1. Report No. FHWA/TX-03/1898-3	2. Government Accession No.	3. Recipient's Catalog No.
4. Title and Subtitle **UTrAp: Finite-Element-Based Software for the Analysis of Curved Trapezoidal Girders under Construction Loads**		5. Report Date October 2002
		6. Performing Organization Code
7. Author(s) C. Topkaya, J. A. Yura, E. B. Williamson, and K. H. Frank		8. Performing Organization Report No. Research Report 1898-3
9. Performing Organization Name and Address Center for Transportation Research The University of Texas at Austin 3208 Red River, Suite 200 Austin, TX 78705-2650		10. Work Unit No. (TRAIS)
		11. Contract or Grant No. Research Project 0-1898
12. Sponsoring Agency Name and Address Texas Department of Transportation Research and Technology Implementation Office P.O. Box 5080 Austin, TX 78763-5080		13. Type of Report and Period Covered Research Report
		14. Sponsoring Agency Code

15. Supplementary Notes

Project conducted in cooperation with the U.S. Department of Transportation, Federal Highway Administration, and the Texas Department of Transportation.

16. Abstract

Due to advances in fabrication technology, the use of steel trapezoidal box girders for curved interchange structures has become popular. The rapid erection, long span capability, economics, and aesthetics of these girders make them more favorable than other structural systems. The response of composite box girders with live loading, as well as the behavior of quasi-closed box girders during construction, must be considered during the design process. Considering both cases, the design for construction loading is the least understood and is the most important. Stresses due to construction loading can reach up to 60-70 percent of the total design stress for a given cross section.

In this report, we document the development and usage of the software package UTrAp. This program allows for the analysis of curved girder bridge systems in a more rigorous manner than traditional grid analysis approaches. UTrAp utilizes a finite element methodology to determine stresses and strains throughout the cross-section of a bridge girder. In addition, internal and external brace members are modeled explicitly so that no assumptions are required (e.g., equivalent plate method) for determining member forces. Unlike currently available software, UTrAp has the capability of modeling the effects of semi-cured concrete. The developed software has been designed to be computationally efficient and easy to use for bridge designers. A graphical user interface (GUI) has been developed along with the finite element analysis routines to aid in entering data and visualizing analysis results. A comparison to measured field data shows that UTrAp provides an accurate characterization of girder response under construction loads.

17. Key Words finite element analysis, curved girder bridge, trapezoidal girder, construction loads, software	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 report) Unclassified	20. Security Classif. (of this page) Unclassified	21. No. of pages 44	22. Price

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

UTrAp: Finite-Element-Based Software for the Analysis of Curved Trapezoidal Girders under Construction Loads

by

C. Topkaya, E. B. Williamson, and J. A. Yura

Research Report 1898-3

Research Project 0-1898

SIMPLIFIED DETAILS FOR TRAPEZOIDAL STEEL BOX BEAMS

conducted for the

Texas Department of 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

October 2002

ACKNOWLEDGEMENTS

DISCLAIMER

The contents of this report reflect the views of the authors, who are responsible for the facts and the accuracy of the data presented herein. The contents do not necessarily reflect the view of the Federal Highway Administration or the Texas Department of Transportation. This report does not constitute a standard, specification, or regulation.

NOT INTENDED FOR CONSTRUCTION,

PERMIT, OR BIDDING PURPOSES

J. A. Yura, P.E, Texas No. 29859

K. H. Frank, P.E, Texas No. 48953

E. B. Williamson

Research Supervisors

TABLE OF CONTENTS

LIST OF FIGURES

LIST OF TABLES

x

SUMMARY

Due to advances in fabrication technology, the use of steel trapezoidal box girders for curved interchange structures has become popular. The rapid erection, long span capability, economics, and aesthetics of these girders make them more favorable than other structural systems. The response of composite box girders with live loading, as well as the behavior of quasi-closed box girders during construction, must be considered during the design process. Considering both cases, the design for construction loading is the least understood and is the most important. Stresses due to construction loading can reach up to 60-70 percent of the total design stress for a given cross section.

In this report, we document the development and usage of the software package UTrAp. This program allows for the analysis of curved girder bridge systems in a more rigorous manner than traditional grid analysis approaches. UTrAp utilizes a finite element methodology to determine stresses and strains throughout the cross section of a bridge girder. In addition, internal and external brace members are modeled explicitly so that no assumptions are required (e.g., equivalent plate method) for determining member forces. Unlike currently available software, UTrAp has the capability of modeling the effects of semi-cured concrete. The developed software has been designed to be computationally efficient and easy to use for bridge designers. A graphical user interface (GUI) has been developed along with the finite element analysis routines to aid in entering data and visualizing analysis results. A comparison to measured field data shows that UTrAp provides an accurate characterization of girder response under construction loads.

PART 1: DEVELOPMENT OF COMPUTATIONAL SOFTWARE FOR POUR SEQUENCE ANALYSIS

A computer program (UTrAp) with a graphical user interface (GUI) was developed for pour sequence analysis. The package consists of an analysis module, which was written in FORTRAN, and a GUI, which was written in Visual Basic. The program was developed for use on personal computers. The following sections provide documentation of the program in detail.

1.1   ANALYSIS MODULE

The analysis module consists of a three-dimensional finite element program with pre- and post-processing capabilities. Input for the analysis module is provided by a text file that is created through use of the GUI. The module itself is capable of generating a finite element mesh, element connectivity data and material properties based on the geometrical properties supplied through the GUI. The program also generates nodal loading based on the values given in the input file. After the pre-processing is completed, the program assembles the global stiffness matrix and solves the equilibrium equations to determine the displacements corresponding to a given analysis case. As a last step, the module post-processes the displacements to compute cross-sectional forces, stresses and brace member forces. The following sections document the formulation of the analysis module and discuss current capabilities and limitations.

1.2   PROGRAM CAPABILITIES

The analysis module is capable of analyzing curved, trapezoidal, steel box girders under construction loads. Currently, this program can analyze only single and dual girder systems with a constant radius of curvature. The number of girders is limited to two because systems with more than two girders are not common in practice, and the solution of such bridges using the finite element method will require computer resources that surpass the capabilities of current personal computers. At the time the computational software for this research project was being developed, it was assumed that a typical PC used by an engineer has 256 MB memory and 1 GHz processor speed. Although not very widely used, personal computers with up to 2 GB memory and 1.7 GHz processor speed were also available. A typical twin girder system with the mesh adopted by this program requires about 700 MB of physical memory for in-core solution.

The analysis module allows the thickness of the plates to vary while the centerline distance of all components (e.g., web, top flange) is held constant. Internal, external and top lateral braces can be specified in the program, and supports can be placed at any location along the bridge length. There is no internal constraint on the number of braces, length of the bridge or number of supports. The program is for linear analysis and is capable of handling multiple analysis cases.

1.3   INPUT REQUIREMENTS

Geometric Properties: The number of girders, radius of curvature, length of the bridge and cross-sectional dimensions are required input. Cross-sectional dimensions include depth of the web, width of the bottom flange, top flange width, width of the concrete deck and thickness of the concrete deck. In addition, the program requires the thickness of the web, bottom flange and top flange along the bridge length. There is no restriction on the number of different plate properties that a user can specify.

Supports: Support locations must be specified by the user. Locations are defined by the distance relative to a coordinate along the arc length. Supports are assumed to be either pinned or a roller. Actual properties of the support bearings are currently not considered in the program.

Braces: Locations, types and areas of internal, external and top lateral braces must be specified.

1

Pour sequence: There can be several analyses performed that are independent of each other. The concrete deck can be divided into segments. The length of each segment must be provided as input. For each analysis, properties of the concrete deck can be varied. There are three properties associated with a deck segment. These properties are the concrete modulus, stud stiffness associated with particular segment and the distributed load on the segment.

1.4 ALGORITHM OF THE ANALYSIS MODULE

Program UTrAp uses 9-noded shell elements, truss and stud elements to construct a finite element mesh. Steel plates and the concrete deck are modeled with shell elements. Braces and studs are modeled with truss and stud elements, respectively. The following paragraphs present the details of the program algorithm.

1.4.1 Node Locations and Element Connectivity

The program automatically forms the node locations and element connectivity based upon the geometric properties of the bridge specified by the user through the GUI. A constant mesh density is used for all bridges. The webs and bottom flanges of the girders are modeled with 4 shell elements while 2 elements are used for the top flanges. The concrete deck is modeled with 10 and 20 shell elements for single and dual girder systems, respectively. Previous work on curved trapezoidal girders (Fan, 1999) revealed that this mesh density is adequate for most typical cross-sectional dimensions. Along the length of the bridge, each element is two-feet long. This mesh density assures elements of aspect ratios less than two for most practical cases. According to the geometrical dimensions and radius of curvature, the program forms the locations of the nodes. For each node, three mutually orthogonal unit vectors (V_1, V_2, V_3) are formed. These unit vectors are used in defining the shell element geometry. Figure 1.1 shows the nodes and unit vectors on a single girder system. Unit vectors are formed in such a way that V_3 points in the direction through the thickness of a shell element and V_2 is tangent to the arc along the bridge length. V_1 is formed such that it is orthogonal to both V_2 and V_3.
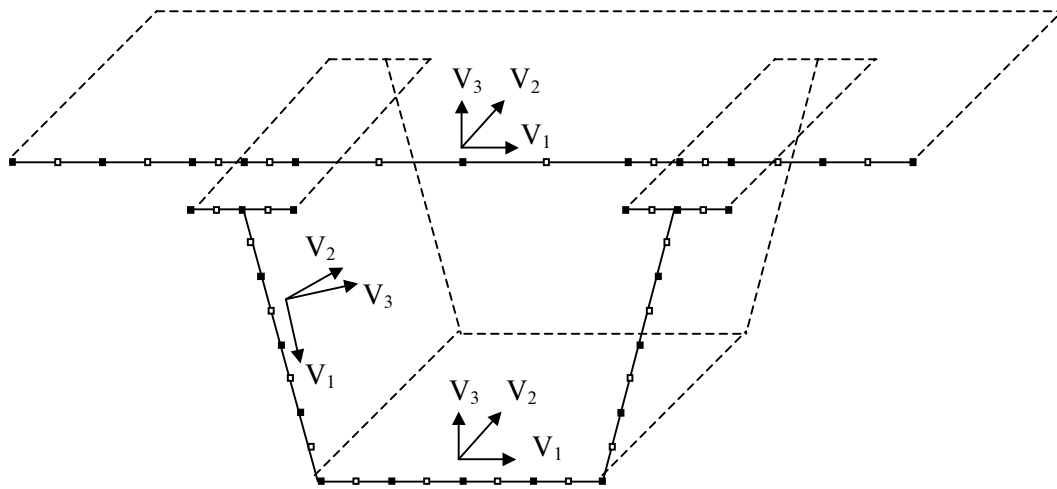
Figure 1.1: Node Locations and Unit Vectors

1.4.2 Modeling of the Physical System

There are several modeling techniques presented in the literature for analyzing the steel girder–concrete deck interaction. One proposed method is to model the steel section with shell elements and the concrete deck with brick elements (Tarhini and Frederick, 1992) as shown in Figure 1.2. Spring elements are placed at the interface to model the shear studs. This type of modeling produces a very large number of

2

degrees of freedom, because, in order to capture the deck response with sufficient accuracy, a large number of brick elements must be used.

In another technique, both the steel cross section and the concrete deck are modeled with shell elements (Brockenbrough, 1986, and Tabsh and Sahajwani, 1997). The steel and concrete sections are connected together with connector (beam) elements (Figure 1.2). The length of the connector elements has to be chosen by the analyst to account for the offset between the mid-surface of shell elements that are used to model the top flange of the steel girder and the mid-surface of the shell elements that represent the concrete deck. This approach is the most favored technique presented in the literature despite the fact that there is no consensus on how to choose the connector length.

In the software developed for the current research, another approach is used to model the cross section which addresses both problems mentioned above. Two types of shell elements were used in modeling a given cross section. In the formulation of a shell element, the three-dimensional domain is represented by a surface. The selection of the representative surface depends on the particular formulation being used. Any surface along the thickness could be considered as a reference surface. Thus, the two types of shell elements used in the program are similar in formulation but differ in the reference surface definitions. For steel sections, the reference surface is considered to be the middle surface while for the concrete deck, the bottom surface is used as a reference (Figure 1.3). Steel sections and the concrete deck are connected together by spring elements that represent the stud connectors. This modeling technique reduces the degrees of freedom when compared with the brick model. In addition, it properly models the interface behavior by eliminating the connector elements and including the girder offset by using the bottom shell surface as the reference surface.
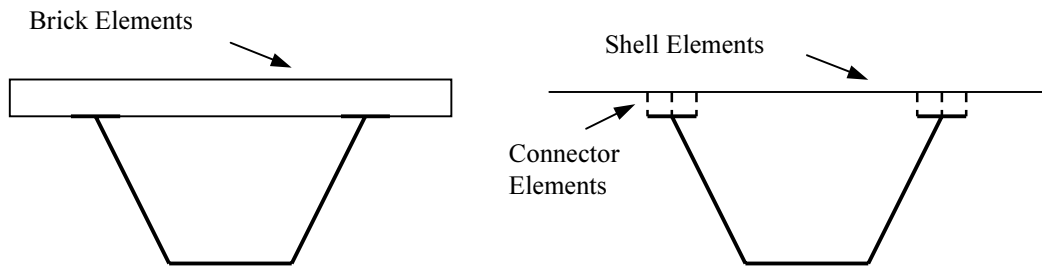

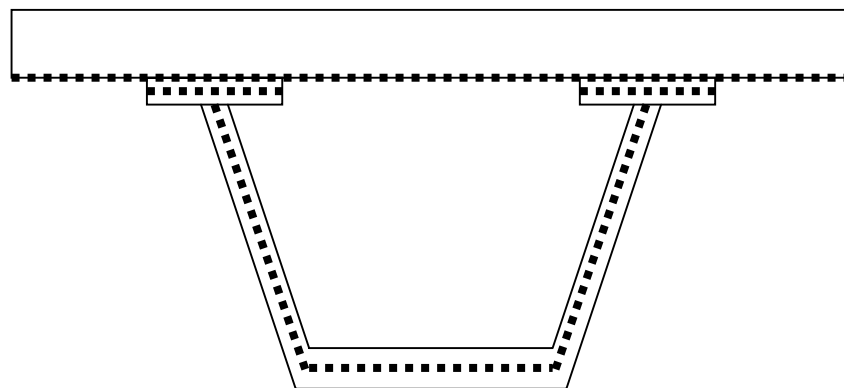**Figure 1.2: Different Modeling Techniques for Deck-Flange Interface**

Figure 1.3: Reference Surfaces for Shell Elements

Internal, external and top lateral braces are modeled with truss elements. The program calculates the nodal connectivity of the brace elements from the supplied location values. Currently, only one type of internal brace and one type of external brace is handled in the program (Figure 1.4). The ones included are the typical types used in practice.
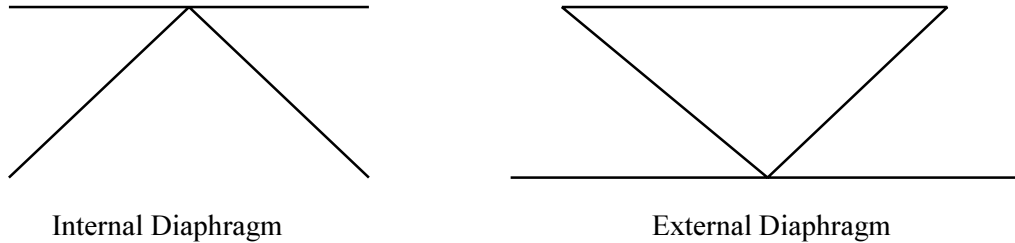

Internal Diaphragm   External Diaphragm

Figure 1.4: Internal and External Diaphragms Used in the Program

Spring elements are used to model the shear studs. For each top flange, three nodes are connected to the concrete deck. The connection between the concrete deck and girder top flanges is achieved by spring elements in all three global directions. Springs are placed every foot along the bridge length even if studs are not physically present. The stiffness properties of each spring element are modified by the analysis module according to the physical distribution of studs in a particular region. Thus, for each spring element there is a corresponding stiffness modification factor. The modification factor is calculated by dividing the stud spacing value by 12 inches. If there are less than three studs per flange, very low modification factors are assigned to the studs which are not present. In the case where the number of studs per flange is greater than three, the modification factor is multiplied by the number of studs per flange and divided by three.

In practice, diaphragms in the form of thick plates are placed at the support locations to reduce stresses caused by high torsional forces. Diaphragms form a solid cross section with very high torsional and distortional stiffness. In order to represent the behavior of the solid diaphragms without adding a large number of degrees of freedom to the finite element model, the program internally assembles a very stiff truss system (Figure 1.5) at the support locations. The stiff truss system prevents the distortion of the cross section by restraining the relative movements of the edges of the cross section.
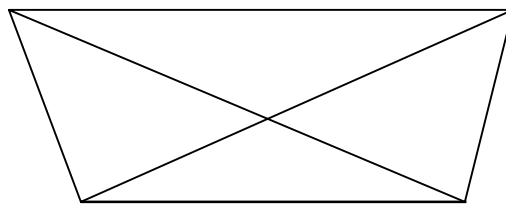
Figure 1.5: Truss System Used at Support Locations

1.4.3   *Assembly of the Global Stiffness Matrix*

Based on the element connectivity and boundary conditions, degrees of freedom are assigned to nodes throughout the structure. After the degrees of freedom have been determined, the global stiffness matrix is assembled. In order to assemble the internal and external braces, a condensation technique is used. First, the truss elements are assembled together to form a superelement. Second, the degrees of freedom which are not shared with the steel girder are condensed out so that no additional degrees of freedom are

introduced to the model. During static condensation, numerical singularities may occur due to round-off errors and because the truss system that is used to represent the internal and external braces has a very low stiffness for modes of displacement perpendicular the plane of the truss. While this mode of response is not important for the actual girder because such deformations could be resisted by bending of the braces, mathematically it can cause numerical problems if it is not addressed in a proper manner. In order to alleviate this problem, very flexible springs in all three global directions were placed at the nodes where four truss members meet. After assembling the springs and the truss elements, static condensation is performed. The following sections summarize the formulation of the element stiffness matrices used in the program.

1.4.4 Shell Element Formulation

A nine-node, isoparametric shell element (degenerated brick) originally developed by Ahmad, Irons and Zienkiewicz (1970) was implemented in the program. At each node, a unit vector V_3 extends through the thickness of the element. The unit vectors undergo rigid body motion during the deformation of the element. The element is mapped into material coordinates (ξ, η, ζ), where ξ, η are the two coordinates in the reference plane and the ζ coordinate points in the direction through the thickness of the shell. The geometry x throughout the element is interpolated as follows:
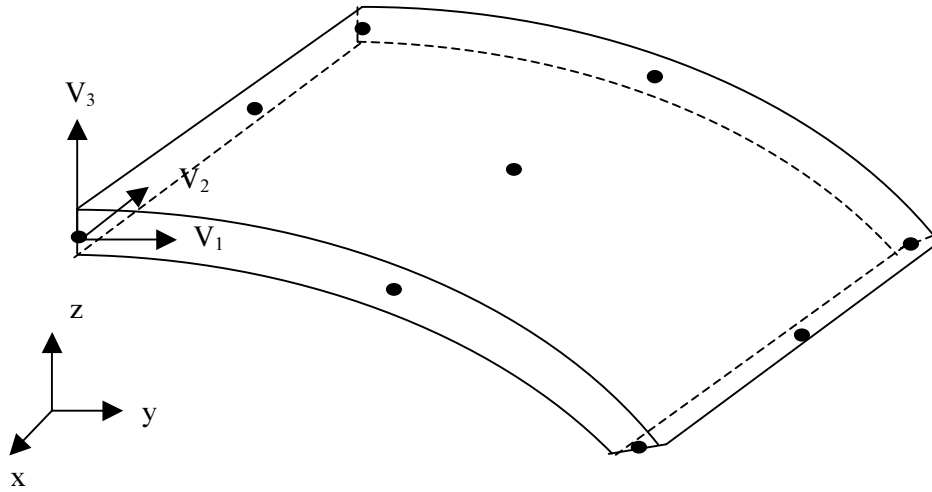
Figure 1.6: Nine-node Shell Element

$$x(\xi,\eta,\zeta) = \sum_{i=1}^{9}\left[\left(x_i + \zeta\frac{h}{2}V_3\right)N_i(\xi,\eta)\right] \tag{1}$$

where h is the thickness of the shell and $N_i(\xi,\eta)$ are the Lagrangian shape functions given explicitly in Bathe (1982) as well as many other texts on the finite element method.

The displacement field is defined by the three displacement (u,v,w) and two rotational (α,β) degrees of freedom. In order to define the rotation axes for α and β, a right-handed triplet of mutually orthogonal unit vectors (V_1, V_2, V_3) is specified as input. Rotations α and β are the rotations about the V_1 and V_2 axes, respectively. The displacement field u is interpolated as follows:

$$u(\xi,\eta,\zeta) = \sum_{i=1}^{9}\left[\left(u_i + \zeta\frac{h}{2}\left(-\alpha_i V_2 + \beta_i V_1\right)\right)N_i(\xi,\eta)\right] \tag{2}$$

where u_i is the vector of Cartesian components of the reference surface displacement at node i.

The element formulation includes the basic shell assumption that the stress normal to any lamina (ζ=constant) is zero. This assumption implies that at any point in the domain, a local rigidity matrix similar to the one used in two-dimensional plane stress analysis must be used. For analysis of an assemblage of shell elements, this local rigidity matrix has to be transformed into global coordinates. For transformation purposes, the local orthogonal coordinate axes consisting of unit vectors t_1, t_2, t_3 should be formed where t_3 is the vector normal to the shell surface at the point of consideration. The orthogonal local axes are formed according to Algorithm 1:

Algorithm 1

1. At the point of consideration:

$$t_1 = \frac{\partial}{\partial\xi}x \qquad t_2 = \frac{\partial}{\partial\eta}x$$

2. Form unit vectors:

$$t_1 = \frac{t_1}{|t_1|} \qquad t_2 = \frac{t_2}{|t_2|}$$

3. Calculate the normal vector t_3:

$$t_3 = t_1 \times t_2$$

By making use of the direction cosines of the orthonormal local axes, a transformation matrix $\underset{\sim}{R}$ is formed. The global rigidity matrix is calculated as follows:

$$D = R^T D^{local} R \tag{3}$$

The stiffness matrix is calculated as:

$$K = \int_{\Omega} B^T DB \ d\Omega \tag{4}$$

where Ω is defined as the domain of the element.

The implementation uses regular integration: 3 Gauss integration points in ξ,η directions and 2 integration points in the ζ direction.

As explained previously, the concrete deck is much thicker than the steel plates that form the cross section. Therefore, the middle surface of the concrete deck has an offset with the middle surface of the top flange plates. In order to use the same reference surface for both the top flange elements and the concrete deck elements, the bottom surface is considered as the reference surface for concrete deck elements. Therefore, the shell element formulation has been modified so that the reference surface can be taken as the bottom surface of the element. With this approach, the deck offset is properly taken into account.

1.4.5   Truss Element Formulation

A standard 3 dimensional, 2-node linear truss element is implemented into the program. The stiffness matrix formulation can be found in standard texts on matrix structural analysis (e.g., see Kassamali () or Hibbeler()).

1.4.6   Spring Element Formulation

A standard 2-node, three-dimensional spring element is implemented into the program. The stiffness matrix formulation can be found in standard texts on matrix structural analysis (e.g., see Kassamali () or Hibbeler()).
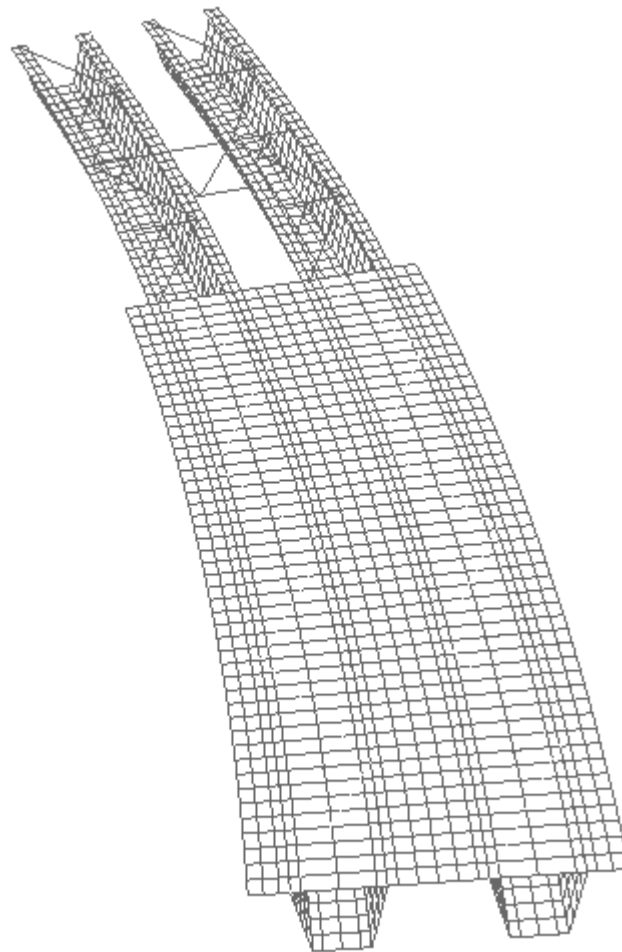

Figure 1.7: Portion of a Finite Element Model

1.4.7   Solution Capability

Large-scale, finite element analyses produce a system of linear equations which requires extensive computer resources to be solved. Until recently, most of these analyses were performed on UNIX workstations. With advances in computer technology, large-scale systems can be handled with personal computers. Because bridges are long and thin structures, the mesh adopted to represent the physical model produces a global stiffness matrix that is sparse in nature. In sparse systems, most of the entries that form the stiffness matrix are zero. A solution for the system of equations can be achieved using either direct or iterative sparse solvers. Iterative solvers were found to create numerical problems in models involving shell elements (Gullerud et al., 2001). Therefore, in this program a direct sparse solver was chosen for the solution of the system of linear equations. A sparse solver developed by Compaq, which is a part of the Compaq Extended Math Library (CXML), is adapted to the program. The solver is supplied as a library file by the Compaq Visual Fortran 6.5 compiler and can be compiled with the finite element program. Only the nonzero entries of the upper triangular half of the matrix need to be stored. In addition, two vectors which are used to define the locations of the nonzero entries are required by the solver. Based on the information of the nonzero entries and their locations, the solver is capable of reordering and factoring the stiffness matrix and solving for displacements. The solver dynamically allocates all the arrays required during the solution process. It is capable of performing operations using the virtual memory whenever available physical memory is not sufficient.

The non-zero terms in the global stiffness matrix are located in rows and columns which correspond to the degrees of freedom that are connected to each other. Before the global stiffness matrix is assembled, the two position vectors that keep track of the locations of the non-zero terms have to be formed. A subroutine was developed for this purpose. The subroutine accepts the nodal connectivity information as input. For every degree of freedom, the associated degrees of freedom are found. This information is then used to form the position vectors.

At the initial stages of the program development, several solvers were tried for adoption to the analysis module. The NASA Vector Sparse Solver, Y12maf sparse solver, a frontal solver, and the CXML sparse solver were compared. The CXML solver was found to be the most efficient in terms of memory usage and speed of solution for the PC platforms tested.

1.4.8   Post-Processing Capability

The program is capable of generating output useful to designers based on the displacements obtained from the solution process. Output from the finite element analysis is first written to text files. Next, this data is read by the GUI to output vertical deflection and cross-sectional rotation along the length of the bridge. In addition, the program calculates axial forces for all top lateral, internal and external braces. Cross-sectional stresses and forces are calculated at every two feet along the bridge length due to the fact that elements have a length of two feet. For each cross section, stresses at the center of the top surface for each element are calculated. Therefore, for each cross section, shear and normal stresses are obtained at 26 and 52 locations, for single and dual girder systems, respectively. These stress components are in the local directions (i.e., normal and perpendicular to the cross section) so that no further transformation of stresses is necessary. In addition to stresses, cross-sectional shear, moment and torsion are calculated. For each element on the cross section, the nodal internal forces and moments are computed. These forces and moments are transformed from global coordinates to local coordinates. Finally, the transformed forces and moments for all elements are summed up to determine the overall stress resultants acting on the cross section.

1.5   GRAPHICAL USER INTERFACE

The Graphical User Interface was designed to provide an environment in which the user easily enters required input data. In addition, the GUI has the capability of displaying both the numerical and

graphical output of the analysis results. Figure 1.8 shows the main form of the interface. The GUI is written in Visual Basic and has the following menus and graphics capabilities.

File Menu: This menu consists of submenus and is used for data management. A user can either start a new project (a new project description) or continue with an existing project. Any changes made to a new or existing project can be saved with the Save Project option.

Geometry Menu: This menu brings the Geometric Properties form to the screen. Information on the number of girders, radius of curvature, length of bridge, girder offset and cross-sectional dimensions should be supplied by using of this form.
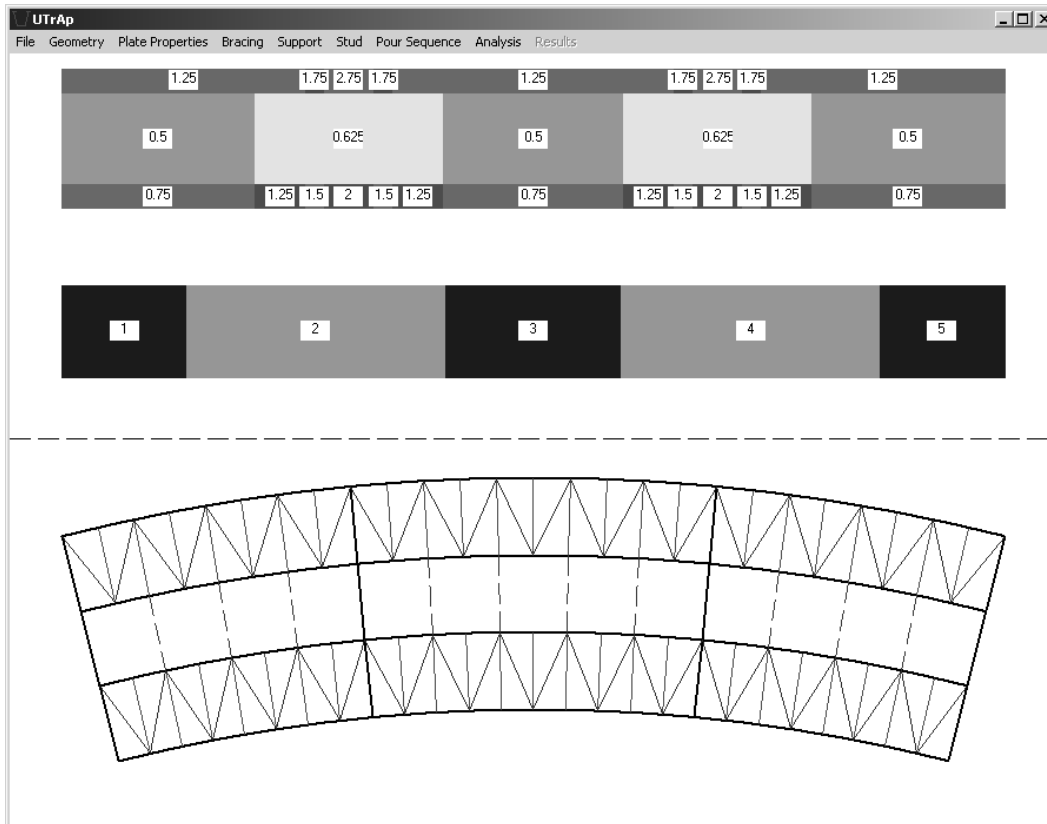
Figure 1.8: The Graphical User Interface

Plate Properties Menu: This menu brings the Plate Properties form to the screen. This form has three folders for entering web thickness, bottom flange thickness, and top flange thickness properties. In each of the folders, the length of the plate and its thickness from the start to the end of the bridge should be provided.

Bracing Menu: This menu brings the Bracing Properties form to the screen. This form has three folders for entering internal brace, external brace, and top lateral brace information. For internal and external braces, the location of the brace, its type and cross-sectional area of its members needs to be specified. For top lateral braces, start and end locations, type and cross-sectional area information needs to be supplied. Each folder has buttons to assign the same type and cross sectional area to all the brace members. In addition, buttons are provided for entering equally spaced braces between two locations.

9

Support Menu: This menu brings the Support Locations form to the screen. In this form, the locations of the supports are entered by the user. It is important to note that the location specified first on the Support Locations form will be assumed to act as a pinned support. All other specified locations will be modeled as a roller support.

Stud Menu: This menu brings the Stud Properties form to the screen. In this form, the spacing of studs and the number of studs per flange along the length of the girder need to be supplied.

Pour Sequence Menu: This menu brings the Pour Sequence form to the screen. In this form, tabulated data related to the pour sequence need to be supplied. The program is capable of performing several analysis cases corresponding to the sequence in which the concrete deck will be constructed. Thus, for each analysis case, the length of each segment needs to be entered; concrete modulus, concrete stiffness and loading information for every deck segment also needs to be supplied.

Analysis Menu: This menu executes the analysis module. Before execution, a text input file, which is read by the analysis module, is prepared based on the information supplied in the graphical user interface. In-depth error checking is performed by the GUI to ensure that the resulting input file represents a valid bridge model.

Results Menu: This menu has eight submenus. The submenus are used to visualize the analysis results. Vertical deflection and cross-sectional rotation of the bridge, brace member forces, cross-sectional stresses and forces can be tabulated or displayed graphically.

1.6   VERIFICATION OF THE COMPUTATIONAL SOFTWARE

Results from the developed software were compared with published solutions. Researchers Fan and Helwig (1999) have developed a hand-based method for predicting the top lateral brace member forces in curved box girders. The proposed method was compared against an independent finite element analysis performed using a commercially available general-purpose program. The predictions of the hand-based method were in excellent agreement with the finite element analysis. In this section, the published finite element analysis results are compared with the results obtained from UTrAp. The bridge analyzed by Fan and Helwig (1999) was a three-span, single girder system having a radius of 954.9 feet and a length of 640 feet. Details of the bridge are given in Figure 1.9. Internal braces were located at every 10 feet, and an X-type top lateral system between internal brace points was utilized. The top lateral brace members were WT 6x13 sections, and the internal brace elements were L 4x4x5/16 sections. The bridge was analyzed under a uniform load of 3.3 k/ft. Because UTrAp only allows for changes in element thickness and not width, the actual cross-sectional properties of the top flange in Section N of the bridge had to be approximated. Thus, a constant top flange width of 14 inches was assumed over the entire length of the bridge. The thickness of the top flange plates in Section N was modified to 3.8 inches to give the same plate cross-sectional area as that in the bridge modeled by Fan and Helwig (1999).

The top lateral members were grouped into two (X1 and X2) according to their orientation. Force levels for these top lateral members obtained from finite element analyses were presented by Fan and Helwig (1999). These force levels are compared with the predictions from UTrAp in Figs. 10 and 11. These graphs clearly demonstrate that the developed software is capable of producing results similar to published solutions.
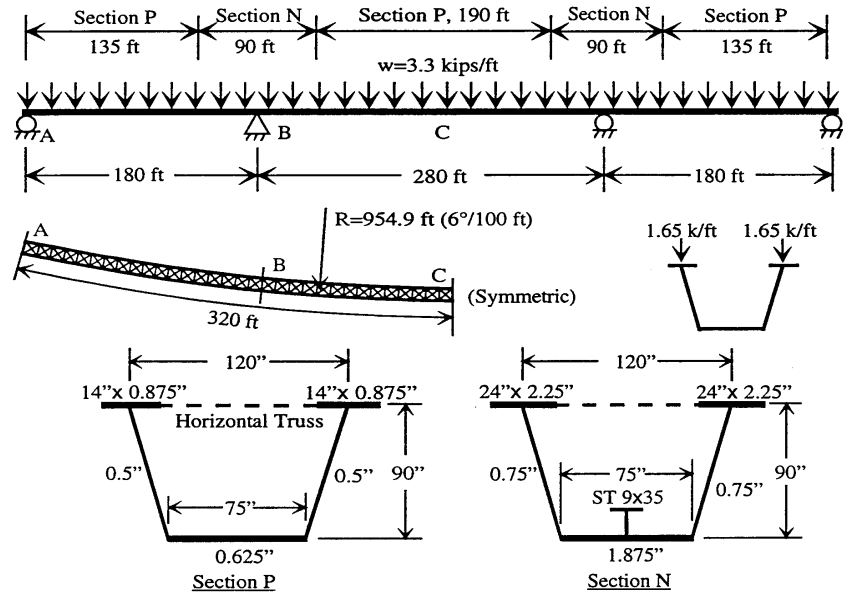
Figure 1.9: Layout and Cross-Sectional Dimensions of the Bridge (Fan (1999))
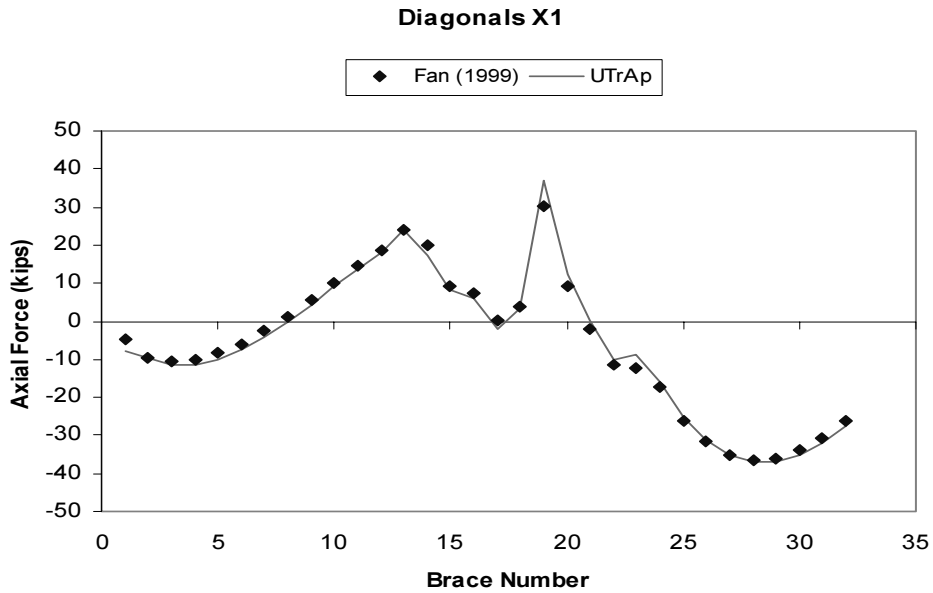

Figure 1.10: Comparison of Published and UTrAp Results for X1 Diagonals
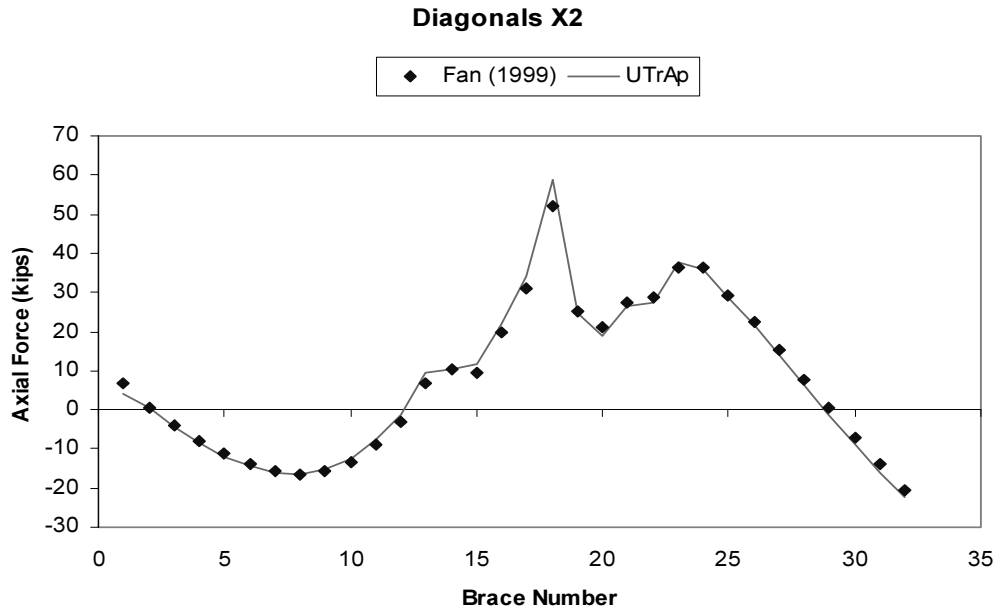
11

Diagonals X2

Figure 1.11: Comparison of Published and UTrAp Results for X2 Diagonals

PART 2: USER'S MANUAL AND EXAMPLE PROBLEM FOR UTRAP

UTrAp is a computer program developed for pour sequence analysis of curved, trapezoidal steel box girders. Currently, only single and dual girder systems with a constant radius of curvature can be analyzed with this program. The program consists of a Graphical User Interface (GUI) and an analysis module. The analysis module relies on the finite element method to compute the response of the three-dimensional bridge structure. Input data is supplied to the program by making use of the GUI. The program can handle multiple analysis cases and has graphics capability to visualize the output. In the following sections, an example problem is presented to demonstrate the steps needed to develop a model for the analysis of a curved girder bridge using UTrAp.

2.1   EXAMPLE PROBLEM DEFINITION

The example problem presented herein is a 3-span, dual girder system with a centerline radius of curvature of 450ft. The bridge is named as "Direct Connect Z" and has a centerline arc length of 493 ft. The plan view of the bridge is given in Figure 2.1.
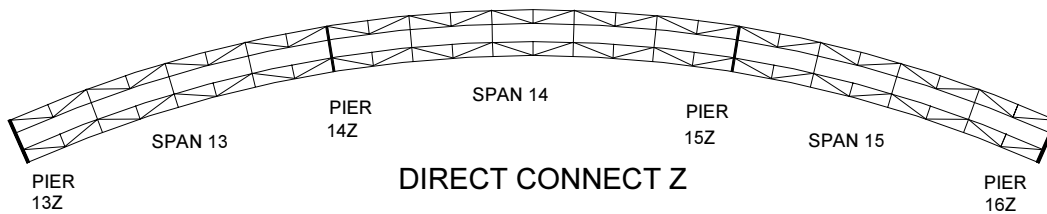
Figure 2.1: Plan View of Direct Connect Z

UTrAp accepts only positive values for the radius of curvature, and the concavity layout of the structure should be similar to the one in Figure 2.1. Therefore, the left end is considered to be the start end of the bridge. In Figure 2.1, the start end is located at PIER 13Z. Positions along the bridge are defined by the distance along the arc length relative to the start end. Cross-sectional dimensions of the Direct Connect Z are given in Figure 2.2. Web depth is measured between the centerline of the top and bottom flanges. The centerline of each girder is offset by 98 inches from the bridge centerline. The concrete deck width and thickness are 360 and 10 inches, respectively.
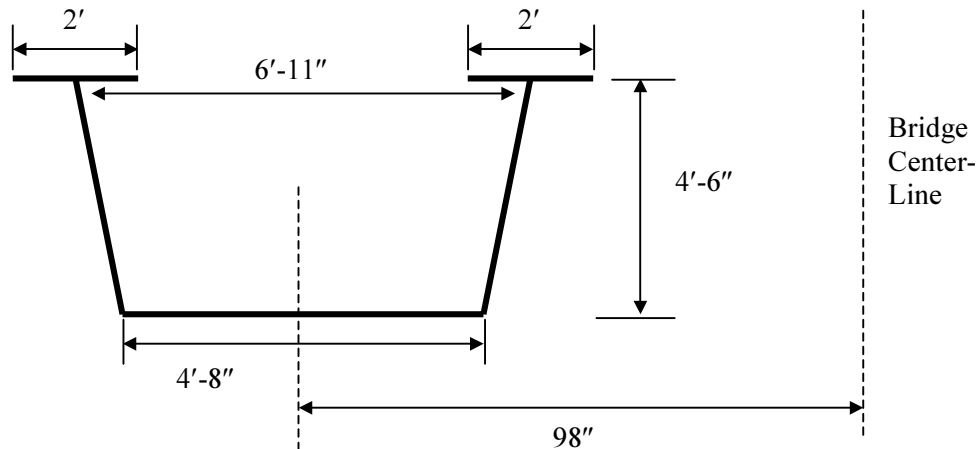
Figure 2.2: Cross-sectional Dimensions

13

The steel plates that make up the girder have variable thickness along the length of the bridge. Table 2.1 provides the details of the plate thickness for the web and flanges. Lengths given in this table are the centerline arc lengths. Properties are listed beginning from the start end of the bridge. In the current version of the program, it is assumed that both girders have the same plate thickness properties.

Table 2.1: Plate Properties

| WEB		BOTTOM FLANGE		TOP FLANGE	
Length(ft.)	Thickness(in.)	Length(ft.)	Thickness(in.)	Length(ft.)	Thickness(in.)
100.5	0.5	100.5	0.75	127	1.25
99	0.625	26.5	1.25	10	1.75
94	0.5	10	1.5	26	2.75
99	0.625	26	2.0	10	1.75
100.5	0.5	10	1.5	147	1.25
		26.5	1.25	10	1.75
		94	0.75	26	2.75
		26.5	1.25	10	1.75
		10	1.5	127	1.25
		26	2.0		
		10	1.5		
		26.5	1.25		
		100.5	0.75		
Σ = **493 ft**		Σ = **493 ft**		Σ = **493 ft**	

Bracing members are provided throughout the girder. Internal, external and top lateral braces are present. Locations of the braces are given in Table 2.2. For internal and external braces, only one location value is required. For top lateral braces, the start and end location of each brace is needed.

There are 23 internal and 26 top lateral braces per girder. In addition, there are 10 external braces between the two girders. Internal braces are in the form of K-trusses, which have members with cross-sectional area of 3.75 in^2. All top lateral braces have a cross-sectional area of 6.31 in^2, and their orientation is given in Figure 2.1. External braces are comprised of truss members with a cross-sectional area of 4.79 in^2. Details of their configuration are provided below.

The bridge has four supports which are located 0, 151.5, 341.5, and 493 feet away from the start end (Pier 13Z). Studs are spaced every 12 inches at both ends of the bridge for a distance of ten feet from the pier. For the remainder of the bridge, studs are spaced at every 24 inches. There are 3 studs per flange over the entire length of the bridge.

The concrete deck is poured in 5 segments. The lengths and the sequence of pours are given in Figure 2.3.

Table 2.2: Location of Braces

| Brace Number	Internal Bracing Location (ft)	External Bracing Location (ft)	Top Lateral Bracing Start Location (ft)	End Location (ft)
1	18.9	37.8	0	18.9
2	37.8	75.6	18.9	37.8
3	56.7	113.4	37.8	56.7
4	75.6	189.5	56.7	75.6
5	94.5	227.5	75.6	94.5
6	113.4	265.5	94.5	113.4
7	132.3	303.5	113.4	132.3
8	170.5	379.3	132.3	151.5
9	189.5	417.3	151.5	170.5
10	208.5	455.1	170.5	189.5
11	227.5		189.5	208.5
12	246.5		208.5	227.5
13	265.5		227.5	246.5
14	284.5		246.5	265.5
15	303.5		265.5	284.5
16	322.5		284.5	303.5
17	360.4		303.5	322.5
18	379.3		322.5	341.5
19	398.3		341.5	360.4
20	417.3		360.4	379.3
21	436.1		379.3	398.3
22	455.1		398.3	417.3
23	474.0		417.3	436.1
24			436.1	455.1
25			455.1	474.0
26			474.0	493.0
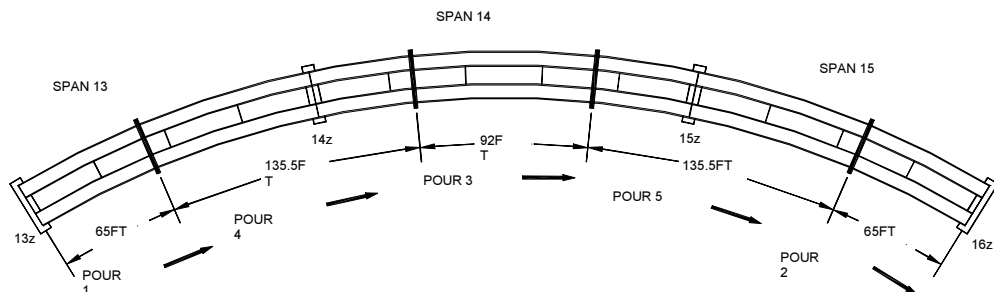
Figure 2.3: Concrete Pour Sequence

This analysis example will focus on the first three pours. The program requires the lengths of the pours and number of analyses to be performed. In this example, 3 analysis cases will be considered. In the first

analysis, the concrete deck is placed on the first segment, and a uniform loading of 3.625 k/ft is applied on that segment to account for the concrete self-weight. In the second analysis, it is assumed that the concrete on the first segment has cured and attained a stiffness of 1000 ksi with a corresponding stud stiffness of 250 k/in. A uniform loading of 3.625 k/ft is applied to the second segment due to the concrete weight. In the third analysis, it is assumed that the concrete and stud stiffness have reached stiffness values of 2000 ksi and 500 k/in, respectively, for the first segment. For the second segment, the concrete and stud stiffness values are assumed to attain values of 1000 ksi and 250 k/in, respectively. A uniform loading of 3.635 k/ft is applied to the third segment to account for the concrete weight. A summary of the analysis parameters are given in Table 2.3.

Table 2.3: Pour Sequence Analysis Parameters

Deck	Length	Analysis 1			Analysis 2			Analysis 3		
		Con. Mod.	Std. Stf.	Load	Con. Mod.	Std. Stf.	Load	Con. Mod.	Std. Stf.	Load
1	65	0	0	3.625	1000	250	0	2000	500	0
2	135.5	0	0	0	0	0	0	0	0	0
3	92	0	0	0	0	0	0	0	0	3.625
4	135.5	0	0	0	0	0	0	0	0	0
5	65	0	0	0	0	0	3.625	1000	250	0
	Σ=**493**									

2.1.1   *User's Guide and Solution of the Example Problem*

The Graphical User Interface of UTrAp has a total of 9 menus. This section describes each of these menus in detail and provides information regarding how data is supplied to UTrAp for analyzing curved, trapezoidal girders of general configuration. In addition, specific information needed to analyze the example bridge described above is provided.

File Menu: This menu has four submenus and is used for data management. Files can be stored and retrieved by making use of this menu. Details of each submenu are as follows:

New Project: This submenu starts a blank project. If a new bridge model is going to be formed, this option should be selected.

Existing Project: This submenu is used to open an existing project. The UTrAp input project files have an extension of *.inp*. When the existing project submenu is invoked, an open file box will appear which is used to select the existing project file.

Save Project: This submenu is used to save a project to the hard disk. It can be used to save the changes made to an existing project or the contents of a newly developed project. When the *Save Project* submenu is invoked, a save file box will appear which is used to name or rename the project file.

Exit: This submenu is used to exit the program.

Example Problem: A new project is formed by making use of the *New Project* submenu.

Geometry Menu: This menu is used to input the geometric properties of the bridge. Values should be typed in the boxes provided. A graphical representation of the cross section is displayed on the geometric properties form. After entering all the required data, the user must press the *Save Data* button in order for the values to be stored in memory. After the *Save Data* button is pressed, the form is closed. If the user does not want to save the values, the *Cancel* button should be pressed. This data saving process is valid for all subsequent forms.

<u>Example Problem</u>: Geometric property values are entered on the form and saved by making use of the *Save Data* button. Figure 2.4 shows the Geometric Properties form with the entered data.

Figure 2.4: Geometric Properties Form

Plate Properties Menu: This menu is used to input the plate properties for the girder. The plate properties form has three folders. Each folder is reserved for the web, the bottom flange or the top flange properties. Properties are input in a tabular form. The length of the plate and its thickness should be entered from the start to the end of the bridge. There are two buttons used to add and remove properties. Their function is explained below.

Add: This button is used to add properties. A change in plate thickness requires the user to specify a new property. The user should enter the number of properties that will be needed to characterize the bridge. After, the number of rows in the table is increased by the total number of properties specified by the user.

Remove: This button is used to remove properties. The property number that is going to be removed should be specified in the box next to the *Remove* button.

<u>Example Problem</u>: In each folder, the number of properties is entered through use of the *Add* button. Typically, a user will enter the total number of plate properties in the box next to the *Add* button before it is pressed. If additional properties are needed, however, they can be added as necessary through repeated use of the *Add* button. All plate properties are entered in a tabular format. The input for the bottom flange plate properties is given in Figure 2.5. Similar data would be provided for the web and top flanges. Once all the necessary plate properties have been specified, the user must select the *Save Data* button in order to store the information in memory.

Bracing Menu: This menu is used to input bracing information for the bridge. The brace properties form has three folders. Each folder is reserved for the internal, external, or the top lateral brace properties. Properties are input in a tabular form. Depending on the version of the program, different geometrical types of braces can be specified for internal and external braces. The location, type and

member cross-sectional area information are required for the internal and external braces. The type, start location, end location, and cross-sectional area are required for the top lateral braces. There are buttons provided to add and remove braces. Functions of the buttons are explained below.

Add: This button is used to add braces. The user should enter the number of braces that will be added to the box next to the *Add* button. The number of rows in the table is increased by the corresponding number entered by the user.

Figure 2.5: Plate Properties Form

Equally Space: This button is used to add braces at equally spaced intervals. The number of braces to be added is specified in the box next to the button. For this button to function properly, two or more location values must be entered. Braces are placed at equal intervals between these values. The location value in the first box must be smaller than the location value in the second box.

Remove: This button is used to remove braces. The brace number that is going to be removed should be specified in the box next to the *Remove* button.

Remove All Braces: This button is used to remove all the braces in a given folder that have been specified previously.

Type: This button is displayed in the internal and external braces folder. It is used to assign the same type to all braces. The brace type should be entered into the box next to this button. The available bracing types and their configurations are displayed in a separate form using *Show Internal/External Brace Types* buttons. If all braces are not of the same geometry, the type can be entered independently for each brace directly on the tabular form.

Area: This button is used to assign the same cross-sectional area value to all brace members. The cross-sectional area value should be entered into the box next to this button. If all braces do not have the same cross-sectional area, the values can be entered independently for each brace directly on the tabular form.

Show Internal/External/Top Lateral Brace Types: These buttons are used to display the types of braces that a user can specify in the program. When this button is pressed, a form that shows the geometry and

types of braces are displayed on the screen. Figure 2.6 shows the types of internal and external braces supported by the current version of the program.

All Type 1: This button is displayed only in the top lateral braces folder. It is used to assign Type 1 to all top lateral braces. Top lateral braces can have only two orientations. Therefore, there are two types of top lateral braces which are shown in Figure 2.7.

All Type 2: This button is displayed only in the top lateral braces folder. It is used to assign Type 2 to all top lateral braces.

Alternating Starting with Type 1: This button is displayed only in the top lateral braces folder. It is used to assign alternating types to consecutive braces. The first brace will be of Type 1 and the second brace will be of Type 2, etc.
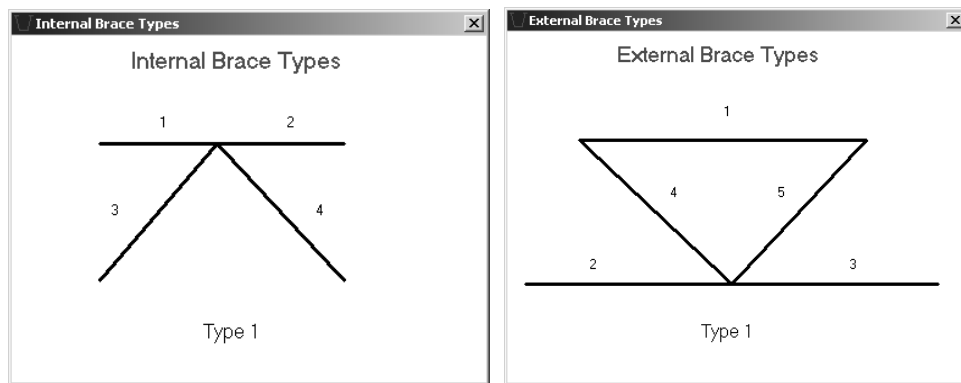
Figure 2.6: Internal and External Brace Types
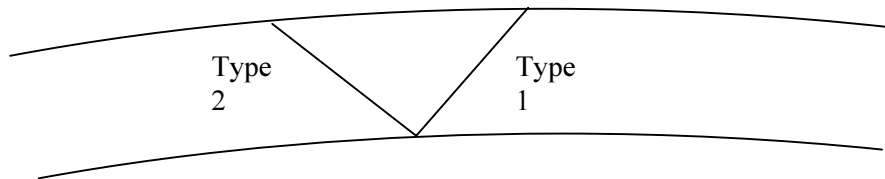
Figure 2.7: Top Lateral Brace Types

Alternating Starting with Type 2: This button is displayed only in the top lateral braces folder. It is used to assign alternating types to consecutive braces. The first brace will be of Type 2 and the second brace will be of Type 1, etc.

Although the current example does not utilize top lateral braces in an X-shaped configuration, some bridges do use such a bracing system. This type of arrangement can be handled by the program simply by specifying both Type 1 and Type 2 braces with the same start and end points. Thus, for each X-brace, the user would need to add two data entry lines with the *Add* button. Using the same starting location and ending location, one of the data entry lines would correspond to a Type 1 brace, and the other would correspond to a Type 2 brace.

Another issue of concern in defining the bracing arrangement for a given bridge relates to the definition of struts and internal braces. As shown in Figure 2.6, struts (members 1 and 2 in the figure) are included in the definition of an internal brace. If a strut acts at location in which an internal brace is not present, an alternative approach is needed for defining the strut. In order to model this situation correctly, the user can simply represent the strut by incorporating it into the definition of the top lateral bracing system.

Because a strut acts across the girder in the radial direction, it has the same start and end location relative to the end of the bridge. Thus, by specifying either a Type 1 or Type 2 strut that has the same start and end location, a user can represent the effects of a strut that acts independently from an internal brace.

Example Problem: Twenty-three internal braces, 10 external braces and 26 top lateral braces are added to the folders by making use of the *Add* button. Brace locations, types and cross-sectional areas are entered into the folders according to the information given in Table 2.2. All internal and external braces are Type 1. Top lateral braces have alternating types starting with Type 2. Figures 8 and 9 show the two folders of the bracing properties form.

Support Menu: This menu is used to input support locations. Locations are input in a tabular form. The program assumes that only one of the supports is pinned and the rest are rollers. The first support specified is considered to be the pinned one. The number of rows of the tabular input form is controlled by the *Add* and *Remove* buttons. Functions of the buttons are explained below.

Add: This button is used to add supports. The user should enter the number of supports that will be added to the box next to the *Add* button. The number of rows in the table is increased by that specific amount.

Remove: This button is used to remove supports. The support number that is going to be removed should be specified in the box next to the *Remove* button.

Example Problem: Four supports are added to the table by making use of the *Add* button. Support locations given in the description of the bridge (Figure 2.1) are entered in the table. Figure 2.10 shows the support locations form along with the entered data.

Figure 2.8: Bracing Properties Form - Internal Braces Folder

Bracing Properties

	Type	Location1 ft.	Location2 ft.	Area (sq.in.)
1	2	0	18.9	6.31
2	1	18.9	37.8	6.31
3	2	37.8	56.7	6.31
4	1	56.7	75.6	6.31
5	2	75.6	94.5	6.31
6	1	94.5	113.4	6.31
7	2	113.4	132.3	6.31
8	1	132.3	151.5	6.31
9	2	151.5	170.5	6.31
10	1	170.5	189.5	6.31
11	2	189.5	208.5	6.31
12	1	208.5	227.5	6.31
13	2	227.5	246.5	6.31
14	1	246.5	265.5	6.31
15	2	265.5	284.5	6.31
16	1	284.5	303.5	6.31
17	2	303.5	322.5	6.31
18	1	322.5	341.5	6.31
19	2	341.5	360.4	6.31
20	1	360.4	379.3	6.31

Tabs: Internal Braces | External Braces | **Top Lateral Braces**

Add [ ] top lateral braces

Equally Space [ ] braces between [ ] ft. and [ ] ft.

Remove top brace number [ ]

Remove All Braces

Identical Brace Properties

All Type 1

All Type 2

Alternating Starting with Type 1

Alternating Starting with Type 2

Area [ ]

Show Top Lateral Brace Types

Save Data | Cancel

Figure 2.9: Bracing Properties Form – Top Lateral Braces Folder

Support Locations

Support No	Location (ft)
1	0
2	151.5
3	341.5
4	493

Add [ ] supports

Remove support [ ]

Save Data

Cancel

Figure 2.10: Support Locations Form

Stud Menu: This menu is used to input stud properties. Properties are input in tabular form. The spacing of the studs along the length of the bridge and the number of studs per flange should be supplied to the program. The number of rows of the tabular input form is controlled by the *Add* and *Remove* buttons. Functions of these buttons are explained below.

Add: This button is used to add properties. The user should enter the number of properties that will be added to the box next to the *Add* button. The number of rows in the table is increased by that specific amount.

Remove: This button is used to remove properties. The property number that is going to be removed should be specified in the box next to the *Remove* button.

Example Problem: For this problem, stud properties change three times along the bridge length. Therefore, three rows are added to the table by making use of the *Add* button. Cells of the table are filled according to the geometry information given in the bridge description. Figure 2.11 shows the stud properties form along with the entered data.

Figure 2.11: Stud Properties Form

Pour Sequence Menu: This menu is used to input pour sequence analysis parameters. Parameters are input in tabular form. The concrete deck can be divided into segments corresponding to each pour, and there can be multiple analyses that are independent from each other. For each analysis, properties of the deck segments and loading on the segments should be provided as input. Properties for a deck segment include the stiffness of concrete and the stiffness of the studs. Lengths of the deck segments are the same for all analyses and their values should be given as input. The tabular form is controlled by four buttons. These buttons are used to add and remove columns and rows from the table. Functions of the buttons are explained below.

Add Analysis Case: This button is used to add a new analysis case to the table. Three columns for analysis parameters are added to the right of the table each time a new analysis is added. The three new columns are used to specify the concrete stiffness, the stud stiffness, and any loading acting on the deck.

Remove Analysis Case: This button is used to remove a specific analysis case. The analysis number that is going to be removed should be entered into the box next to this button. Three columns related with the analysis number specified are removed from the table.

Add Deck Property After: As mentioned before, the concrete deck can be divided into segments. At least one deck property must be specified. This button is used to add a new deck property row to the table. The new deck property is added after the deck number specified in the box next to this button. If no deck

has been defined in the table previously, a value of zero should be used. Specifying a value of zero adds blank cells to the first row.

Remove Deck Property: This button is used to remove a deck property row. The number of the deck property to be removed should be entered into the box next to this button. The specified row is deleted from the table.

Example Problem: In this problem, the concrete deck is divided into five segments. These deck segments are added to the table by making use of the *Add Deck Property After* button. There are a total of three analyses to be performed. These analysis cases are added to the table by using the *Add Analysis Case* button. The table is filled with parameters specified in Table 2.3. Figure 2.12 shows the pour sequence form together with the input data.

Figure 2.12: Pour Sequence Form

Analysis Menu: This menu is used to perform the finite element analysis using the data entered on each of the forms. As the user inputs data using the forms, a graphical representation of the overall bridge properties is displayed in the main form of the program. This form contains three figures. On the very top figure, the plate thickness along the length is shown in elevation view. The middle figure shows the deck numbers and their relative lengths. The bottom figure is a plan view of the bridge showing all the supports and braces. Figure 2.13 shows the main form after all the data are provided.

When the user invokes the Analysis menu, the program verifies that the properties used to define the bridge are consistently defined. For example, the length of all plates and decks should add up to the bridge length, and brace and support locations should be admissible. If any of the entries are missing or violate the geometric constraints, the program will give an error message. If all entries are permissible, then the Analysis menu calls the Analysis Module to perform the finite element analysis. The analysis module runs under the DOS environment. A user can trace the progress of the analysis by observing the messages displayed in the DOS screen. Figure 2.14 shows a representative analysis screen. The DOS screen automatically disappears when the analysis is completed.

Results Menu: This menu consists of 8 submenus and is used to visualize the output. Details of the submenus will be given in the following sections along with figures obtained from the solution of the example bridge.
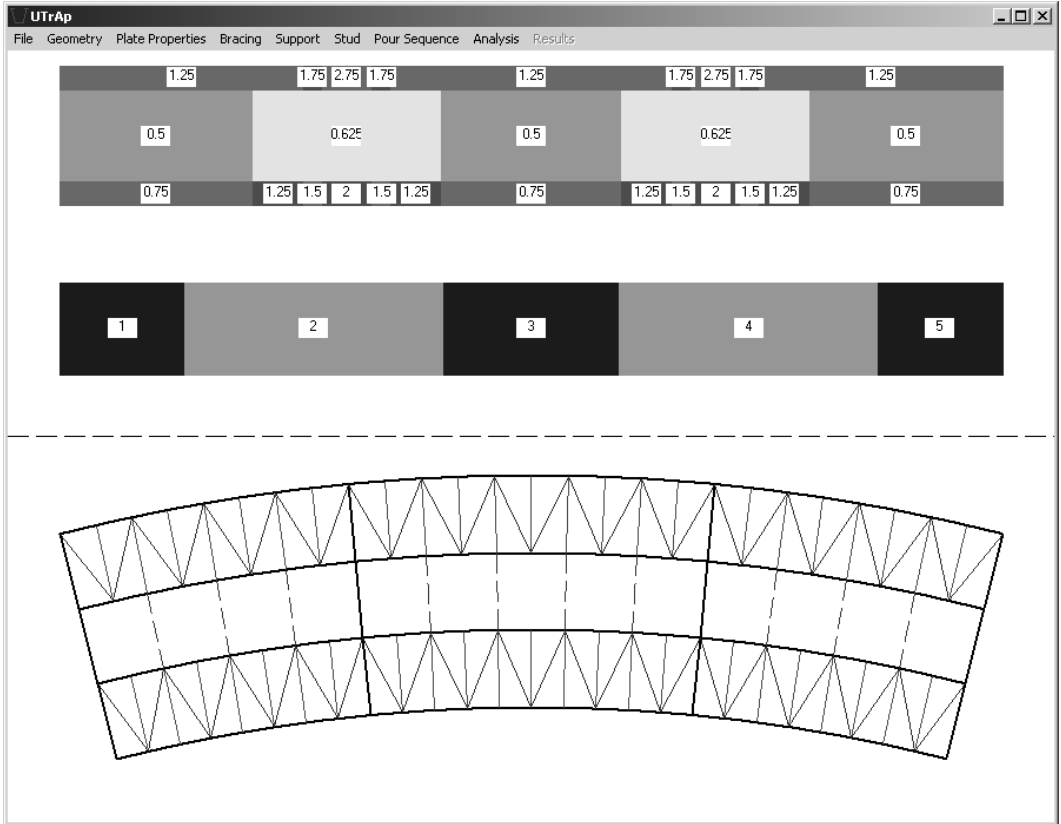
Figure 2.13: Main form of UTrAp
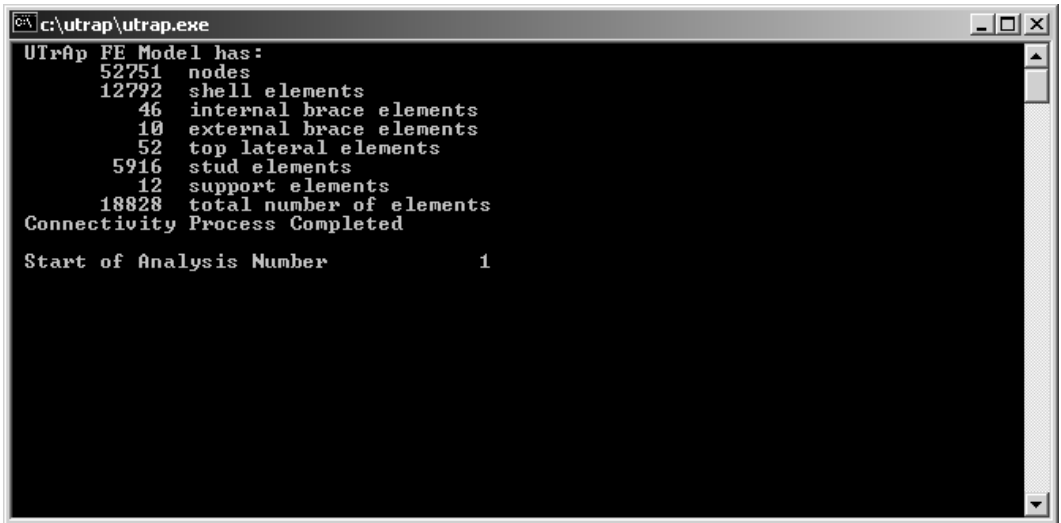
Figure 2.14: DOS Screen for an Analysis

Deflections/Cross Sectional Rotations Submenus: These submenus are used to visualize the vertical deflections and cross-sectional rotations of the bridge. Because they have identical properties, both menus will be explained together in this section. Deflection values are the vertical deflection of the center of the bottom flange. Rotation values correspond to the rotation of the bottom flange. For twin girder systems, only the deflection/rotation of the outer girder is reported. Both tabulated and graphical output

24

can be displayed. Tabulated output is in the form of deflection/rotation values at every two feet along the length of the bridge. The user can request deflection/rotation values for each analysis or the summed deflection/rotation values after each case. Deflections/Cross Sectional Rotations forms have four buttons to control the display of results. Functions of the buttons are explained below.

Tabulate Incremental Deflections/Rotations: This button is used to display the tabulated results of incremental deflection/rotation at every two feet along the bridge length. Values are presented for all analysis cases and are not summed. Figure 2.15 shows the deflections form with the results for the example bridge.

Figure 2.15: Deflections Form

Tabulate Total Deflections: This button is used to display the tabulated results of total deflection/rotation at every two feet along the bridge length. Deflection/rotation values after each analysis are presented. Total deflection values include the summation of all previous analyses. For example, values in column 3 are the summation of deflections/rotations resulting from analysis 1, 2, and 3.

Plot Incremental Deflections: This button displays the incremental deflection/rotation diagram. Incremental deflections/rotations due to all analyses are displayed on the same graph. Figure 2.16 shows a typical deflection diagram.

Plot Total Deflections: This button displays the total deflection/rotation diagram. Total deflections/rotations due to all analyses are displayed on the same graph in a manner similar to that shown in Figure 2.16 with the exception that each curve represents the summation of the deflections/rotations from all previous analysis cases.

Cross-Sectional Forces Submenu: This submenu is used to visualize the cross-sectional forces. Information on shear, moment and torsion are available. For twin girder systems, quantities are summed for the two girders. Both tabulated and graphical output can be displayed. Tabulated output is in the form of shear, moment and torsion values for every two feet along the length of the bridge. In addition, shear, moment and torsion diagrams can be displayed graphically. The Cross-Sectional Forces form has six buttons to control the display of results. Functions of the buttons are explained below.

Tabulate Shear: This button is used to display the tabulated results of shear every two feet along the bridge length. Incremental values are presented for all analysis cases. Figure 2.17 shows the Cross-Sectional Forces form with the results for the example bridge.

Tabulate Moment: This button is used to display the tabulated results of internal bending moment at locations every two feet along the bridge length. Incremental values are presented for all analysis cases.
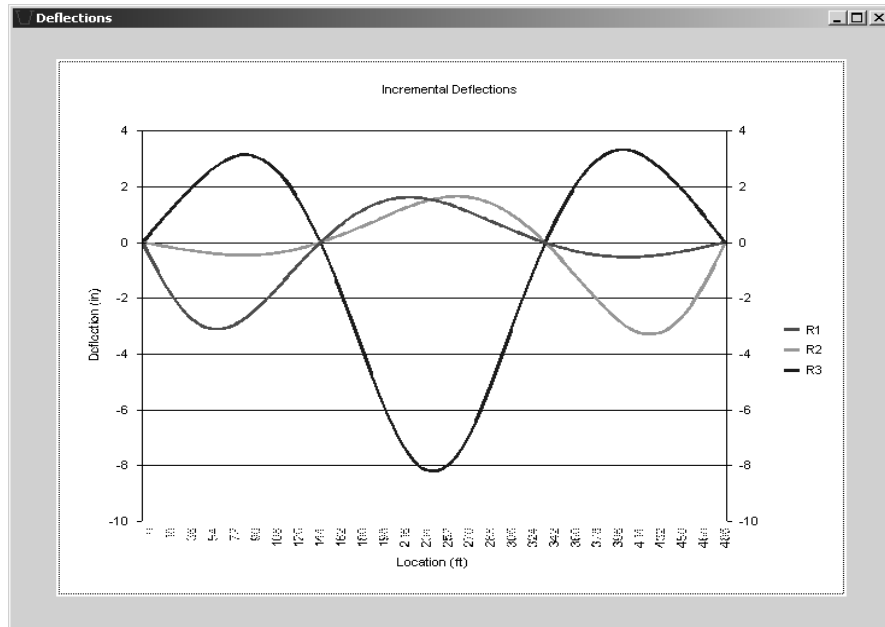
Figure 2.16: Deflection Diagram
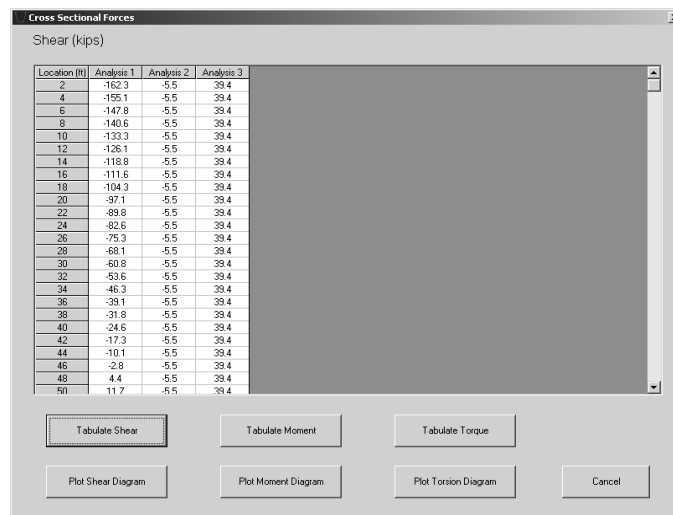
Figure 2.17: Cross Sectional Forces Form

Tabulate Torque: This button is used to display the tabulated results of torque every two feet along the bridge length. Incremental values are presented for all analysis cases.

Plot Shear Diagram: This button displays the shear diagram. Shear values for all analyses are displayed on the same graph. Figure 2.18 shows a typical shear diagram.

Plot Moment Diagram: This button displays the moment diagram. Moment values for all analyses are displayed on one graph.

Plot Torque Diagram: This button displays the torque diagram. Torque values for all analyses are displayed on the same graph.
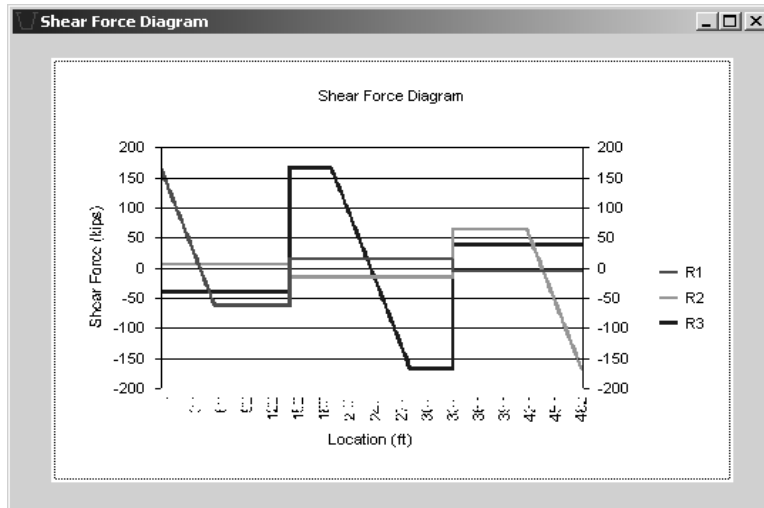
Figure 2.18: Shear Force Diagram

Stresses Submenu: This submenu is used to visualize the cross-sectional stresses. The analysis module calculates normal and shear stresses at certain locations of the cross section every two feet along the bridge length. These calculations are performed for all analysis cases. The locations on the cross section where stresses are calculated are referred to as "section points." There are 26 and 52 section points on the cross section for the single and dual girder systems, respectively. The stresses form is used to tabulate the stress values along the length of the bridge for all section points. Both shear and normal stress can be tabulated in incremental or total format. In incremental format, results of all analyses are independent of each other. In total format, results after an analysis include the summation of all previous analyses. Radio buttons are placed on the form to select between shear and normal stress as well as between incremental and normal values. This form is also used to display the stress diagram. Variation of normal or shear stress along the bridge length can be plotted for a specified section point. Furthermore, this form can be used to display stresses at all section points at a certain cross section along the bridge length. The stresses form has three buttons that interact with three scroll-down boxes. Functions of the buttons are explained below.

Tabulate Stresses: This button is used to tabulate the stress values along the bridge length for all section points. Normal or shear stress can be tabulated depending on the user's selection. An analysis case must be selected using the scroll-down boxes. In addition, total or incremental values can be displayed. Figure 2.19 shows a tabulated stress output in the stresses menu.

Plot Stress Diagram: This button is used to display the variation of normal or shear stress along the bridge length for a certain section point.   An analysis case and section point must be selected using the scroll-down boxes. Figure 2.20 shows a plot of normal stress along the bridge length for analysis number 1 and section point 52.

Visualize Cross Sectional Stresses: This button is used to display the stresses at all section points for a certain cross section. An analysis case and a location must be selected using the scroll-down boxes. Figure 2.21 shows the normal stress distribution due to analysis 1 in a cross section that is 101 feet away from the start end. Section points and stress values are given on the cross section diagram. The arrow in the figure shows the center of the arc that defines the curvature of the bridge.
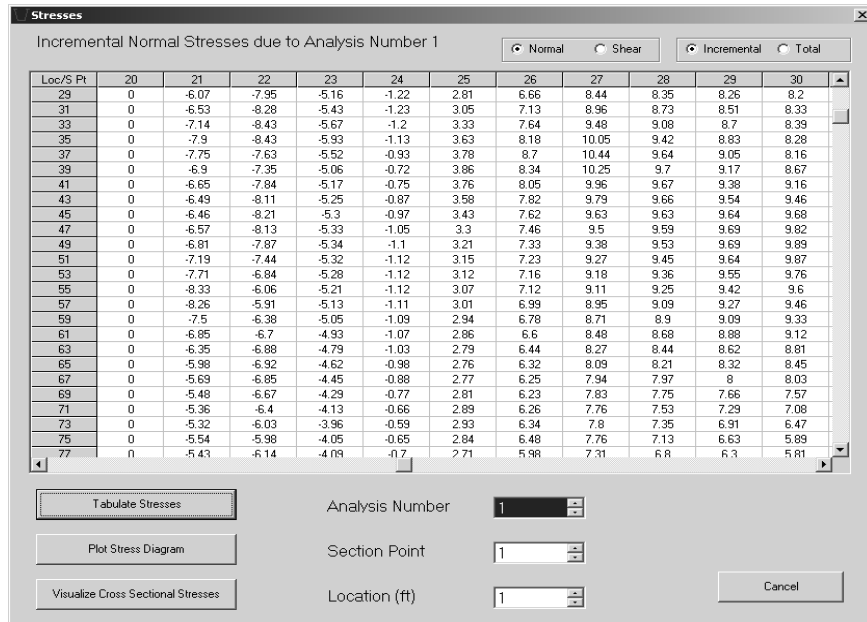
Figure 2.19: Tabulated Cross Sectional Stresses

Loc/S Pt	20	21	22	23	24	25	26	27	28	29	30
29	0	-6.07	-7.95	-5.16	-1.22	2.81	6.66	8.44	8.35	8.26	8.2
31	0	-6.53	-8.28	-5.43	-1.23	3.05	7.13	8.96	8.73	8.51	8.33
33	0	-7.14	-8.43	-5.67	-1.2	3.33	7.64	9.48	9.08	8.7	8.39
35	0	-7.9	-8.43	-5.93	-1.13	3.63	8.18	10.05	9.42	8.83	8.28
37	0	-7.75	-7.63	-5.52	-0.93	3.78	8.7	10.44	9.64	9.05	8.16
39	0	-6.9	-7.35	-5.06	-0.72	3.86	8.34	10.25	9.7	9.17	8.67
41	0	-6.65	-7.84	-5.17	-0.75	3.76	8.05	9.96	9.67	9.38	9.16
43	0	-6.49	-8.11	-5.25	-0.87	3.58	7.82	9.79	9.66	9.54	9.46
45	0	-6.46	-8.21	-5.3	-0.97	3.43	7.62	9.63	9.63	9.64	9.68
47	0	-6.57	-8.13	-5.33	-1.05	3.3	7.46	9.5	9.59	9.69	9.82
49	0	-6.81	-7.87	-5.34	-1.1	3.21	7.33	9.38	9.53	9.69	9.89
51	0	-7.19	-7.44	-5.32	-1.12	3.15	7.23	9.27	9.45	9.64	9.87
53	0	-7.71	-6.84	-5.28	-1.12	3.12	7.16	9.18	9.36	9.55	9.76
55	0	-8.33	-6.06	-5.21	-1.12	3.07	7.12	9.11	9.25	9.42	9.6
57	0	-8.26	-5.91	-5.13	-1.11	3.01	6.99	8.95	9.09	9.27	9.46
59	0	-7.5	-6.38	-5.05	-1.09	2.94	6.78	8.71	8.9	9.09	9.33
61	0	-6.85	-6.7	-4.93	-1.07	2.86	6.6	8.48	8.68	8.88	9.12
63	0	-6.35	-6.88	-4.79	-1.03	2.79	6.44	8.27	8.44	8.62	8.81
65	0	-5.98	-6.92	-4.62	-0.98	2.76	6.32	8.09	8.21	8.32	8.45
67	0	-5.69	-6.85	-4.45	-0.88	2.77	6.25	7.94	7.97	8	8.03
69	0	-5.48	-6.67	-4.29	-0.77	2.81	6.23	7.83	7.75	7.66	7.57
71	0	-5.36	-6.4	-4.13	-0.66	2.89	6.26	7.76	7.53	7.29	7.08
73	0	-5.32	-6.03	-3.96	-0.59	2.93	6.34	7.8	7.35	6.91	6.47
75	0	-5.54	-5.98	-4.05	-0.65	2.84	6.48	7.76	7.13	6.63	5.89
77	0	-5.43	-6.14	-4.09	-0.7	2.71	5.98	7.31	6.8	6.3	5.81

Figure 2.20: Stress Diagram For Section Point 52

Top Lateral Forces Submenu: This submenu is used to display the forces in the top lateral braces. Force values can be tabulated or visualized as a bar graph. Forces due to each analysis or total forces after each analysis case can be displayed. Four buttons are used to control the output in this form. Functions of these buttons are explained below.

Tabulate Incremental Forces: This button is used to tabulate the forces in the top lateral members due to each analysis case. Positive values correspond to tension forces in the brace members. This convention is used throughout the program. The *Top Lateral Forces* form, along with the tabulated results, are shown in Figure 2.22 .

28

Figure 2.21: Cross-Sectional Stresses

Top Lateral Forces

Incremental Forces (kips)

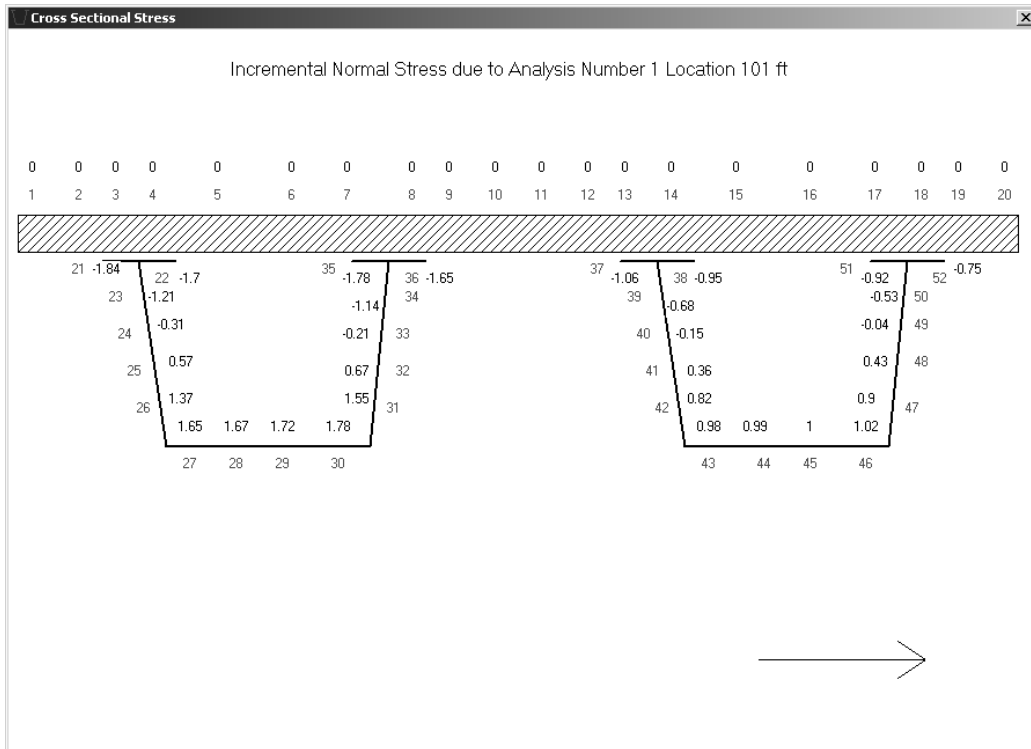| Brace #	Type	Loc1 (ft)	Loc2 (ft)		Analysis 1	Analysis 2	Analysis 3
1	2	0	18.9		-51.58	-2.07	8.2
2	1	18.9	37.8		12.01	1.07	-4.17
3	2	37.8	56.7		-42.62	-2.74	11.87
4	1	56.7	75.6		-23.58	0.99	-4.62
5	2	75.6	94.5		0.14	-5.21	37.13
6	1	94.5	113.4		-30.68	-5.26	33.86
7	2	113.4	132.3		28.65	-0.38	7.6
8	1	132.3	151.5		-20.21	-11.38	71.02
9	2	151.5	170.5		34.26	5.67	-32.06
10	1	170.5	189.5		-6.15	-13.85	66.72
11	2	189.5	208.5		19.59	11.61	-71.5
12	1	208.5	227.5		5.12	-9.38	7.42
13	2	227.5	246.5		2.19	12.4	-54.85
14	1	246.5	265.5		11.89	2.54	-55.89
15	2	265.5	284.5		-9.28	5.32	9.44
16	1	284.5	303.5		11.48	20.8	-70.81
17	2	303.5	322.5		-13.83	-6.74	67.49
18	1	322.5	341.5		7.07	35.32	-33.64
19	2	341.5	360.4		-12.91	-21.16	73.99
20	1	360.4	379.3		2.07	29.71	5.21
21	2	379.3	398.3		-7.15	-32.14	35.69
22	1	398.3	417.3		-2.94	-0.61	34.92
23	2	417.3	436.1		-0.09	-24.03	-4.51
24	1	436.1	455.1		-6.41	-44.7	18.63
25	2	455.1	474		4.99	13.58	-7.98
26	1	474	493		-6.99	-53.47	14.82
27	2	0	18.9		-48.79	-2.19	8.44
28	1	18.9	37.8		19.22	0.99	-3.54
29	2	37.8	56.7		-35.1	-2.94	13.68

Tabulate Incremental Forces | Tabulate Total Forces | Plot Incremental Forces | Plot Total Forces | Cancel

Figure 2.22: Tabulated Top Lateral Brace Forces

Tabulate Total Forces: This button is used to tabulate the forces in top lateral members after each analysis case. Values of all previous analyses are summed.

Plot Incremental Forces: This button is used to display the bar chart of top lateral brace forces. Incremental force values due to each analysis case are displayed. Figure 2.23 shows a typical bar chart of brace forces.
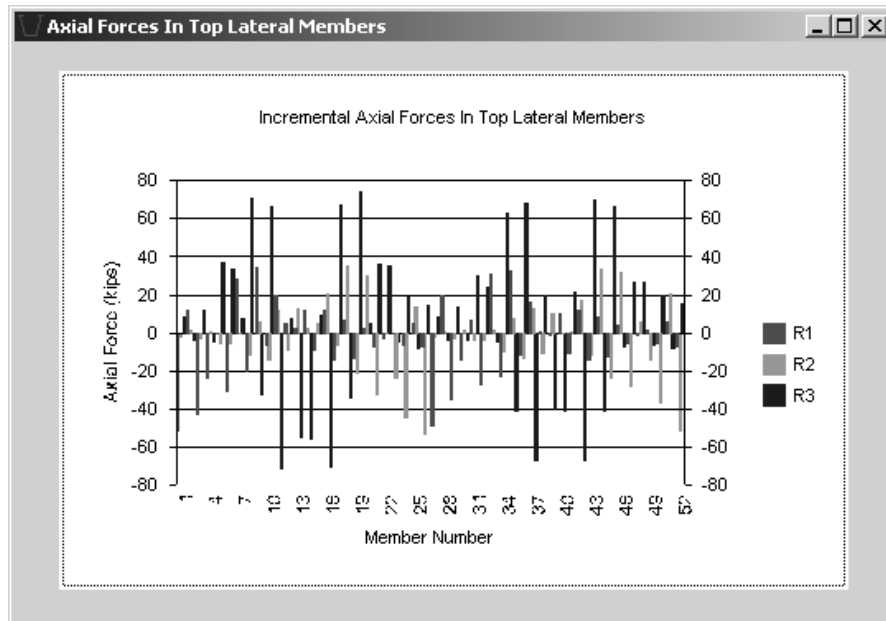

Figure 2.23: Bar Chart of Top Lateral Forces

Plot Total Forces: This button is used to display a bar chart of top lateral brace forces. Total force values after each analysis case are displayed.

Internal Brace Forces and External Brace Forces Submenus: These submenus are used to display the member axial forces for internal and external braces. Because they have identical properties, both menus will be explained in this section. Axial force values can be tabulated or visualized as a bar graph. Axial forces due to each analysis or total forces after each analysis case can be displayed. Four buttons that act together with a scroll-down box are used to control the output in these forms. Functions of these buttons are explained below.

Tabulate Incremental Forces: This button is used to tabulate the forces in brace members due to each analysis case. Because internal and external braces are made up of several members, only results for a certain member can be displayed. Therefore, the member number must be selected using the scroll-down box. The configuration of internal and external braces, and the corresponding member numbers were presented previously (Figure 2.6). Figure 2.24 shows the *Internal Brace Forces* form together with the table of axial force values for member 2 of all internal braces.

Tabulate Total Forces: This button is similar to the *Tabulate Incremental Forces* button. It is used to tabulate the total forces after each analysis.

Plot Incremental Forces: This button is used to display a bar chart of axial force values for a certain member number. The member number must be selected using the scroll-down box. Figure 2.25 shows a bar chart of axial force values for member 2 of all internal braces.

Plot Total Forces: This button is similar to the *Plot Total Forces* button. It is used to display the total forces after each analysis.
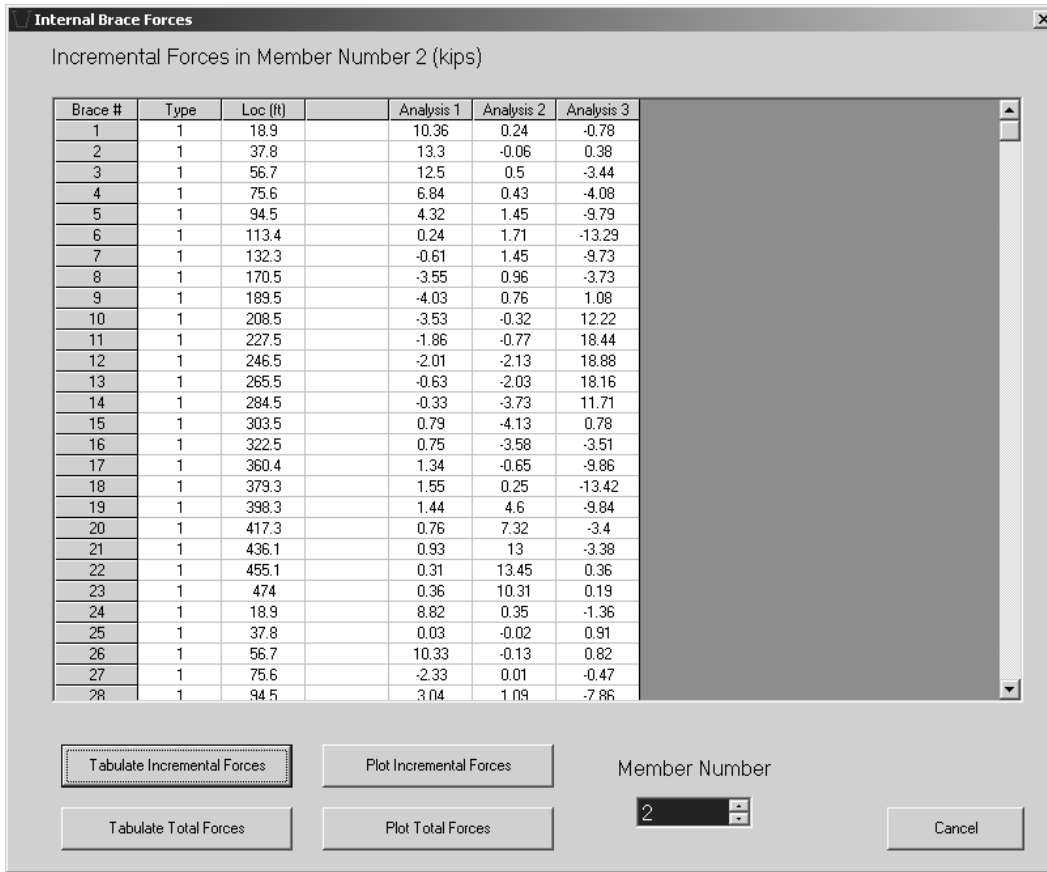
Internal Brace Forces

Incremental Forces in Member Number 2 (kips)

Brace #	Type	Loc (ft)		Analysis 1	Analysis 2	Analysis 3
1	1	18.9		10.36	0.24	-0.78
2	1	37.8		13.3	-0.06	0.38
3	1	56.7		12.5	0.5	-3.44
4	1	75.6		6.84	0.43	-4.08
5	1	94.5		4.32	1.45	-9.79
6	1	113.4		0.24	1.71	-13.29
7	1	132.3		-0.61	1.45	-9.73
8	1	170.5		-3.55	0.96	-3.73
9	1	189.5		-4.03	0.76	1.08
10	1	208.5		-3.53	-0.32	12.22
11	1	227.5		-1.86	-0.77	18.44
12	1	246.5		-2.01	-2.13	18.88
13	1	265.5		-0.63	-2.03	18.16
14	1	284.5		-0.33	-3.73	11.71
15	1	303.5		0.79	-4.13	0.78
16	1	322.5		0.75	-3.58	-3.51
17	1	360.4		1.34	-0.65	-9.86
18	1	379.3		1.55	0.25	-13.42
19	1	398.3		1.44	4.6	-9.84
20	1	417.3		0.76	7.32	-3.4
21	1	436.1		0.93	13	-3.38
22	1	455.1		0.31	13.45	0.36
23	1	474		0.36	10.31	0.19
24	1	18.9		8.82	0.35	-1.36
25	1	37.8		0.03	-0.02	0.91
26	1	56.7		10.33	-0.13	0.82
27	1	75.6		-2.33	0.01	-0.47
28	1	94.5		3.04	1.09	-7.86

Tabulate Incremental Forces   Plot Incremental Forces   Member Number

Tabulate Total Forces   Plot Total Forces   2   Cancel
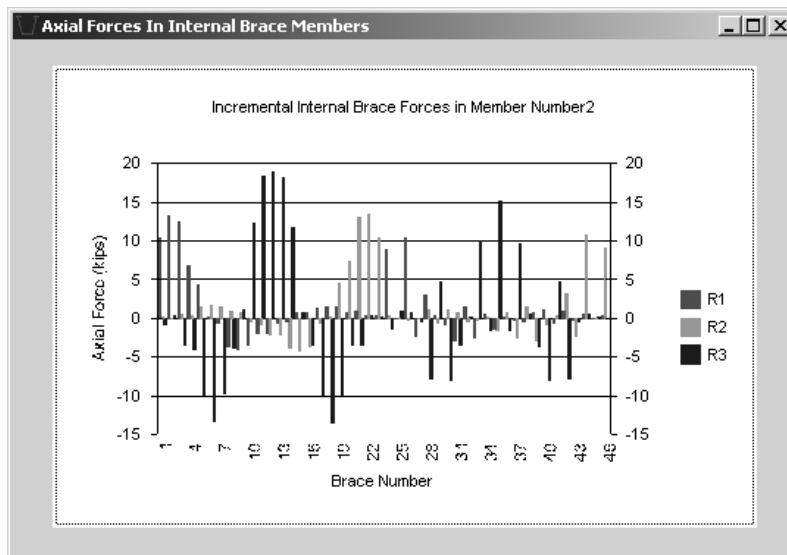
Figure 2.24: Internal Brace Forces Form

Figure 2.25: Bar Chart of Internal Brace Forces

31

Analysis Summary Submenu: This submenu is used to display the maximum values of several important quantities for each analysis. Maximum deflection, shear force, axial stress etc. are tabulated in this form. The Analysis Summary form is shown in Figure 2.26.

Analysis Summary

Summary of Maximum Quantities

Quantity	Value	Location (ft)		Value	Location (ft)
		Analysis 1			Analysis 2
Incremental Upward Deflection (in)	1.61	222		1.65	264
Incremental Downward Deflection (in)	-3.11	60		-3.29	426
Total Upward Deflection (in)	1.61	222		3.06	244
Total Downward Deflection (in)	-3.11	60		-3.79	424
Incremental Rotation (rad)	0.005244	74		0.005571	418
Total Rotation (rad)	0.005244	74		0.00668	418
Incremental Shear (kips)	162.3	1		168.3	491
Incremental Positive Moment (kip-ft)	3911.9	149		4020.9	339
Incremental Negative Moment (kip-ft)	-2055.8	149		-2144.4	339
Incremental Torque(kip-ft)	384.9	1		400.6	491
Incremental Positive Normal Stress (ksi)	10.44	37		10.7	455
Incremental Negative Normal Stress (ksi)	-8.43	33		-8.69	457
Incremental Shear Stress (ksi)	2.08	3		2.12	489
Total Positive Normal Stress (ksi)	10.44	37		11.31	455
Total Negative Normal Stress (ksi)	-8.43	33		-9.33	437
Total Shear Stress (ksi)	2.08	3		2.24	489
Incremental Positive Top Lateral Axial Force (kips)	34.26	9		35.32	18
Incremental Negative Top Lateral Axial Force (kips)	-51.58	1		-53.47	26
Total Positive Top Lateral Axial Force (kips)	34.26	9		42.39	18
Total Negative Top Lateral Axial Force (kips)	-51.58	1		-60.46	26
Incremental Positive Top Lateral Axial Stress (ksi)	5.429477	9		5.597465	18
Incremental Negative Top Lateral Axial Stress (ksi)	-8.174327	1		-8.473851	26
Total Positive Top Lateral Axial Stress (ksi)	5.429477	9		6.717908	18

Figure 2.26: Analysis Summary Form

2.1.1.1 Final Comments

After an analysis is performed, the user can reanalyze the system by making modifications to the geometry or the pouring sequence.

The program works under Windows 98, Windows 2000, and Windows XP operating systems. A physical memory of 1 GB is recommended for problems involving twin girders. In cases where the physical memory is not sufficient, the program uses the computer's virtual memory. Using virtual memory, however, significantly increases the time needed to obtain a solution.