as a pressure load of 0.064 ksf with the upper left corner and lower right corner location as shown in Figure 4.13b. Also, concentrated loads of 18 kips are placed in much the same way as truck wheel loads are placed. Figure 4.13b shows that the lane load is assumed to curve and the ends are radial. As in the truck loads, the program ignores any loads not on elements. The lane load placement was originally developed based on radial diaphragms. With a skewed end diaphragm the shaded area may or may not be ignored based on the skew and the lane load placement. Thus, the lane load should be placed as far to the left of substructure 1 in order to cover the bridge. For placement of the lower right corner, the lane load is still radial so that some lane load is on both sides of the second support. When the support skew is slight, the error in this approximation is not seen to be too significant.

4.7 EXAMPLE 6

Example 6 is shown in Figure 4.15 and is identical to example 5 except that the radius for bays 1 thru 8 is a negative 500 feet. This reverse curvature appears to be quite complicated but the exact same input is needed for example 5 as in example 6 as shown in Appendix B. The only values that need to be changed are the positive 500-foot radius for the bridge geometry to a negative 500 feet. Also, the radius of the global placement of the truck and lane loads need to be changed to a negative 500 feet. Centerline span lengths = 110.0 and 110.0 feet.

Centerline radius for bays 1 - 8 = -500.0 feet.
Centerline radius for bays 9 - 13 = 1,000.0 feet.
Diaphragm spacing for bays 2-6 and 8-12 = 16.0 feet.
Diaphragm spacing for bays 1,7 and 13 = 20.0 feet.
Concrete slab thickness = 7.5 inches.
Slab overhang = 3.0 feet each side.
Soffit = 2.0 inches.
Girder spacing = 8 feet 10 inches.
K- bracing for diaphragms.
Area of diagonal diaphragm braces = 2.860 in^2.
Area of horizontal diaphragm braces = 5.18 in^2.
Elastic modulus of steel = 29,000 ksi.
Elastic modulus of concrete = 3,625 ksi.
Density of Concrete = 0.150 k / ft^3.
Density of Steel = 0.490 k / ft^3.
Poisson ratio for concrete = 0.17.
Poisson ratio for steel = 0.30.

4.8 EXAMPLE SUMMARY

These six examples demonstrate the wide variety of bridges which can be solved with this program. With the exception of example 4, very little input is required for any of these examples compared to other

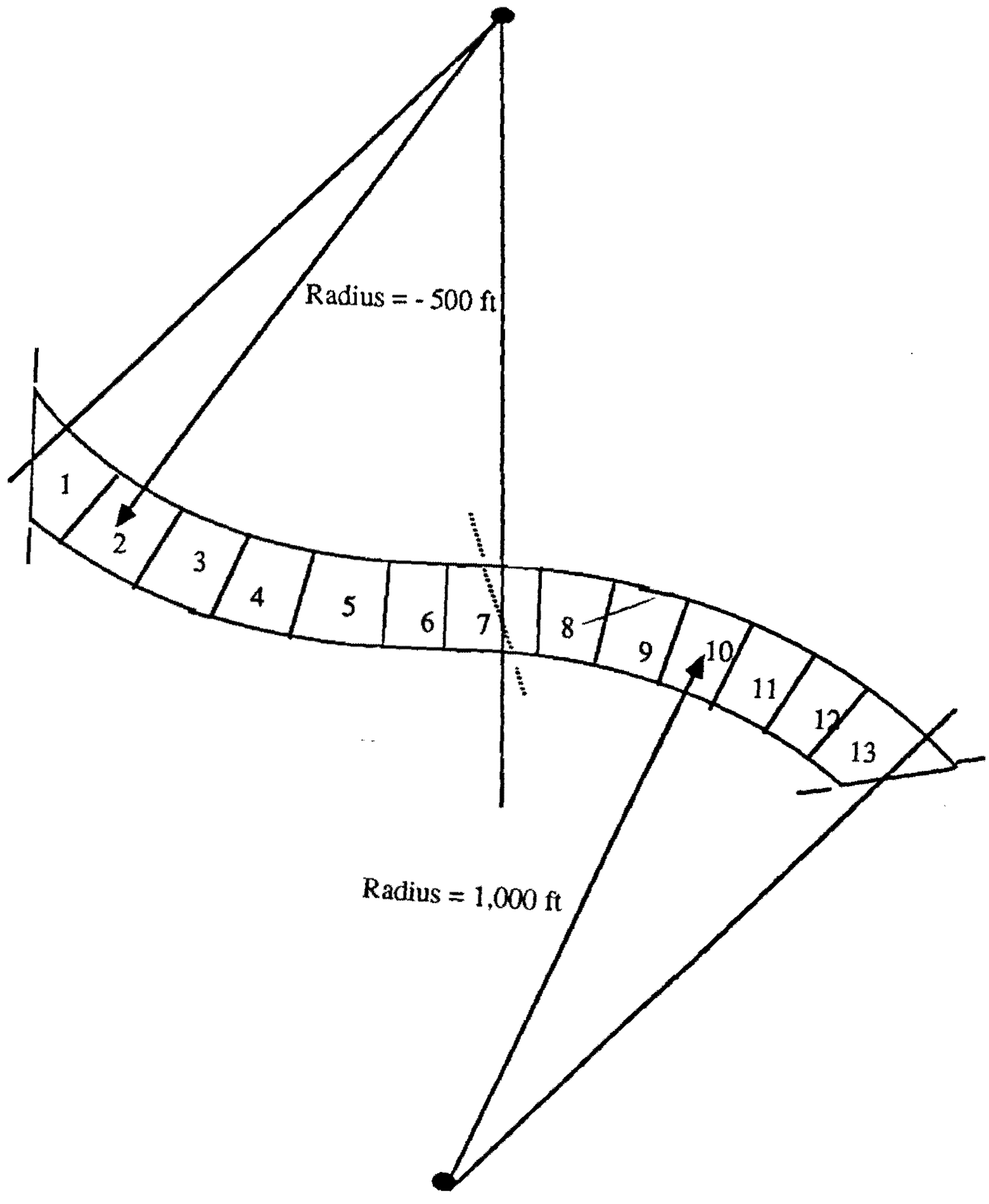


Figure 4.15 Plan view of reverse curvature of Example 6

computer programs. It may be best to approximate designs with the V-load method and finalize the design with this program. The other part of this research project was to automate the V-load design on a personal computer. A data file for the finite element solution can be directly taken from the data required for the V-load program. In this way, the data preparation need only be done once. A comparison of the accuracy of the V-load method for different bridge parameters is given in Chapter 5.

CHAPTER 5. PARAMETER STUDY AND PROGRAM VERIFICATION

5.1 INTRODUCTION

This chapter compares two bridge geometries with changing parameters to the V-load method. This was done to verify that each program worked for these geometries and to study the behavior of curved bridges with different parameters. In this way, the range of parameters could be established for the V-load method for acceptable error. Also this chapter uses a third bridge to verify the accuracy of the finite element analysis with two other finite element programs.

The three bridges studied were:

1) The bridge of example 1 in Chapter 4. The radius was changed from 500 feet to 1,000 feet for both the dead load and truck load cases. The diaphragm spacing also changed from 10 to 20 feet for each radius.
2) A three-span curved bridge with a 32.7-degree curve. The four-girder bridge plans were supplied by the Texas Highway Department and represent an actual design. Three radii, different diaphragm spacings, and radial and skewed supports were used to study different bridge parameters in a comparison with the V-load method. Both dead load and truck loads were used on some of the parameter studies.
3) The bridge of example 2 in Chapter 4. The three-span bridge plans were also supplied by the Texas Highway department. In this case, two finite element solutions were available for comparison and the program could be better verified.

5.2 PARAMETER STUDY ON A SIMPLE SPAN CURVED BRIDGE

This bridge was termed scheme A and was chosen because of its simplicity in geometry. With the simple geometry, curved bridge behavior could be more easily understood. The bridge was also a good validation of both the V-load and the finite element programs in that the radius could be increased to a large value and the results could be computed by hand since the bridge is statically determinate. The spacing of the girders was chosen to coincide with the wheels of the truck so that distribution factors would not be involved when comparing the V-load and the finite element results.

The results of the parameter study are shown in Table 5.1. The table shows that two radii of 500 and 1,000 feet with two diaphragm spacings of 10 and 20 feet were used as a simple parameter study on the 100-foot bridge. The values studied were the maximum stress due to dead load, worst case live load, and total load. Also, the warping stresses were determined for the dead load and total load case.

The table shows that as the radius decreased, more of the load was transfered from the inner to the outer girder for a given diaphragm spacing. This transfer of load was independent of the diaphragm spacing. The transfer of load from the inner to outer girder was much less evident for the composite girder in the live load case. The live load case was different because the slab provided torsional stiffness and this stiffness dominated over the steel girder stiffness.

The radius change also had a big influence on the warping stresses. When the radius was halved, the warping stresses almost doubled. Also, when the diaphragm spacing was doubled, the warping stresses almost doubled. The function of the diaphragms was to brace the girders and to make the bridge torsionally stiffer. As the load transfer was increased from the inner to outer girder due to a decrease in radius, the diaphragms had

Table 5.1 Maximum stresses on bottom flange for Scheme A (ksi)

Radius = 1,000 ft, Diaphragm spacing = 10 ft.

	OUTER GIRDER					INNER GIRDER				
	DL	LL	Tot	WDL	WTot	DL	LL	Tot	WDL	Wtot
FE	23.6	12.34	35.95	-3.9	-5.12	11.77	9.57	21.34	-3.1	-4.1
V	24.37	14.70	39.07	-1.80	-3.48	11.58	7.53	19.11	-.88	-1.72

Radius = 500 ft, Diaphragm spacing = 10 ft.

	OUTER GIRDER					INNER GIRDER				
	DL	LL	Tot	WDL	WTot	DL	LL	Tot	WDL	Wtot
FE	29.62	13.76	43.88	-8.40	-11.11	5.88	8.21	14.10	-5.62	-7.50
V	30.82	18.32	49.14	-4.10	-8.52	5.25	3.97	9.22	-.76	-1.65

Radius = 1,000 ft, Diaphragm spacing = 20 ft.

	OUTER GIRDER					INNER GIRDER				
	DL	LL	Tot	WDL	WTot	DL	LL	Tot	WDL	Wtot
FE	23.87	12.38	36.25	-6.95	-10.20	11.95	9.70	21.65	-4.40	-6.99
V	24.21	14.63	38.83	-3.85	-7.39	11.74	7.60	19.34	-1.84	-3.75

Radius = 500 ft, Diaphragm spacing = 20 ft.

	OUTER GIRDER					INNER GIRDER				
	DL	LL	Tot	WDL	WTot	DL	LL	Tot	WDL	Wtot
FE	29.92	13.72	43.64	-15.82	-22.91	6.02	8.41	14.43	-7.15	-11.73
V	30.51	18.17	48.68	-9.84	-18.64	5.56	4.41	9.67	-1.71	-3.86

DL = Dead load due to concrete and steel on non-composite girder.
LL = Maximum stress due to single truck load on composite girder.
Tot = Sum of DL and LL
WDL = Maximum warping normal stress on non-composite girder.
WTot = Sum of warping normal stress due to DL and LL.

to transfer more of this load. Many things affect warping stresses and this example showed that the stresses can be significant.

The V-load compared well against the finite element solution for bending stresssses, especially in the dead load noncomposite case. The dead load bending stresses were predicted with an error of less than 4%. The live load error was around 18% for the 1,000-foot radius and 32% for the 500-foot radius. The total error for the total stress was still less than 10%. The reason the live load error was so high was the fact that the V-load method did not take into account the torsional slab stiffness in its model. The V-load transferred too much load to the outer girder in the composite case. This was not a conservative model since the equal load was taken off the inner girder. The outer girder would end up being overconservative and the inner girder would be unconservative.

The warping stresses did not compare too well between the two solutions. The place of maximum stress did not agree either. The finite element gave maximum stresses between diaphragms while the V-load gave maximum stresses at diaphragm locations. The V-load assumed the diaphragms provided a rigid support so the maximum moment would most likely occur at that point. The finite element solution probably gave higher numbers because the diaphragms did not act in such a rigid manner for this simply-supported, two-girder case.

A few other things may have accounted for the discrepancy between the V-load and the finite element solution. The results of the finite element were given as an axial stress for the flange and warping stresses at the flange nodes. The bending stress could be thought of as an average stress for the member. For positive moments, as in this example, this was not bad since the gradient was very low but for continuous spans the program would most likely not give the true peak negative moment stress because of the steep gradient. The warping stresses also had a steep gradient at the diaphragms but the finite element solution, like the V-load, gave the warping stresses at node locations. The V-load used more increments along the span to determine both the bending stresses and the warping stresses compared to the finite element model. A difference in results can be attributed to the difference in the refinement of the finite element mesh and the increments of the V-load.

5.3 PARAMETER STUDY ON A THREE-SPAN CURVED BRIDGE

The bridge for scheme B is shown in Figure 5.1 and was taken from plans supplied by the Texas Highway Department. A few simplifications were made to more easily model the bridge. First, the overhang was increased so that each girder would have the same amount of concrete on each side. Also the same flange sizes were used for each girder and the flange sizes in the positve moment region were made the same for each span.

Most of the same things studied for scheme A were studied in scheme B. The maximum positive and negative bending and warping stresses were determined for both the V-load and the finite element solutions. Case 1, Case 2, and Case 3 were used to study the effect of diaphragm spacing on the maximum stresses. Case 1, Case 4, and Case 5 were used to study the effect of the radius on the maximum stresses. Case 6 and Case 7 are shown in Figures 5.2 and 5.3 and were used to study the effect of skew on the maximum stresses. Also, the effect of live load on different radii bridges were used in Cases 1 and 4.

Table 5.2 shows the results in the study of the effect of the radius on the maximum bending stress. The first thing that is evident is that the stresses are low. Perhaps the bridge design was conservative and the simplifying assumptions made it more so. The maximum stress did not change much when the radius doubled

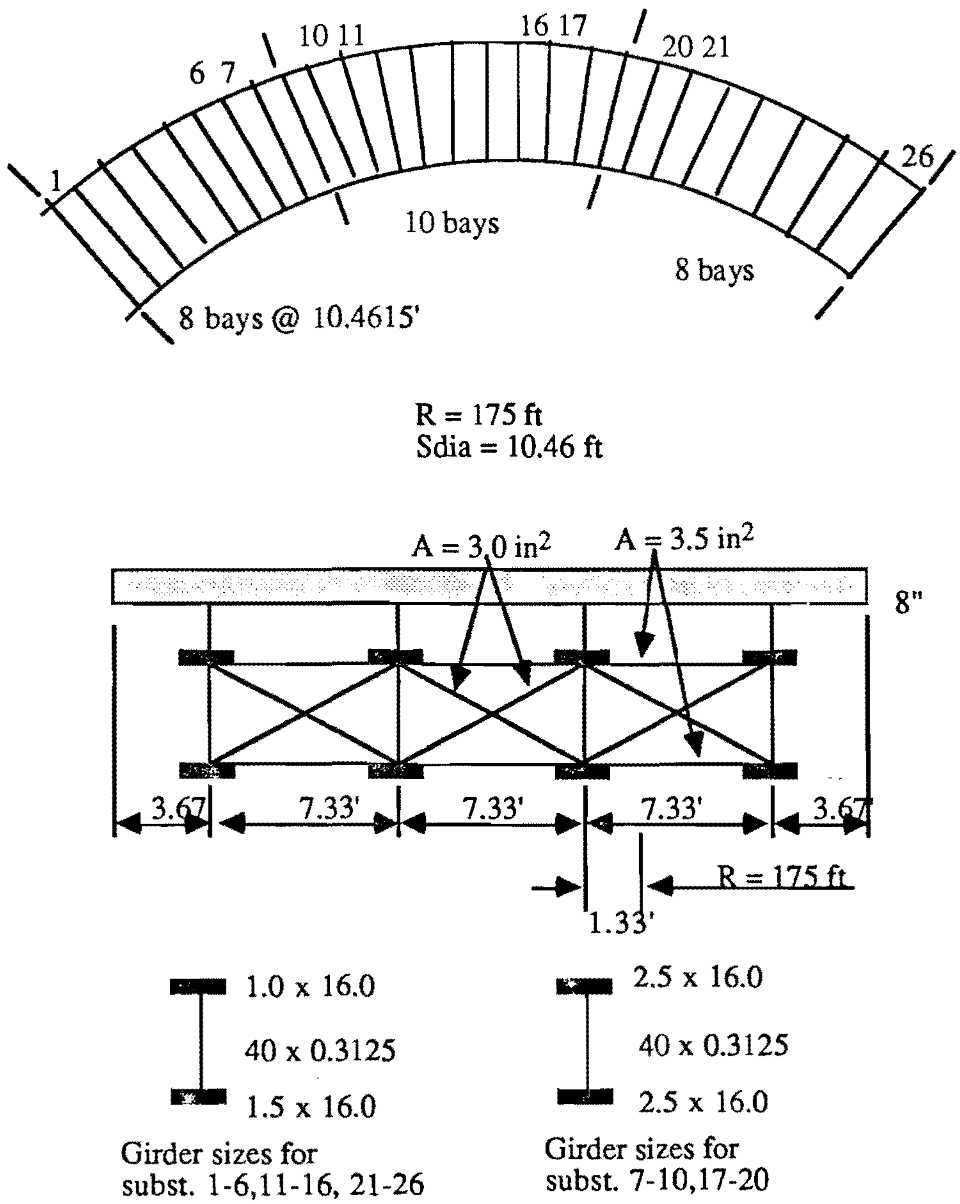
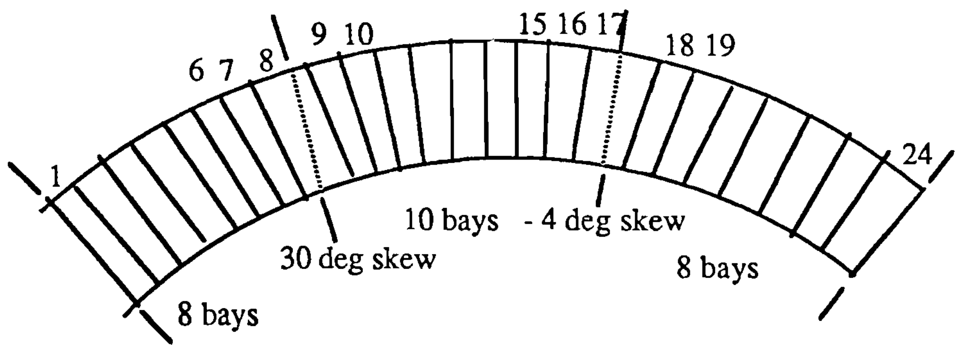
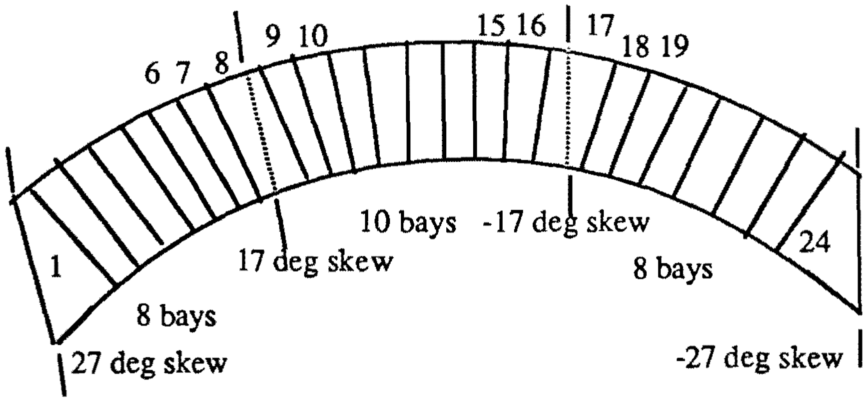


Figure 5.1 Case 1 of scheme B.



R = 175 ft
Sdia = 10.46 ft except for substructures
8 and 17 with Sdia = 20.92 ft

Figure 5.2 Case 6 of scheme B.



R = 175 ft
Sdia = 10.46 ft except for substructures
8 and 17 with Sdia = 20.92 ft

Figure 5.3 Case 7 of scheme B.

Table 5.2 Maximum bending stresses at the centerline of the bottom flange for Scheme B (ksi).

Case 1 Radius = 175 ft, Diaphragm spacing = 10.4615 ft

Load Case	Max Mom	Girder 1		Girder 2		Girder 3		Girder 4	
		V-load	Fem	V-load	Fem	V-load	Fem	V-load	Fem
DL	Pos. Neg.	7.27 -8.72	6.52 -7.47	5.56 -7.57	5.24 -6.90	4.03 -6.50	3.90 -6.12	2.61 -5.49	2.31 -5.29
LL1 LL2	Pos. Pos.		5.88 3.32		3.70 2.84		1.17 2.02		-1.46 0.88

Case 4 Radius = 350 ft, Diaphragm spacing = 10.4615 ft

Load Case	Max Mom	Girder 1		Girder 2		Girder 3		Girder 4	
		V-load	Fem	V-load	Fem	V-load	Fem	V-load	Fem
DL	Pos. Neg.	5.81 -7.64	5.57 -6.65	5.03 -7.10	4.88 -6.36	4.29 -6.57	4.20 -5.88	3.58 -6.06	3.45 -5.47
LL1 LL2	Pos. Pos.		5.29 2.78		3.37 2.58		1.30 2.17		-0.79 1.55

Case 5 Radius = 700 ft, Diaphragm spacing = 10.4615 ft

Load Case	Max Mom	Girder 1		Girder 2		Girder 3		Girder 4	
		V-load	Fem	V-load	Fem	V-load	Fem	V-load	Fem
DL	Pos. Neg.	5.15 -7.13	5.07 -6.36	4.77 -6.87	4.71 -6.14	4.40 -6.61	4.36 -5.98	4.05 -6.35	4.00 -5.78

LL1 = One truck loaded for maximum positive moment over girders 1 and 2.
LL2 = One truck loaded for maximum positive moment over girders 2 and 3.

from 175 to 350 feet, but there was a slight decrease when the radius increased from 350 to 700 feet. The V-load compared quite well for the dead load positive stress and a little worse for the dead load negative stress. The percent difference was always less than 10% for the dead load, however.

As stated in Section 5.2, the normal bending stresses for the flange were average stresses. Since the negative moment gradient at the support was steep, the true peak stress was most likely missed by the finite element due to averaging. The maximum stresses given in Table 5.2 represent extrapolations of the plot of bending stresses along the span. The difference between the stress printed out and the peak stress at the support in these cases varied up to 5%. The user is cautioned that the axial stresses printed out by the program should be considered as stresses at the midpoint between nodes.

Two live loads were run for both Case 1 and Case 4. The V-load marched a truck across the bridge and determined the worst longitudinal load position for positive moment. The wheel spacing of the truck was chosen to be directly over the girders so that the V-load program could better determine the proper distribution factors. The truck was then placed over girders 1 and 2 for one load case and over 2 and 3 for the other load case. The results are shown in Table 5.2. The load placed on the outer two girders produced the maximum stresses and torsion in the bridge. This load case also caused a negative moment in girder 1. The truck over the middle girders controlled for the inner girders. Also, the doubling of the radius caused a slight decrease in the bending stresses for the outer girder. The change in stress with the increase in radii was less in the live load case than the dead load case because the slab provided so much stiffness. This live load study showed that the truck needs to be moved around a lot to get the worst case for each girder. Also, depending on the truck placement, the stress range on the girders can change and this is important in fatigue design. The V-load can help in this placement but it has distribtion factors built in the program which may or may not help give the worst load case.

Table 5.3 shows the results of the effect of different diaphragm spacings on the warping stresses. As can be expected, the largest warping stresses occured when the diaphragm spacing was the largest. A dramatic decrease occurred when the diaphragm spacing was cut in half from 20.92 feet to 10.46 feet. A further decrease in warping stresses occurred when the diaphragm spacing was decreased to 6.97 feet. It was evident that there was a diminishing point of returns for decreasing the diaphragm spacing.

Table 5.3 shows that the V-load was quite good for determining the warping stresses when compared to the finite element solutions. It can also be noted that the warping stresses were only significant for a diaphragm spacing of 20.92 feet. This large diaphragm spacing probably exceeded what most designers would recommend and it was double the designed spacing in this case. The V-load agreed better for the continuous case perhaps because the four girders and the continuous spans forced the diaphragms to act in a more rigid manner.

A plot of the finite element warping stresses along half the span for girders one and four is shown in Figure 5.4. The three lines on each plot represent the three diaphragm spacings studied. A couple of things are evident from the plot. First, the dramatic difference in warping stresses for each diaphragm spacing is evident. The 20.92-foot diaphragm spacing dominates in relation to the other diaphragm spacings. The diaphragm serves as a reaction point for the flange as shown in Figures 1.2. 1.4 and 1.5b. When there is a positive bending moment as in Figure 1.4, the axial stresses in the flange are not collinear and the flange is supported by the diaphragm. Tension stresses were developed on the outer point of the flanges when the moment was positive. This is shown in Figure 5.4 which a positive sign represents a tension on the outer fiber of the flange. The plot resembles a continuous span with supports at diaphragm locations. The first interior

Table 5.3 Maximum warping stresses of the bottom flange for Scheme B (ksi).

Case 1 Radius = 175 ft, Diaphragm spacing = 10.4615 ft

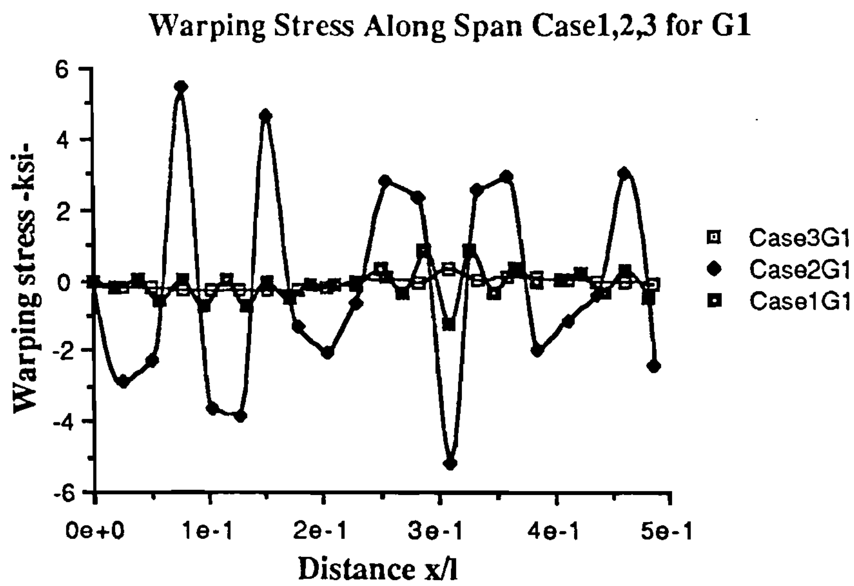
Load Case	Sign	Girder 1		Girder 2		Girder 3		Girder 4	
		V-load	Fem	V-load	Fem	V-load	Fem	V-load	Fem
DL	Pos. Neg.	1.75 -1.81	1.42 -1.57	1.30 -1.50	1.24 -1.28	0.90 -1.22	1.06 -1.01	.56 -.97	0.89 -.73

Case 2 Radius = 175 ft, Diaphragm spacing = 20.923 ft

Load Case	Sign	Girder 1		Girder 2		Girder 3		Girder 4	
		V-load	Fem	V-load	Fem	V-load	Fem	V-load	Fem
DL	Pos. Neg.	6.58 -6.40	5.46 -5.16	4.93 -5.19	3.80 -4.52	3.48 -4.10	2.80 -3.64	2.22 -3.16	1.65 -2.80

Case 3 Radius = 175 ft, Diaphragm spacing = 6.97 ft

Load Case	Sign	Girder 1		Girder 2		Girder 3		Girder 4	
		V-load	Fem	V-load	Fem	V-load	Fem	V-load	Fem
DL	Pos. Neg.	0.77 -0.85	0.36 -0.27	0.57 -0.70	0.28 -.30	0.39 -0.58	0.22 -.32	0.27 -.46	0.24 -.36


Warping Stress Along Span Case1,2,3 for G1
Warping stress -ksi-
6
4
2
0
-2
-4
-6
0e+0
1e-1
2e-1
3e-1
4e-1
5e-1
Distance x/l
Case3G1
Case2G1
Case1G1

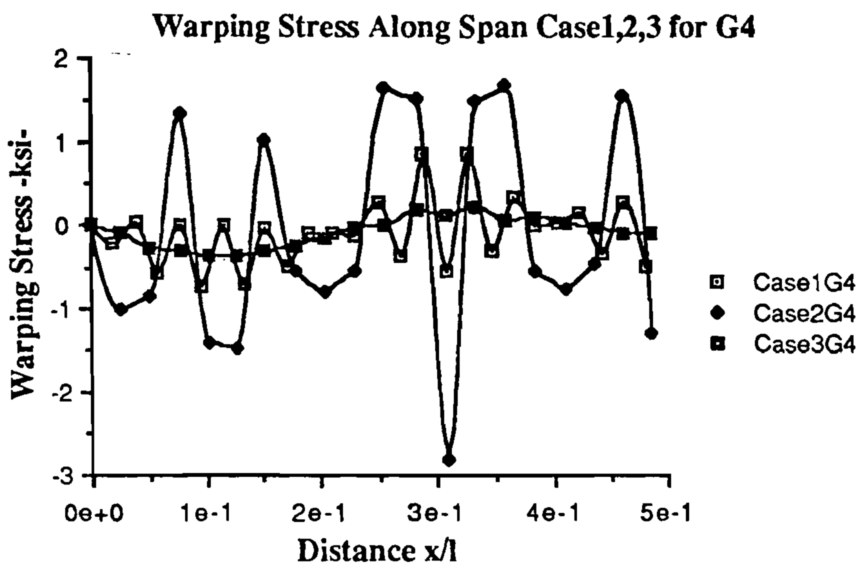

Warping Stress Along Span Case1,2,3 for G4
Warping Stress -ksi-
2
1
0
-1
-2
-3
0e+0
1e-1
2e-1
3e-1
4e-1
5e-1
Distance x/l
Case1G4
Case2G4
Case3G4

Figure 5.4

support is near 0.31 and the stress in the outer fiber goes in compression in the negative moment region. The plot for the inner girder looks much like the one for the outer girder except the warping stresses are smaller. This is most likely due to the load transfer to the outer girder due to the curvature.

Plots of the stresses for girder 1 and 4 of case 6 and case 7 along half the span is shown in Figure 5.5. The plots show how dramatic a difference the skewed supports make. The top graph pertains to the bridge in Figure 5.2. The positive moment for the outer girder was greater than inner girder in the exterior span and less in the interior span. The maximum stress for the inner girder of the interior span could be attributed to the sharp curvature of the bridge. Both plots showed a local stress variation near the inflection point of the other girder. The infection point of girder one would require no load transfer to the outer girder since the flange forces are balanced. This was the reason that there was a slight increase in stress in girder 4 when girder 1 was at its inflection point. With radial supports, these inflection points would line up transversely and this local maximum would not be as evident. The warping stresses for the skewed supports were pretty high near the skewed supports but dropped off to low levels away from the support. This is understandable since this model had the diaphragms farther apart due to the curvature. Also, diaphragms were not placed over the support so that the diaphragms would only be radial. Consequently, the unsupported length of the warping of the flange doubled and the warping stresses increased sharply. A different modeling of the diaphragms at the skewed support would change the warping stresses.

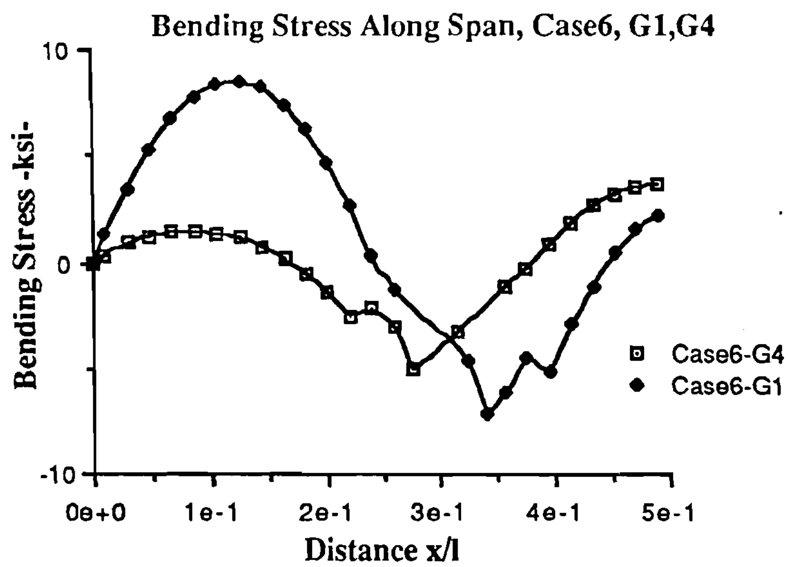
5.4 KURV87 COMPARISON WITH TWO OTHER MODELS

Because the V-load method was an approximate method, a more stringent comparison of the finite element was needed. The Texas Department of Transportaton had determined a typical bridge to analyze and it represents Example 2 in Chapter 4.

The bridge division used the Shell 8 program to develop two finite element models to analyze the bridge. The first model used plate elements to model the flange, slab and web elements. This model was deemed to be not too reliable since the aspect ratio for the flange elements was over 20. The second model used beam elements for the flanges in much the same way as Kurv87 did. The difference between these two models was the slab placement. The slab was placed at the same location as the flange in the bridge division model while Kurv87 used rigid links as shear studs and placed the concrete slab physically away from the top flange. This difference should not have been important for the dead load noncomposite case since the slab was only supplying load and no stiffness.

In the course of the model comparison it was decided to look at the effect of the bottom lateral braces. The bridge division ran its truss model with and without the bottom bracing as did the Kurv87 program. As stated in Chapter 4, the bridge division's truss model used braces that switched back and forth in every bay. For easier modeling, Kurv87 used X-bracing in each bay with half the area so that the dead load was the same and the stiffness was about the same. The plate model was not run without the braces.

The results of the study are shown in Table 5.4. The maximum deflection predicted by the Kurv87 program fell in the middle of the two maximum deflections predicted by the truss and plate models. An average percent difference for the Kurv87 program and the truss model was about 1.8% for the exterior span. An average percent difference for the Kurv87 program and the plate model was about -2.7%. The Kurv87 program predicted a little less rotation than the other two models in exterior span. The deflections predicted by the Kurv87 program in the interior span was greater than the other two models. The numbers were still relatively close, however.


Bending Stress Along Span, Case6, G1,G4
10
0
-10
0e+0
1e-1
2e-1
3e-1
4e-1
5e-1
Distance x/l
Bending Stress -ksi-
Case6-G4
Case6-G1

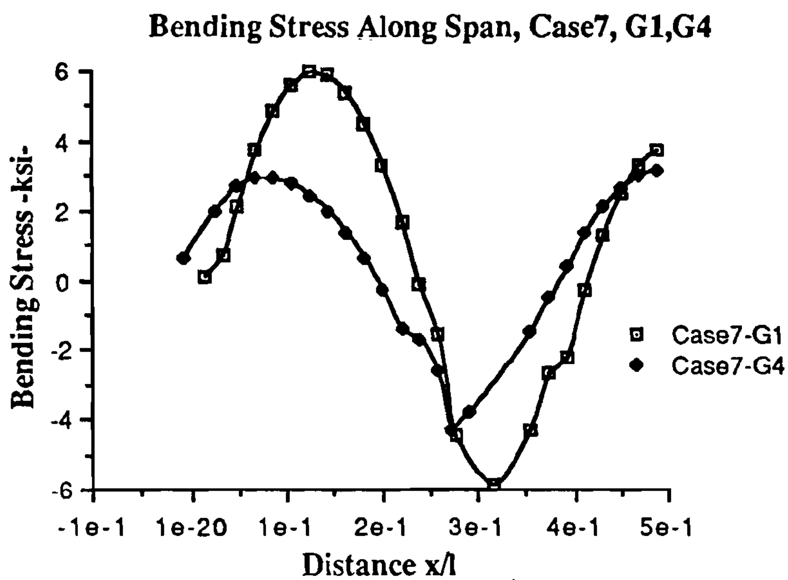

Bending Stress Along Span, Case7, G1,G4
6
4
2
0
-2
-4
-6
-1e-1
1e-20
1e-1
2e-1
3e-1
4e-1
5e-1
Distance x/l
Bending Stress -ksi-
Case7-G1
Case7-G4

Figure 5.5

Table 5.4 Deflections computed by various finite element models for scheme C (ft)

Maximum exterior span deflection

Girder	Shell 8 Truss with brace	Shell 8 Truss w/o brace	Shell 8 Plate w/o brace	Kurv87 with brace	Kurv87 w/o brace
1	-0.563	-0.563	-0.548	-0.553	-0.565
2	-0.471	-0.472	-0.461	-0.469	-0.475
3	-0.388	-0.382	-0.371	-0.385	-0.387
4	-0.290	-0.292	-0.288	-0.302	-0.298

Maximum interior span deflection

Girder	Shell 8 Truss with brace	Shell 8 Truss w/o brace	Shell 8 Plate w/o brace	Kurv87 with brace	Kurv87 w/o brace
1	-0.080	-0.081	-0.097	-0.113	-0.103
2	-0.113	-0.114	-0.125	-0.138	-0.133
3	-0.146	-0.147	-0.154	-0.163	-0.163
4	-0.179	-0.179	-0.183	-0.188	-0.193

Highway truss model uses beam elements for flanges and diaphragms and plate elements for the slabs and webs.
Highway plate model uses plate elements for the flange, slab and web elements.
With and without the brace refers to bottom lateral bracing for the bottom flange.

The bracing had little influence on the deflection of the members. The brace seemed to stiffen up the bridge torsionally because the rotation was slightly less. The difference in modeling the X-bracing did not seem to represent a problem except that the deflections between the two models were slightly closer without the brace than with the brace. The effect of the bracing probably did not show up too much since the 5-degree curvature was pretty low. The bottom lateral bracing most likely would have a much larger effect for a sharper curved bridge.

5.5 SUMMARY

The effect of diaphragm spacing, radius, skew, and loading was studied for both bending and warping stresses. The following conclusions can be drawn:

1) When the radius decreased, the warping stress and the load transfer to the outer girder increased.
2) When the diaphragm spacings increased, the warping stresses increased.
3) The V-load was very good for predicting dead load stresses but it was not nearly as good at predicting live load stresses since it ignored the contribution of the slab stiffness. The error for the sum of the dead load and live load stresses was still less than 10%, however.
4) The V-load predicted warping stresses better for the four-girder, three-span bridge than for the two-girder simple span bridge.
5) It is difficult to make general statements on the skewed bridges except for some obvious statements. The maximum bending stresses near the skewed support did not occur at the same distance away from the support or even on the same relative side of the support. The warping stresses in this model were much larger near the support due to a larger diaphragm spacing.
6) The Kurv87 program will give the proper peak stresses for the positive moment region but it will not give the true peak stress at a support. The user must extrapolate the correct stress at the support or run a fine mesh for a final design check.
7) The Kurv87 program compared extremely well with two other finite element models and the user can be confident in the results obtained.
8) The effect of the bottom lateral bracing was not seen to be too significant in the bridge studied even though a little less rotation was evident. The influence of the bottom bracing most likely will be more evident for more sharply curved bridges.

CHAPTER 6. CONCLUSIONS AND RECOMMENDATIONS

6.1 CONCLUSIONS

The Kurv87 program developed at The University of Texas at Austin provides an easier way to analyze curved as well as straight steel bridges. The program easily handles the following types of problems:

1) Erection procedure in which multiple stages can be superimposed on each other.
2) "S" curved bridges with intermittent straight segments.
3) Support settlements.
4) Specified support spring stiffnesses.
5) Multiple trucks for a single load case.
6) Multiple lane loads for a single load case.
7) Slight skew for interior and exterior supports.
8) Severe skew of interior supports.
9) Curb loads and superimposed surface loads.
10) Dead loads on non-composite sections superimposed on live loads on composite sections.
11) Orthotropic slab properties over negative moment regions.
12) Three different diaphragm configurations with or without bottom lateral bracing.

The program also provides the capacity to add custom substructures between data-generated ones. Any loads not covered can be input on any element in the bridge. This would most likely cover wind or impact loads. In this way, most any possible bridge can be analyzed on varying degrees of input difficulty.

The strength of this program is that designs can be easily modified to account for the iteration in the design process. Along with the ease in changing the design, the user can study the effects of different parameters on bridge behavior so that the most efficient structure can be designed. The user can also study different erection schemes to determine the most efficient erection procedure for the bridge.

Along with the ease in input, the output is also very easy to interpret. The stresses are output by element types. These five types are: 1) rigid link elements, 2) diaphragm elements, 3) flange elements, 4) web elements, and 5) slab elements. In addition, a plot of concrete slab stresses can be made for the composite case. The user can concentrate on looking at one type of element like the flanges and ignore the rest of the ouput. The user can thus develop the model and analyze the results in a reasonably short time.

A small premium is paid with all this simplicity. Any change of the bridge can only occur at diaphragm locations. The most stringent requirement probably applies to flanges which must change at diaphragms locations. This is not too bad for radial bridges since the diaphragm spacing or flange changes can be moved so that they coincided with each other. The worst case is for skewed supports in which the flanges change on a line parallel with the support while the diaphragms remain radial. Approximations to the model or custom substructures must be used in these cases.

The use of this program requires a respectable level of competence in finite element analysis on the part of the user. Even with all the simplified input, the user must model the bridge correctly in order to get meaningful results.

6.2 RECOMMENDATIONS FOR FUTURE RESEARCH

Further research in the area of bridge analysis could involve modifying the Kurv87 program or developing a more powerful analysis tool. The following are recommendations on further research into bridge analysis programs:

1) Modify the data generator to generate both steel I-girder and steel box girder geometries.
2) Modify the data generator to generate a triangular substructure for severely skewed end supports.
3) Modify the data generator so the actual flange locations as opposed to the average flange thicknesses can be used in the vertical placement of the nodes.
4) Modify the data generator so that the girder depth does not have to be constant over the bridge. This most likely would be needed for the negative moment regions of straight bridges.
5) Enhance the program so that pinned connections can be used between girders. This would be useful in analyzing the erection stages.
6) Enhance the program to be able to handle multiple load cases.
7) Adapt a graphics package so that the input could be taken directly from another data file representing a description of the bridge.

A good foundation has been laid for further enhancements to this progam so that many aspects of bridge design can be easily and efficiently accomplished.

APPENDIX A

CHANGING THE COMPUTER PROGRAM

Listed in the following table are size limitaions which can be changed so that different sized problems can be solved.

LIMITATION	VARIABLES TO CHANGE	SUBROUTINE OR COMMON BLOCK
12 Wheels	Centtr(12), Centlg(12), Trld	Readtr in Genpuz
10 Spans	Nsubbc(10)	Common /Puz/
10 Girders	Sgir(10),Masin(70)	
Max. No. of divisions between diaphr = 8	Dthet(8)	Trdelt, Lanld and Truck in Genpuz
No. of Elements = 200	Itp(200),Kbeam(200),P Poc(24,16,200)	Common in Recpuz Loads,Gensol, Unit2 in in Genpuz
Maximum number of substructures = 16	Poc(24,16,200),Pr(24,16) X(450,16), Szq(3,200,16),Sgq(3,200,16)	Loads, Gensol, Unit2 Common in Puzf83 Common /dum/ in Recpuz
Number of Nodes (max number = 162)	Xq(162),Yq(162),Zq(162) R(162), Angle(162)	Common in Genpuz

To change the plate girder to include a box girder the following changes need to be made:

1) Change node locations in subroutine NODGEN.
2) Change element generator and element node numbering in subroutine ELNOD.
3) Make sure variables NXND, NXB and NXT give correct values.

To change the graphics the folliwing subroutines need to be changed:

1) Subroutine PLOTSG in Genpuz which calls subroutines PLOTXY and PLOTYZ.
2) Subroutine PLOTSL in Recpuz.

APPENDIX B
COMPUTER PROGRAM LISTING OF INPUT FILES FOR EXAMPLES IN CHAPTER 4

```
FILE -EXAMID-, DEAD LOAD 1 NONCOMPOSITE, CONCRETE + STEEL LOAD
 100 FT SIMPLE SPAN, RADIUS =  500 FT, MAR 25 , 1987 , TRUSS
    1     0     0     1     0     1     0     0     1
    1     0     5
    1
    5     2     4     2
    500.0        7.5        3.0        3.0       20.0
      3.0       -3.0
     50.0     0.375         0.5      0.875        0.0
      0.5       10.0      0.875       10.0
      0.5       10.0      0.875       10.0
    1     1     2     0
     1.430      0.10       0.10      0.001    1                 4
     2.590      0.10       0.10      0.001    2     3     5     6
       1.0       1.0       0.17       0.15
   29000.0       0.3       0.49
       0.0       0.0       0.0         0.0
0
```

```
FILE -EXAMIT-, THIS IS A TEST CASE FOR TRUCK LOADING, TRUSS
 100 FOOT SIMPLE SPAN, RADIUS =   500 FOOT, APR 13, 1987
    1    0    0    2    0    1    0    0    0
    1    0    5
    1
    5    2    4    2
    500.0       7.5       3.0       3.0      20.0
      3.0      -3.0
     50.0     0.375       0.5     0.875       0.0
      0.5      10.0     0.875      10.0
      0.5      10.0     0.875      10.0
    1    1    2    0
     1.430     0.10      0.10     0.001    1                   4
     2.590     0.10      0.10     0.001    2     3     5     6
    3625.0   3625.0      0.15      0.00
   29000.0      0.3      0.00
       0.0      0.0       0.0       1.0
    1
HS 20-44 0      1.0      54.0       0.0     500.0 FORW
    6
       4.0     3.00     14.00
       4.0    -3.00     14.00
      16.0     3.00       0.0
      16.0    -3.00       0.0
      16.0     3.00    -14.00
      16.0    -3.00    -14.00
0
```

```
HOUSTON BRIDGE DIVISION. 4 GIRDER BRIDGE FROM MR. ED SUCHICKI.
3 SPAN, 184.71 FT, 215.5 FT, 184.71 FT, 5 DEG CURVE. DL1 NON-COMPOSITE
   11     0     1     0     0     0     0     0     1
    3    12    12    14    12
    1
    3     1     1     4
1145.9156        8.0         3.0         3.0    15.3929
      12.0       4.0        -4.0       -12.0
      72.0    0.5625         0.0         0.0         0.0
       1.0      18.0         1.5        18.0
       1.0      18.0         1.5        18.0
       1.0      18.0         1.5        18.0
       1.0      18.0         1.5        18.0
    1     1     1     4
     1.525      0.01        0.01       0.001    1     2     3     4     5     6
     1.405      0.01        0.01       0.001
    1     2
    2     1
    3     4
    4     3
       1.0       1.0        0.17        0.15
   29000.0       0.3        0.49
       0.0       0.0         0.0        0.0
0
    2
    2
    1          4
   10          0
       1.0      18.0         2.0        18.0
       1.0      18.0         2.0        18.0
```

```
0
        1.0       18.0       2.0       18.0
        1.0       18.0       2.0       18.0
0
    2
    2
    1         3
   10         0
        1.0       18.0       1.5       18.0
        1.0       18.0       1.5       18.0
        1.0       18.0       1.5       18.0
        1.0       18.0       1.5       18.0
0
    2
    2
    1         4
   10         0
        3.0       18.0       3.0       18.0
        3.0       18.0       3.0       18.0
        3.0       18.0       3.0       18.0
        3.0       18.0       3.0       18.0
0
    2
    2
    1         3
   10         0
        1.0       18.0       1.5       18.0
        1.0       18.0       1.5       18.0
        1.0       18.0       1.5       18.0
        1.0       18.0       1.5       18.0
0
```

```
  2
  2
  1         4
 10         0
     1.0       18.0       2.0      18.0
     1.0       18.0       2.0      18.0
     1.0       18.0       2.0      18.0
     1.0       18.0       2.0      18.0
0
  2
  2
  1         3
 10         0
     1.0       18.0       1.5      18.0
     1.0       18.0       1.5      18.0
     1.0       18.0       1.5      18.0
     1.0       18.0       1.5      18.0
0
  2
  2
  1         4
 10         0
     3.0       18.0       3.0      18.0
     3.0       18.0       3.0      18.0
     3.0       18.0       3.0      18.0
     3.0       18.0       3.0      18.0
0
  2
  2
  1         3
 10         0
```

```
             1.0        18.0        1.5        18.0
             1.0        18.0        1.5        18.0
             1.0        18.0        1.5        18.0
             1.0        18.0        1.5        18.0
0
         2
         2
         1          4
        10          0
             1.0        18.0        2.0        18.0
             1.0        18.0        2.0        18.0
             1.0        18.0        2.0        18.0
             1.0        18.0        2.0        18.0
0
         2
         2
         1          3
        10          0
             1.0        18.0        1.5        18.0
             1.0        18.0        1.5        18.0
             1.0        18.0        1.5        18.0
             1.0        18.0        1.5        18.0
```

```
EXAMPLE 3, 2 SPAN, 3 GIRDERS, DEAD LOAD ONLY, NON-COMPOSITE
SEVERE SKEW INTERIOR SUPPORT, SPRING STIFFNESSES ARE SPECIFIED
    5     0     1     0     0     0     9     0     1
    2     7     7     6
    0     3   1.0E+30
    0     3   2.0E+30
    0     3   3.0E+30
    7     3   3.0E+30
    7     3   2.0E+30
    7     3   1.0E+30
   13     3   1.0E+30
   13     3   2.0E+30
   13     3   3.0E+30
    1
    5     1     2     3
       0.0        7.5        3.0        3.0       16.0
    8.8333        0.0    -8.8333
      60.0     0.4375       0.80        1.0        0.0
    0.5625       16.0        1.0       16.0
    0.5625       16.0        1.0       16.0
    0.5625       16.0        1.0       16.0
    2     2     2     0
     1.430       0.10       0.10      0.001     1     4
     2.590       0.10       0.10      0.001     3     6
       1.0        1.0       0.17     0.15
  29000.00        0.3       0.49
       0.0        0.0        0.0        0.0
0
```

```
     2
     2
     1            1
    10            0
    0.5625          16.0        1.00        16.0
    0.5625          16.0        1.00        16.0
       1.00         16.0        1.25        16.0
0
     4
     2
     1            1
    10            0
       1.00         16.0        1.25        16.0
       1.00         16.0        1.25        16.0
       1.00         16.0        1.25        16.0
     3    1
0
     2
     2
     1            1
    10            0
       1.00         16.0        1.25        16.0
     0.5625         16.0         1.0        16.0
     0.5625         16.0         1.0        16.0
0
     2
     2
     1            5
    10            0
     0.5625         16.0         1.0        16.0
     0.5625         16.0         1.0        16.0
     0.5625         16.0         1.0        16.0
0
```

```
EXAMPLE 4. 2 SPAN,3 GIRDERS. DEAD LOAD ONLY. NON-COMPOSITE
SEVERE SKEW ALL SUPPORTS, SPRING STIFFNESSES ARE SPECIFIED
    8    0    2    0    0    0    3    0    0
    2    7    7    5
    7    3  3.0E+30
    7    4  2.0E+30
    7    5  1.0E+30
    5
   49   17    1    0
   13   12
    5   14
   14   10
    8    4
    7   10
   10    4
   12   13
    3    8
    6    9
    2    5
    2    5    5
    1    7    6    2
    3    9
    4   11    9    3
    4   11
   12   15   13
    5   17   15   12
    6   16   17    5
    7   19   18    6
```

13	20									
8	21									
9	22									
14	23	20	13							
10	24	21	8							
11	25	22	9							
14	23									
10	24									
11	25									
8	11									
9	10									
10	11									
20	26									
23	25									
21	25									
24	25									
23	24									
21	27									
24	27									
22	27									
25	27									
24	25									
16	20									
17	21									
18	22									
1		-21.434	-11.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0
2		-16.000	-8.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0
3		-16.0	-8.833	5.075	1.0	0.0	0.0	0.0	1.0	0.0
4		-16.0	-8.833	0.0	1.0	0.0	0.0	0.0	1.0	0.0
5		0.0	0.0	5.420833	1.0	0.0	0.0	0.0	1.0	0.0
6		0.0	-8.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0

7	0.0		-11.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0	
8	0.0		0.0	5.075	1.0	0.0	0.0	0.0	1.0	0.0	
9	0.0		-8.833	5.075	1.0	0.0	0.0	0.0	1.0	0.0	
10	0.0		0.0	0.0	1.0	0.0	0.0	0.0	1.0	0.0	
11	0.0		-8.833	0.0	1.0	0.0	0.0	0.0	1.0	0.0	
12	16.0		8.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0	
13	16.0		8.833	5.075	1.0	0.0	0.0	0.0	1.0	0.0	
14	16.0		8.833	0.0	1.0	0.0	0.0	0.0	1.0	0.0	
15	21.434		11.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0	
16	21.434		8.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0	
17	21.434		0.0	5.420833	1.0	0.0	0.0	0.0	1.0	0.0	
18	21.434		-8.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0	
19	21.434		-11.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0	
20	21.434		8.833	5.075	1.0	0.0	0.0	0.0	1.0	0.0	
21	21.434		0.0	5.075	1.0	0.0	0.0	0.0	1.0	0.0	
22	21.434		-8.833	5.075	1.0	0.0	0.0	0.0	1.0	0.0	
23	21.434		8.833	0.0	1.0	0.0	0.0	0.0	1.0	0.0	
24	21.434		0.0	0.0	1.0	0.0	0.0	0.0	1.0	0.0	
25	21.434		-8.833	0.0	1.0	0.0	0.0	0.0	1.0	0.0	
26	21.434		4.167	2.5375	1.0	0.0	0.0	0.0	1.0	0.0	
27	21.434		-4.167	2.5375	1.0	0.0	0.0	0.0	1.0	0.0	
2		0	29000.0	2.860		20.0		-0.008732		0.02	0.02
2		0	29000.0	2.860		20.0		-0.008732		0.02	0.02
2		0	29000.0	5.180		20.0		-0.017526		0.02	0.02
2		0	29000.0	2.860		20.0		-0.008732		0.02	0.02
2		0	29000.0	2.860		20.0		-0.008732		0.02	0.02
2		0	29000.0	5.180		20.0		-0.017526		0.02	0.02

2			2						
2			2						
2			2						
2			2						
3	1	1	1	1.0	1.0	7.5	0.17	-0.150	
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	
2			1	29000.0	16.0	59487.0	-0.054444	1.0	16.0
3	1	1	1	1.0	1.0	7.5	0.17	-0.150	
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	
2			1	29000.0	16.0	59487.0	-0.054444	1.0	16.0
2			1	29000.0	16.0	59487.0	-0.054444	1.0	16.0
2			1	29000.0	16.0	59487.0	-0.054444	1.0	16.0
2			0	29000.0	2.860	20.0	-0.008732	0.02	0.02
2			0	29000.0	2.860	20.0	-0.008732	0.02	0.02
2			0	29000.0	5.180	20.0	-0.017526	0.02	0.02
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01
2			0	29000.0	2.590	10.0	-0.008813	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004366	0.01	0.01

```
  0              0    29000.0      2.590      10.0 -0.008313      0.01     0.01
  2              2
  2              2
  2              2
 44
 27     0    0    1
  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   26   27
 13     *    3   21.434
  4     0    0    3    0    0    0
 10     0    0    3    0    0    0
 14     0    0    3    0    0    0
      0.0       0.0    3.0E+30     0.0       0.0       0.0
      0.0       0.0    2.0E+30     0.0       0.0       0.0
      0.0       0.0    1.0E+30     0.0       0.0       0.0
0
  1
  3     1    1    3
      0.0       7.5        3.0     3.0      16.0
   8.8333       0.0    -8.8333
     60.0    0.4375       0.80     1.0       0.0
   0.5625      15.0        1.0    16.0
   0.5625      16.0        1.0    16.0
   0.5625      15.0        1.0    16.0
  2     2    2    0
    1.430      0.10       0.10   0.001    1    4
    2.590      0.10       0.10   0.001    2    5
      1.0       1.0       0.17    0.15
 29000.00       0.3       0.49
      0.0       0.0        0.0      0.0
0
```

```
     2
     2
     1           1
    10           0
    0.5625         15.0       1.00       16.0
    0.5625         15.0       1.00       15.0
       1.00        15.0       1.25       16.0
0
     4
     2
     1           1
    10           0
     0.5625        15.0        1.0       16.0
       1.00        15.0       1.25       16.0
       1.00        15.0       1.25       15.0
     3     2
0
     4
     2
     1           1
    10           0
       1.00        14.0       1.25       16.0
       1.00        15.0       1.25       15.0
     0.5625        16.0        1.0       16.0
     2     1
0

     2
     2
     1           1
```

```
  10          0
    1.00        15.0        1.25        16.0
   0.5625       15.0         1.0        16.0
   0.5625       15.0         1.0        16.0
0
   2
   2
   1          4
  10          0
   0.5625       15.0         1.0        16.0
   0.5625       15.0         1.0        15.0
   0.5625       15.0         1.0        16.0
0
   5
  44    27    1    0
   6    12
   9    12
   7    12
  10    12
   9    13
   7    13
  10    13
   8    13
  11    13
  10    11
   2     5
   3     7
   4     8
   2    18   17    1
   3    19   19    2
   4    11   13    3
```

5	14	4									
6	20										
7	21										
8	15										
9	22	20	6								
10	23	21	7								
11	15	15	8								
9	22										
10	23										
11	15										
18	25	24	17								
19	25	18									
18	20										
14	15										
19	21										
25	25										
20	25										
22	27	25	20								
22	27										
26	23										
21	27										
23	27										
21	15										
15	24										
16	23										
20	23										
21	22										
22	25										
1		0.0		11.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0

2	0.0	8.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0
3	0.0	0.0	5.420833	1.0	0.0	0.0	0.0	1.0	0.0
4	0.0	-8.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0
5	0.0	-11.833	5.420833	1.0	0.0	0.0	0.0	1.0	0.0
6	0.0	9.833	5.075	1.0	0.0	0.0	0.0	1.0	0.0
7	0.0	0.0	5.075	1.0	0.0	0.0	0.0	1.0	0.0
8	0.0	-8.833	5.075	1.0	0.0	0.0	0.0	1.0	0.0
9	0.0	8.833	0.0	1.0	0.0	0.0	0.0	1.0	0.0
10	0.0	0.0	0.0	1.0	0.0	0.0	0.0	1.0	0.0
11	0.0	-8.833	0.0	1.0	0.0	0.0	0.0	1.0	0.0
12	0.0	4.41667	2.5375	1.0	0.0	0.0	0.0	1.0	0.0
13	0.0	-4.41667	2.5375	1.0	0.0	0.0	0.0	1.0	0.0
14	5.434	-8.83333	5.42083	1.0	0.0	0.0	0.0	1.0	0.0
15	5.434	-8.83333	5.075	1.0	0.0	0.0	0.0	1.0	0.0
16	5.434	-8.83333	0.0	1.0	0.0	0.0	0.0	1.0	0.0
17	21.434	11.83333	5.42083	1.0	0.0	0.0	0.0	1.0	0.0
18	21.434	8.83333	5.42083	1.0	0.0	0.0	0.0	1.0	0.0
19	21.434	0.0	5.42083	1.0	0.0	0.0	0.0	1.0	0.0
20	21.434	8.83333	5.075	1.0	0.0	0.0	0.0	1.0	0.0
21	21.434	0.0	5.075	1.0	0.0	0.0	0.0	1.0	0.0
22	21.434	8.83333	0.0	1.0	0.0	0.0	0.0	1.0	0.0
23	21.434	0.0	0.0	1.0	0.0	0.0	0.0	1.0	0.0
24	42.859	11.83333	5.42083	1.0	0.0	0.0	0.0	1.0	0.0
25	37.434	8.83333	5.42083	1.0	0.0	0.0	0.0	1.0	0.0
26	37.434	8.83333	5.075	1.0	0.0	0.0	0.0	1.0	0.0
27	37.434	8.83333	0.0	1.0	0.0	0.0	0.0	1.0	0.0

2	0	29000.0	1.430	10.0	-0.004866	0.01	0.01
2	0	29000.0	1.430	10.0	-0.004866	0.01	0.01
2	0	29000.0	1.430	10.0	-0.004866	0.01	0.01
2	0	29000.0	1.430	10.0	-0.004866	0.01	0.01

2			0	29000.0	2.590	10.0	-0.008813	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004866	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004866	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004866	0.01	0.01
2			0	29000.0	1.430	10.0	-0.004866	0.01	0.01
2			0	29000.0	2.590	10.0	-0.008813	0.01	0.01
2			2						
2			2						
2			2						
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
3	1	1	1	1.0	1.0	7.5	0.17	-0.150	
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	
2			1	29000.0	16.0	59487.0	-0.054444	1.0	16.0
2			1	29000.0	16.0	59487.0	-0.054444	1.0	16.0
2			1	29000.0	16.0	59487.0	-0.054444	1.0	16.0
4	4	1	1	1.0	1.0	7.5	0.17	-0.150	
3	1	1	1	1.0	1.0	7.5	0.17	-0.150	
2			2						
2			2						
2			2						
2			2						
2			1	29000.0	9.0	10587.0	-0.03063	0.5625	16.0
4	4	1	0	29000.0	29000.0	0.4375	0.30	-0.490	

```
  2          1  29000.0     16.0   59487.0 -0.054444    1.0    16.0
  2          0  29000.0    2.860      20.0 -0.008732   0.02    0.02
  2          0  29000.0    2.860      20.0 -0.008732   0.02    0.02
  2          0  29000.0    5.180      20.0 -0.017526   0.02    0.02
  2          0  29000.0    2.860      20.0 -0.008732   0.02    0.02
  2          0  29000.0    2.860      20.0 -0.008732   0.02    0.02
  2          0  29000.0    5.180      20.0 -0.017526   0.02    0.02
  2          0  29000.0    2.860      20.0 -0.008732   0.02    0.02
  2          0  29000.0    2.860      20.0 -0.008732   0.02    0.02
  2          0  29000.0    5.180      20.0 -0.017526   0.02    0.02
 44
 27    0    0    1
  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   26   27
 13    3    3   21.434
 16    0    0    3    0    0    0
 23    0    0    3    0    0    0
 27    0    0    3    0    0    0
     0.0       0.0   3.0E+30     0.0        0.0        0.0
     0.0       0.0   2.0E+30     0.0        0.0        0.0
     0.0       0.0   1.0E+30     0.0        0.0        0.0
0
```

```
EXAMPLE 5D, 2 SPAN, 4 GIRDERS, DEAD LOAD 1, NON-COMPOSITE
3 SLIGHTLY SKEWED SUPPORTS, -20, -5 AND 20 DEGREES. POSITIVE CLOCKWISE.
     7     0     0     1     0     1     0     0     1
     2     7     7     6
     3
     1     1     2     4
     500.00        7.5        3.0        3.0       20.0
      13.25   4.41667  -4.41667    -13.25
      60.0    0.6875        0.8      1.000        2.0
    0.6875      16.0     1.5625       16.0
    0.625       16.0     1.3125       16.0
    0.5625      16.0     1.0000       16.0
    0.5000      16.0     0.625        16.0
     3     3     2     0
       1.43       0.10       0.10      0.001     1                   4
       2.59       0.10       0.10      0.001     2     3     5     6
        1.0        1.0       0.17       0.15
    29000.0        0.3       0.49
        0.0        0.0        0.0        0.0
     1
        0.0      -20.0
       10.0      -10.0
       20.0        0.0
0
     2
     2
     4        16.0
     1           4
2
```

```
     2
     2
     1          1
    10          0
     1.375         16.0     1.6875       16.0
    1.1875         16.0     1.4375       16.0
    1.0000         16.0     1.25         16.0
    0.6875         16.0     0.875        16.0
0
     3
     2
     1          1
     4       20.0
     2
       0.0          0.0
      10.0         -5.0
      20.0          0.0
0
     2
     2
     1          1
     4       16.0
0
     2
     3
     1          4
     3     1000.0
    10          0
    0.6875         16.0     1.5625       16.0
    0.625          16.0     1.3125       16.0
    0.5625         16.0     1.0000       16.0
```

```
   0.5000        16.0   0.625       16.0
0
     3
     2
     1          1
     4      20.0
     3
       0.0        0.0
       7.5        7.5
      15.0       15.0
0
```

```
EXAMPLE 5T, 2 SPAN, 4 GIRDERS, LIVE LOAD 2 TRUCKS, COMPOSITE
3 SLIGHTLY SKEWED SUPPORTS, -20, -5 AND 20 DEGREES. POSITIVE CLOCKWISE.
    7    0    0    2    0    1    0    0    0
    2    7    7    6
    3
    1    1    2    4
    500.00       7.5        3.0        3.0       20.0
     13.25   4.41667   -4.41667     -13.25
     60.0    0.6875         0.8      1.000       2.0
   0.6875       16.0     1.5625       16.0
   0.625        16.0     1.3125       16.0
   0.5625       16.0     1.0000       16.0
   0.5000       16.0     0.625        16.0
    3    3    2    0
      1.43      0.10       0.10      0.001     1                   4
      2.59      0.10       0.10      0.001     2     3     5       6
    3625.0    3625.0       0.17       0.15
   29000.0       0.3       0.49
       0.0       0.0        0.0        1.0
    2
HS20-44A 0       1.0      110.0       -5.0      500.0 FORW
    6
       4.0      3.00      14.00
       4.0     -3.00      14.00
      16.0      3.00        0.0
      16.0     -3.00        0.0
      16.0      3.00     -14.00
      16.0     -3.00     -14.00
HS20-44B 1       1.0      110.0        5.0      500.0 BACK
```

```
     1
        0.0       -20.0
       10.0       -10.0
       20.0         0.0
0
     2
     2
     4        16.0
     1           4
0
     2
     3
     1           1
    11         1.0
    10           0
     1.375         16.0     1.6875       16.0
    1.1875         16.0     1.4375       16.0
    1.0000         16.0     1.25         16.0
    0.6875         16.0     0.875        16.0
0
     3
     2
     1           1
     4        20.0
     2
       0.0          0.0
      10.0         -5.0
      20.0          0.0
0
     2
     2
```

```
       1         1
       4      16.0
0
       2
       4
       1         4
       3    1000.0
      11    3625.0
      10         0
      0.6875          16.0     1.5625        16.0
      0.625           16.0     1.3125        16.0
      0.5625          16.0     1.0000        16.0
      0.5000          16.0     0.625         16.0
0
       3
       2
       1         1
       4      20.0
       3
         0.0           0.0
         7.5           7.5
        15.0          15.0
0
```

```
EXAMPLE 5, 2 SPAN, 4 GIRDERS, LIVE LOAD 2 LANE LOADS, COMPOSITE
3 SLIGHTLY SKEWED SUPPORTS, -20, -5 AND 20 DEGREES. POSITIVE CLOCKWISE.
    7    0    2    2    0    1    0    0    0
    2    7    7    6
    3
    1    1    2    4
    500.00        7.5        3.0        3.0        20.0
     13.25    4.41667   -4.41667     -13.25
     60.0     0.8875        0.8      1.000        2.0
   0.6875        16.0     1.5625       16.0
   0.625         16.0     1.3125       16.0
   0.5625        16.0     1.0000       16.0
   0.5000        16.0     0.625        16.0
    3    3    2    0
      1.43       0.10       0.10      0.001    1              4
      2.59       0.10       0.10      0.001    2    3    5    6
    3625.0     3625.0       0.17       0.15
   29000.0        0.3       0.49
       0.0        0.0        0.0        2.0
    2
HS20-44A          1.0       55.0       -5.0       500.0
    1     0.064
      18.0        0.0      -8.00
       5.0      -55.0       -5.0       55.0
HS20 44A          1.0       55.0        5.0       500.0
    1     0.064
      18.0        0.0      -8.00
       5.0      -50.0       -5.0       52.0
```

```
    1
      0.0       -20.0
     10.0       -10.0
     20.0         0.0
0
    2
    2
    4       16.0
    1          4
0
    2
    3
    1          1
   11        1.0
   10          0
    1.375        16.0    1.6875       16.0
   1.1875        16.0    1.4375       16.0
   1.0000        16.0    1.25         16.0
   0.6875        16.0    0.875        16.0
0
    3
    2
    1          1
    4       20.0
    2
     0.0          0.0
    10.0         -5.0
    20.0          0.0
0
    2
    2
    1          1
```

```
     4        16.0
0
     2
     4
     1             4
     3      1000.0
    11      3625.0
    10             0
    0.6875          16.0     1.5625        16.0
    0.625           16.0     1.3125        16.0
    0.5625          16.0     1.0000        16.0
    0.5000          16.0     0.625         16.0
0
     3
     2
     1             1
     4        20.0
     3
       0.0           0.0
       7.5           7.5
      15.0          15.0
0
```

```
EXAMPLE 6D, 2 SPAN, 4 GIRDERS, DEAD LOAD 1, NON-COMPOSITE
3 SLIGHTLY SKEWED SUPPORTS, -20, -5 AND 20 DEGREES. POSITIVE CLOCKWISE.
    7    0    0    1    0    1    0    0    1
    2    7    7    6
    3
    1    1    2    4
   -500.00       7.5       3.0       3.0       20.0
     13.25   4.41667  -4.41667    -13.25
     60.0    0.6875        0.8     1.000       2.0
   0.6875       16.0    1.5625      16.0
   0.625        16.0    1.3125      16.0
   0.5625       16.0    1.0000      16.0
   0.5000       16.0    0.625       16.0
    3    3    2    0
      1.43      0.10      0.10     0.001    1              4
      2.59      0.10      0.10     0.001    2    3    5    6
       1.0       1.0      0.17      0.15
   29000.0       0.3      0.49
       0.0       0.0       0.0       0.0
    1
       0.0     -20.0
      10.0     -10.0
      20.0       0.0
0
    2
    2
    4       16.0
    1          4
0
```

```
    2
    2
    1          1
   10          0
    1.375          16.0      1.6875        16.0
   1.1875          16.0      1.4375        16.0
   1.0000          16.0      1.25          16.0
   0.6875          16.0      0.875         16.0
0
    3
    2
    1          1
    4       20.0
    2
      0.0          0.0
     10.0         -5.0
     20.0          0.0
0
    2
    2
    1          1
    4       16.0
0
    2
    3
    1          4
    3     1000.0
   10          0
   0.6875          16.0      1.5625        16.0
   0.625           16.0      1.3125        16.0
   0.5625          16.0      1.0000        16.0
```

```
   0.5000        16.0    0.625        16.0
0
    3
    2
    1          1
    4       20.0
    3
      0.0        0.0
      7.5        7.5
     15.0       15.0
0
```

```
EXAMPLE 6T, 2 SPAN, 4 GIRDERS, LIVE LOAD 2 TRUCKS, COMPOSITE
3 SLIGHTLY SKEWED SUPPORTS, -20, -5 AND 20 DEGREES. POSITIVE CLOCKWISE.
    7    0    0    2    0    1    0    0    0
    2    7    7    6
    3
    1    1    2    4
  -500.00      7.5       3.0       3.0       20.0
    13.25  4.41667  -4.41667    -13.25
    60.0   0.6875        0.8     1.000       2.0
  0.6875      16.0    1.5625      16.0
  0.625       16.0    1.3125      16.0
  0.5625      16.0    1.0000      16.0
  0.5000      16.0    0.625       16.0
    3    3    2    0
     1.43     0.10      0.10     0.001    1              4
     2.59     0.10      0.10     0.001    2    3    5    6
   3625.0   3625.0      0.17      0.15
  29000.0     0.3       0.49
      0.0     0.0        0.0       1.0
    2
HS20-44A 0     1.0     110.0      -5.0    -500.0 FORW
    6
      4.0     3.00     14.00
      4.0    -3.00     14.00
     16.0     3.00       0.0
     16.0    -3.00       0.0
     16.0     3.00    -14.00
     16.0    -3.00    -14.00
HS20-44B 1     1.0     110.0       5.0    -500.0 BACK
```

```
    1
      0.0       -20.0
     10.0       -10.0
     20.0         0.0
0
    2
    2
    4       16.0
    1          4
0
    2
    3
    1          1
   11        1.0
   10          0
    1.375        16.0    1.6875      16.0
   1.1875        16.0    1.4375      16.0
   1.0000        16.0    1.25        16.0
   0.6875        16.0    0.875       16.0
0
    3
    2
    1          1
    4       20.0
    2
     0.0          0.0
    10.0         -5.0
    20.0          0.0
0
    2
    2
```

```
    1          1
    4       16.0
0
    2
    4
    1          4
    3     1000.0
   11     3625.0
   10          0
   0.6875        16.0     1.5625        16.0
   0.625         16.0     1.3125        16.0
   0.5625        16.0     1.0090        16.0
   0.5000        16.0     0.625         16.0
0
    3
    2
    1          1
    4       20.0
    3
      0.0          0.0
      7.5          7.5
     15.0         15.0
0
```

```
EXAMPLE 6LL 2 SPAN, 4 GIRDERS, LIVE LOAD 2 LANE LOADS, COMPOSITE
3 SLIGHTLY SKEWED SUPPORTS, -20, -5 AND 20 DEGREES. POSITIVE CLOCKWISE.
    7     0     2     2     0     1     0     0     0
    2     7     7     6
    3
    1     1     2     4
   -500.00      7.5       3.0       3.0      20.0
     13.25   4.41667  -4.41667    -13.25
     60.0    0.8875       0.8     1.000       2.0
   0.6875      16.0    1.5625      16.0
   0.625       16.0    1.3125      16.0
   0.5625      16.0    1.0000      16.0
   0.5000      16.0    0.625       16.0
    3     3     2     0
      1.43      0.10      0.10     0.001     1                   4
      2.59      0.10      0.10     0.001     2     3     5     6
    3625.0    3625.0      0.17      0.15
   29000.0      0.3       0.49
       0.0      0.0        0.0       2.0
    2
HS20-44A         1.0      55.0      -5.0    -500.0
    1      0.064
      18.0       0.0     -8.00
       5.0     -55.0      -5.0      55.0
HS20 44A         1.0      55.0       5.0    -500.0
    1      0.064
      18.0       0.0     -8.00
       5.0    -58.0       -5.0      52.0
```

```
      1
        0.0      -20.0
       10.0      -10.0
       20.0        0.0
  0
      2
      2
      4        16.0
      1           4
  0
      2
      3
      1           1
     11         1.0
     10           0
      1.375        16.0   1.6875       16.0
     1.1875        16.0   1.4375       16.0
     1.0000        16.0   1.25         16.0
     0.6875        16.0   0.875        16.0
  0
      3
      2
      1           1
      4        20.0
      2
        0.0        0.0
       10.0       -5.0
       20.0        0.0
  0
      2
      2
      1           1
```

```
    4       16.0
0
    2
    4
    1          4
    3    1000.0
   11    3625.0
   10          0
   0.6875        16.0     1.5625        16.0
   0.625         16.0     1.3125        16.0
   0.5625        16.0     1.0000        16.0
   0.5000        16.0     0.625         16.0
0
    3
    2
    1          1
    4       20.0
    3
      0.0         0.0
      7.5         7.5
     15.0        15.0
0
```

APPENDIX C
GRAPHICAL DISPLAY OF INPUT DOCUMENTATION

GUIDE FOR INPUT DATA

SECTION 2.4.1 INITIAL DATA FOR PROGRAM

A) Control Cards

Identification of run (two lines)

1

1 80

2

1 80

Control card

3

Nogp	Ipl	Idef	Jrec	Irec	Ircsum	Nospg	Kgen	Komp
1	6	11	16	21	26	31	36	41 45

B) Support conditions

Nsubbc(1), Nsubbc(2)

Nspan	Nfix			(3)	(4)	(5)	(6)	(7)	(8)	(9)	(10)
1 (I)	6 (I)	11 (I)	16 (I)	21 (I)	26 (I)	31 (I)	36 (I)	41 (I)	46 (I)	51 (I)	56 (I) 60

C) Prescribed support conditions

Jspg	Kspg	Vspg	
1 (I)	6 (I)	11 (F)	20

Nospg times

D) Erection Procedure

Kgrst	Kgren	Ksbst	Ksben	
1 (F)	11 (F)	21 (F)	31 (F)	40

Irec times

SECTION 2.4.2 IROUTE = 1

A) Substructure index

Iroute	= 1
1 (I) 5	

B) Bridge Geometry

1

Nv	Idivt	Idivl	Ngir	
1 (I)	6 (I)	11 (I)	16 (I)	20

2

Rref	Tc	Ovin	Ovout	Sdia
1 (F)	11 (F)	21 (F)	31 (F)	41 (F) 50

C) Girder information

1

Rgir(1)	Rgir(2)	Rgir(3)	Rgir(4)	Rgir(5)	Rgir(6)	Rgir(7)	Rgir(8)
1 (F)	11 (F)	21 (F)	31 (F)	41 (F)	51 (F)	61 (F)	71 (F) 80

Ngir times

2

Girdwd	Tw	Tftavg	Tfbavg	Soff
1 (F)	11 (F)	21 (F)	31 (F)	41 (F) 50

3

Tft	Bft	Tfb	Bfb
1 (F)	11 (F)	21 (F)	31 (F) 40

Ngir times

D) Diaphragm information

Ibrace(1), Ibrace(2)

		Ndifbr	Ibrbot
1 (I)	6 (I)	11 (I)	16 (I) 20

2

Adia(1-6)	Diy(1-6)	Diz(1-6)	Torj(1-6)	Ibr(1)	Ibr(2)	Ibr(3)	Ibr(4)	Ibr(5)	Ibr(6)
1 (F)	11 (F)	21 (F)	31 (F)	41 (I)	46 (I)	51 (I)	56 (I)	61 (I)	66 (I) 70

Ndifbr times

E) Bottom horizontal bracing information

1

Adia(7)	Diy(7)	Diz(7)	Torj(7)
1 (F)	11 (F)	21 (F)	31 (F) 40

2 Ngsbot, Ngebot

1 (I)	6 (I) 10

Ibrbot times

F) Material properties

1

Ec(1)	Ec(2)	Poisc	Densc
1 (F)	11 (F)	21 (F)	31 (F) 40

2

Est	Poisst	Densst
1 (F)	11 (F)	21 (F) 30

G) Slab load information

Curbld(1)	Curbld(2)	Surfld	Zlivld
1 (F)	11 (F)	21 (F)	31 (F) 40

H) Truck load information (Zlivld = 1.0)

1

Ntruck
1 (I) 5

2

Name	Ibid	Dlf	Xpos	Ypos	Radtr		Idir
1 (A)	9(I)	11 (F)	21 (F)	31 (F)	40	50	52 (A) 55

3

Nwheel
1 (I) 5

4

Trld	Centtr	Centlg
1 (F)	11 (F)	21 (F) 30

Nwheel times

Ntruck times

I) Lane load information (Zlivld = 2.0)

1

Nlanld
1 (I) 5

2

Name		Dlf	Xpos	Ypos	Radln
1 (A) 8		11 (F)	21 (F)	31 (F) 40	50

3

Nconld	Wdist
1 (I) 6	(F) 15

4

Trld	Centtr	Centlg
1 (F)	11 (F)	21 (F) 30

Nconld times

Wy(1)	Wx(1)	Wy(2)	Wx(2)
1 (F)	11 (F)	21 (F)	31 (F) 40

Nlanld times

J) Additional loads

Idof	Jsubld	Kelno		P(1)	P(2)	P(3)	P(4)
1 (I)	6 (I)	11 (I) 15		21 (F)	31 (F)	41 (F)	51 (F) 60

SECTION 2.4.3 IROUTE = 2

A) Substructure index

1

Iroute	= 2
1 (I) 5	

B) Number of values to be changed

Noval
1 (I) 5

C) Update of values

Nalt	Value
1 (I)	6 (F) 15

→ Noval times

D) Additional loads

Idof	Jsubld	Kelno		P(1)	P(2)	P(3)	P(4)
1 (I)	6 (I)	11 (I) 15		21 (F)	31 (F)	41 (F)	51 (F)

60

SECTION 2.4.4 IROUTE = 3

A) Substructure index

1	
Iroute	= 3
1 (I) 5	

B) Data for Either sections 2.4.2 or 2.4.3

C) Slight skew support placement

Nsbc
1 (I) 5

D) Slight skew mesh geometry

Dsks	Dske
1 (F)	11 (F) 20

(Idivl + 1) times

E) Additional loads

Idof	Jsubld	Kelno		P(1)	P(2)	P(3)	P(4)
1 (I)	6 (I)	11 (I) 15		21 (F)	31 (F)	41 (F)	51 (F) 60

SECTION 2.4.5 IROUTE = 4

A) Substructure index

1	
Iroute	= 4
1 (I) 5	

B) Data for Either sections 2.4.2 or 2.4.3

C) Severe skew support placement

Ngs	Nge	
1 (I)	6 (I)	10

E) Additional loads

Idof	Jsubld	Kelno		P(1)	P(2)	P(3)	P(4)	
1 (I)	6 (I)	11 (I)	15	21 (F)	31 (F)	41 (F)	51 (F)	60

SECTION 2.4.6 IROUTE = 5

A) Substructure index

Iroute	= 5
1 (I) 5	

B) Control cards for this substructure

Noel	Nopt	Nv	R0
1 (I)	6 (I)	11 (I)	16 (F) 25

C) Element node numbers for this substructure

I	J	K	L
1 (I)	6 (I)	11 (I)	16 (I) 20

Noel times

D) Cylindrical coordinates (R0 > 0, R0 < 0)

No		D	A	Z
1 (I) 5		11 (F)	21 (F)	31 (F) 40

E) Cartesian coordinates

No		Xq	Yq	Zq	E1(1)	E1(2)	E1(3)	E2(1)	E2(2)	E2(3)

1 (I) 5 11 (F) 21 (F) 31 (F) 41 (F) 46 (F) 51 (F) 56 (F) 61 (F) 66 (F) 70

Nopt times

F) Element types and material properties
Use the same order of elements as in section 2.4.6, part C.

Web elements

= 0

Nodes	Ntri	Quadt	Kbeam	E1	E2	Thq	V21	Gm

1 (I) 6 (I) 11 (I) 16 (I) 21 (F) 31 (F) 41 (F) 51 (F) 61 (F) 70

Slab elements

= 1

Nodes	Ntri	Quadt	Kbeam	E1	E2	Thq	V21	Gm

1 (I) 6 (I) 11 (I) 16 (I) 21 (F) 31 (F) 41 (F) 51 (F) 61 (F) 70

Beam elements

= 2 = 0

Nodes		Kbeam	Em	A	Gj	Gm	Iy	Iz

1 (I) 5 16 (I) 21 (F) 31 (F) 41 (F) 51 (F) 61 (F) 70 (F) 80

Flange elements

= 2 = 1

Nodes		Kbeam	Em	A	Gj	Gm	Tf	Bf

1 (I) 5 16 (I) 21 (F) 31 (F) 41 (F) 51 (F) 61 (F) 70 (F) 80

Rigid link elements

= 2		= 2
Nodes		Kbcam
1 (I) 5		16 (I) 20

G) Number of elements per each substructure

Noel	-Masnd	"	"	"	"	"	"	"	"	"	"	"	"	"	"
1 (I)	6 (I)	11 (I)	16 (I)	21 (I)	26 (I)	31 (I)	36 (I)	41 (I)	46 (I)	51(I)	56 (I)	61 (I)	66 (I)	71 (I)	76 (I) 80

→ Nv times

H) Control cards for this substructure

			= 1
Mas(1)	Mas(2)	Lpgi5	Mas(3)
1 (I)	6 (I)	11 (I)	16 (I) 20

I) Substructure master nodes (local nodes which will be renumbered to master nodes)

Mas(4)	Mas(5)	"	"	"	"	"	"	"	"	"	"	"	"	"	"
1 (I)	6 (I)	11 (I)	16 (I)	21 (I)	26 (I)	31 (I)	36 (I)	41 (I)	46 (I)	51(I)	56 (I)	61 (I)	66 (I)	71 (I)	76 (I) 80

→ Mas(1) times

J) Nodes having boundary conditions excluding master nodes

Node	Bc-x	Bc-y	Bc-z	Bc-xx	Bc-yy	BC-zz
1 (I)	6 (I)	11 (I)	16 (I)	21 (I)	26 (I)	31 (I) 35

→ Mas(2) times

K) Specified boundary conditions for non-master nodes

X-trans	Y-trans	Z-trans	X-rot	Y-rot	Z-rot
1 (F)	11 (F)	21 (F)	31 (F)	41 (F)	51 (F) 60

Lpgi5 times

L) Input required for data generator program

Nxnd	Nbc5	Lpgo5	Dcent5
1 (I)	6 (I)	11 (I)	16 (F) 25

M) Boundary condition codes for master nodes of this substructure

Node	Bc-x	Bc-y	Bc-z	Bc-xx	Bc-yy	Bc-zz
1 (I)	6 (I)	11 (I)	16 (I)	21 (I)	26 (I)	31 (I) 35

Nbc5 times

N) Specified boundary conditions for master nodes

X-trans	Y-trans	Z-trans	X-rot	Y-rot	Z-rot
1 (F)	11 (F)	21 (F)	31 (F)	41 (F)	51 (F) 60

Lpgo5 times

O) Additional loads

Idof	Jsubld	Kelno		P(1)	P(2)	P(3)	P(4)
1 (I)	6 (I)	11 (I)	15	21 (F)	31 (F)	41 (F)	51 (F) 60

BIBLIOGRAPHY

1. "Guide Specifications for Horizontally Curved Highway Bridges," The American Association of State Highway and Transportation Officials, Washington D.C., 1980.

2. Heins, C. P., *Bending and Torsional Design in Structural Members*, D. C. Heath and Company, 1975.

3. Heins, C. P., "Steel Curved I and Box Girder Design and Analysis," *Analysis and Design of Bridges*, Martinus Nijhoff Publishers, The Hague, 1984.

4. "V-Load Analysis: An Approximate Procedure, Simplified and Extended, for Determining Moments and Shears in Designing Horizontally-Curved Open-Framed Highway Bridges," USS Highway Structures Design Handbook, Chapter 12, Vol. 1.

5. Timoshenko, S. P., *Theory of Plates and Shells*, McGraw-Hill, New York, 1959.

6. Brockenbrough, R. L., "Distribution Factors for Curved I-Girder Bridges," *Journal of Structural Engineering*, Vol.112, No. 10, October 1986.

7. Heins, C. P., "Box Girder Design," AISC Engineering Journal, Vol. 15, No. 4, Dec 1978.

8. Wilson, L., Bathe, K.J., Peterson, F.E., Dovey, H., "User's Guide to Program SAP," University of Calif. Dept of Civil Engineering, Nov. 1972.

9. Schelling, D.R., Heins, C.P., Sikes,G.H., "State of the Art of Curved Girder Bridge Programs," *Computer and Structures*, Vol. 9, pp. 27-37, Pergamon Press, London, 1978.

10. Heins, C.P., Sahin, A., "Natural Frequency of Curved Box Girder Bridges," *ASCE St. Div. Journal.*, Vol. 105, No. ST5, May 1979.

11. Heins, C.P., Sheu, F.H., "Computer Analysis of Steel Box Girder Bridges," *Civil Engineering Dept. Report Univ of Maryland*, June 1981.

12. Heins, C. P., Olenick,J.C., "Curved Box Beam Bridges Analysis, " *Computer and Structures Journal*, Vol. 6, pp. 65-73, Pergamon Press, London, 1976.

13. Weaver, W., Johnson,P.R., *Finite Elements for Structural Analysis*, Prentice-Hall, Englewood Cliffs, New Jersey, 1984.

14. Fiechtl, A., Fenves, G. L., Frank, K. H., "Approximate Analysis of Horizontally Curved Girder Bridges," Research Report No. 360-2F, Center for Transportation Research, The University of Texas at Austin, November 1987.

15. Davis, B.G., Hoffman, T.R., *Fortran : A Structured, Disciplined Style*,.McGraw-Hill, New York, 1983.

16. Yargicoglu, A., Johnson, C.P., "Temperature Induced Stresses in Highway Bridges by Finite Element Analysis and Field Tests," Research Report No. 23-3F, Center for Highway Research, The University of Texas at Austin, July 1978.

17. Johnson, C.P., Thepchatri, T., Will, K., "Static and Buckling Analysis of Highway Bridges by Finite Element Procedures," Research Report No. 155-1F, Center for Highway Research, The University of Texas at Austin, 1973.