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.

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 Fig. 1.

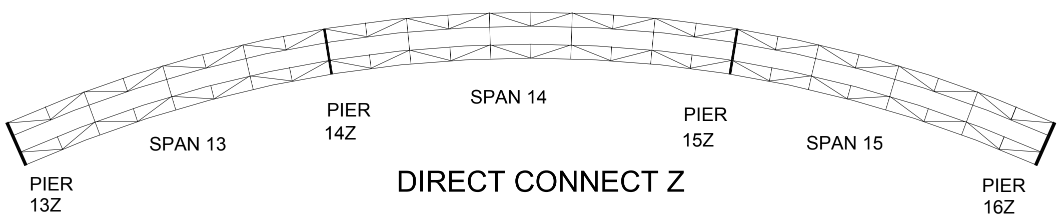


Figure 1: Plan View of Direct Connect Z

UTrAp accepts only positive values for the radius of curvature, and the concavity layout of the structure should be similar to the one in Fig. 1. Therefore, the left end is considered to be the start end of the bridge. In Fig. 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. Fig. 2 provides cross-sectional dimensions of Direct

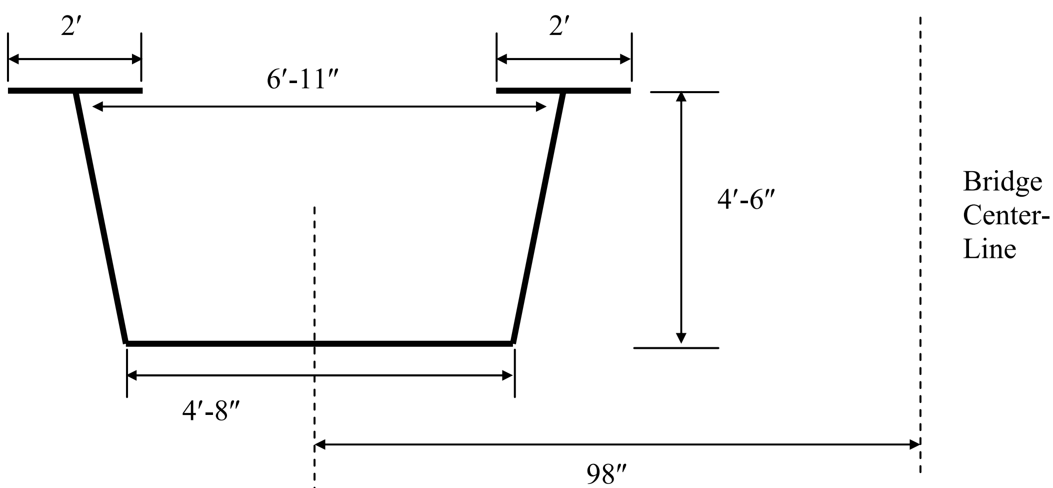


Figure 2: Cross-sectional Dimensions

Connect Z. 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.

The steel plates that make up the girder have variable thickness along the length of the bridge. Table 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 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. 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 Fig. 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 Fig. 3.

Table 2: Location of Braces

	Internal Bracing	**External Bracing**	**Top Lateral Bracing**	
Brace Number	Location (ft)	Location (ft)	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

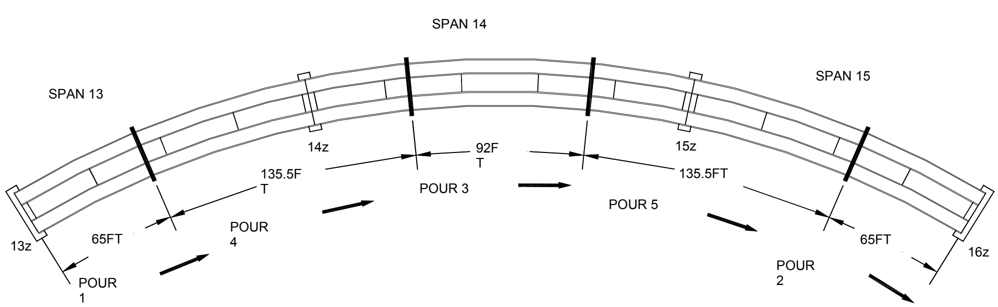


Figure 3: Concrete Pour Sequence

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 3.

Table 3: Pour Sequence Analysis Parameters

		Analysis 1			**Analysis 2**			**Analysis 3**		
Deck	Length	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									

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.

Example Problem: Geometric property values are entered on the form and saved by making use of the *Save Data* button. Figure 4 shows the Geometric Properties form with the entered data.

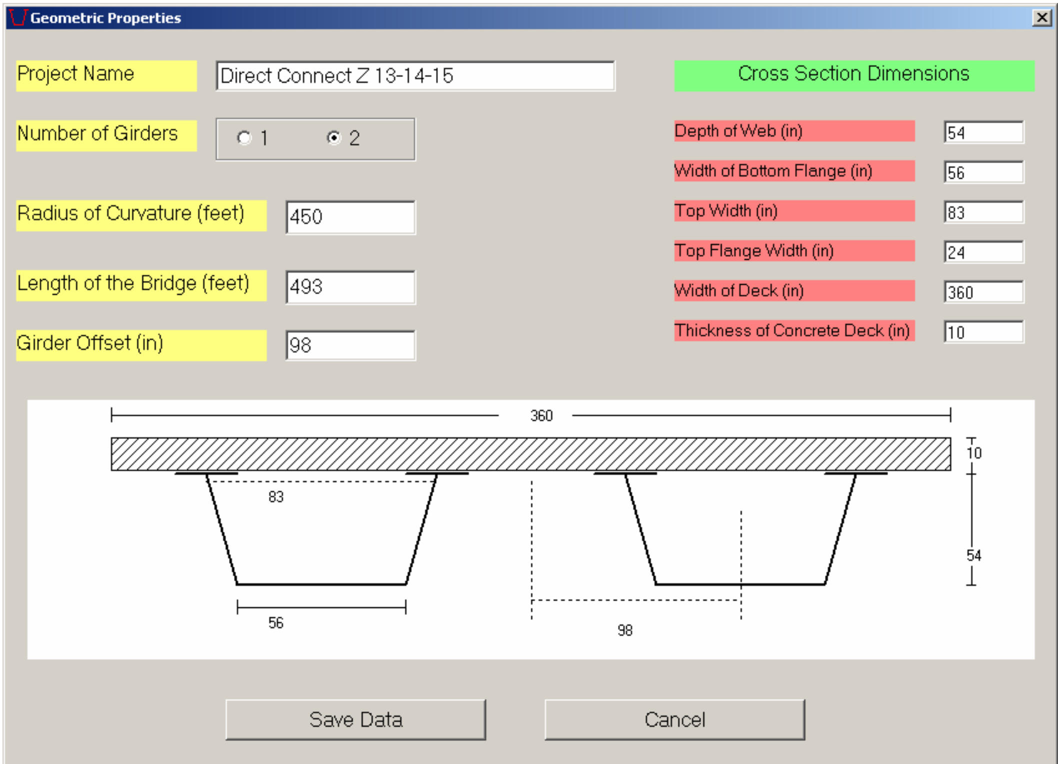


Figure 4: Geometric Properties Form

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.

Example Problem: 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 Fig. 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.

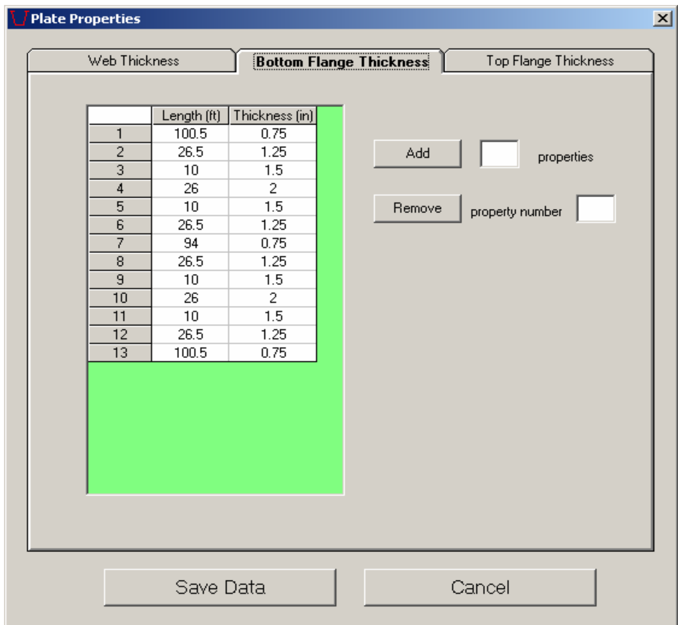


Figure 5: Plate Properties Form

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 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 Fig. 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.

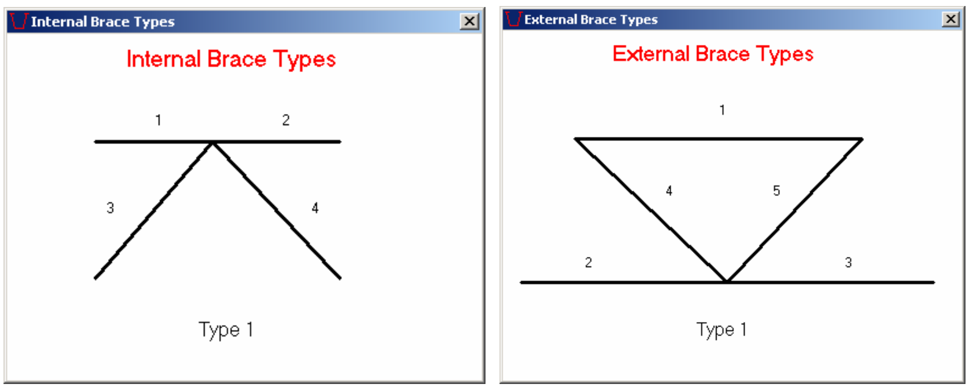


Figure 6: Internal and External Brace Types

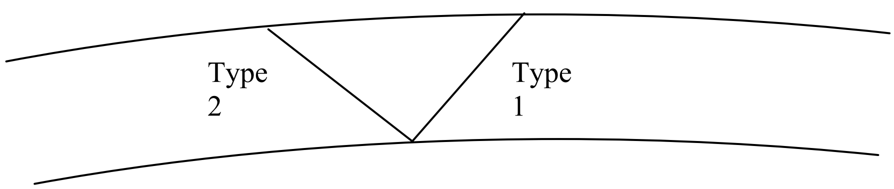


Figure 7: Top Lateral Brace Types

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 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.

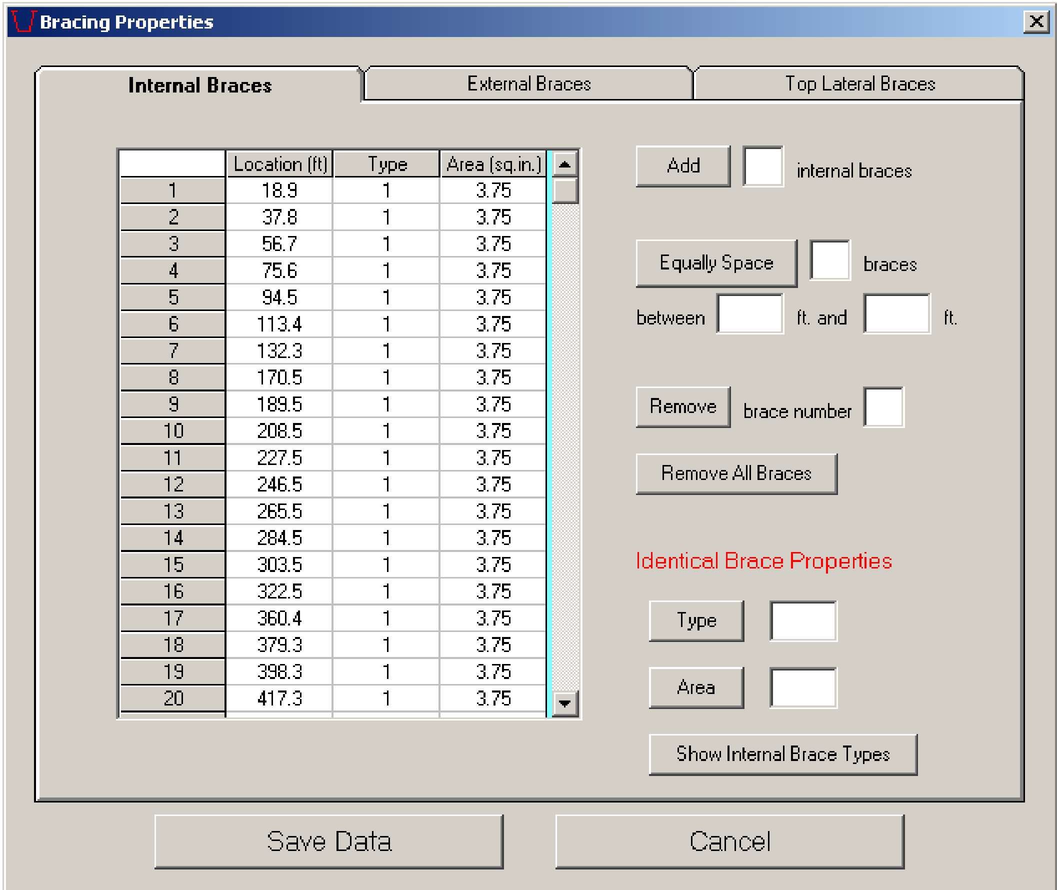
Example Problem: Twenty-three internal braces, 10 external braces and 26 top lateral braces are added to the folders by making use of the *Add* button. Brace locations, types and cross-sectional areas are entered into the folders according to the information given in Table 2. All internal and external 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.

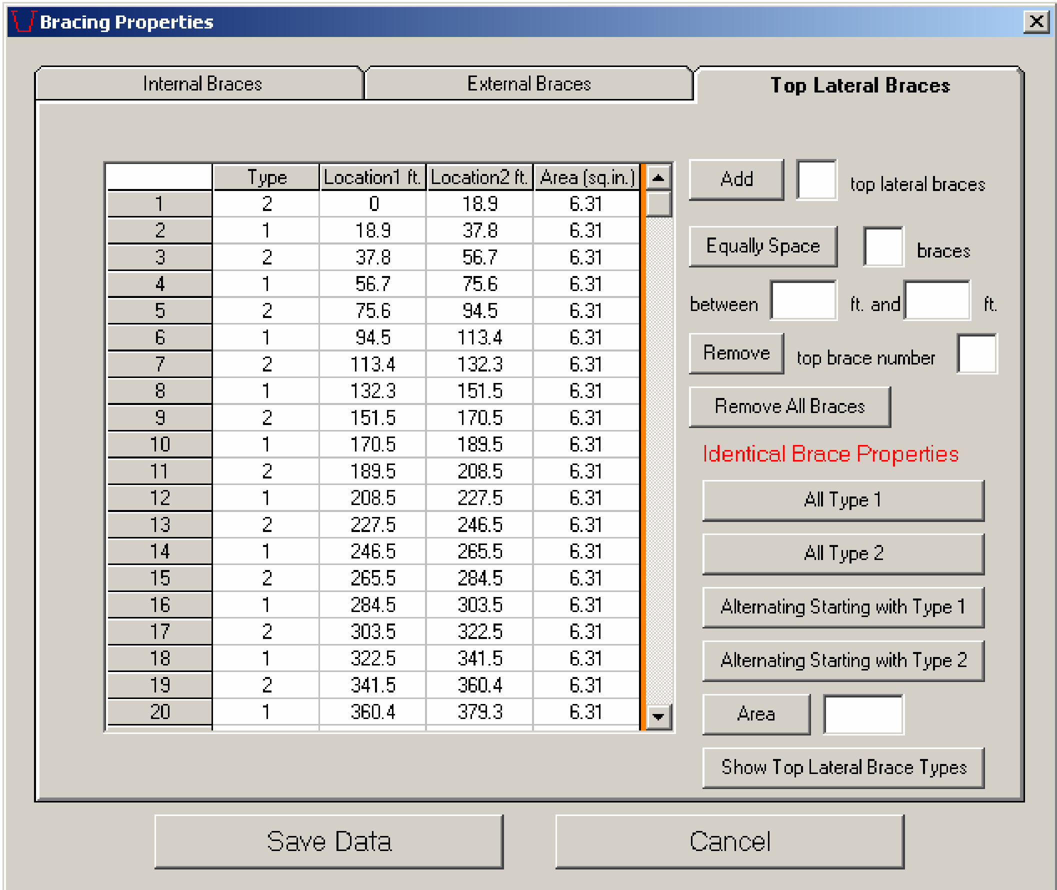
Remove: This button is used to remove supports. The support number that is going to be removed should be specified in the box next to the *Remove* button.

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 (Fig. 1) are entered in the table. Figure 10 shows the support locations form along with the entered data.



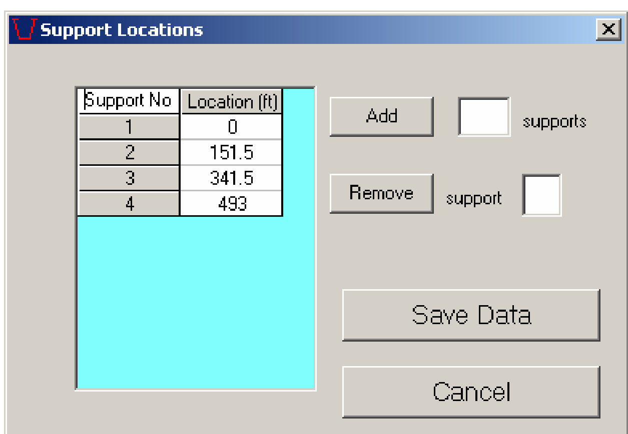
	Location (ft)	Type	Area (sq.in.)
1	18.9	1	3.75
2	37.8	1	3.75
3	56.7	1	3.75
4	75.6	1	3.75
5	94.5	1	3.75
6	113.4	1	3.75
7	132.3	1	3.75
8	170.5	1	3.75
9	189.5	1	3.75
10	208.5	1	3.75
11	227.5	1	3.75
12	246.5	1	3.75
13	265.5	1	3.75
14	284.5	1	3.75
15	303.5	1	3.75
16	322.5	1	3.75
17	360.4	1	3.75
18	379.3	1	3.75
19	398.3	1	3.75
20	417.3	1	3.75

Figure 8: Bracing Properties Form - Internal Braces Folder



	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

Figure 9: Bracing Properties Form – Top Lateral Braces Folder



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

Figure 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 11 shows the stud properties form along with the entered data.

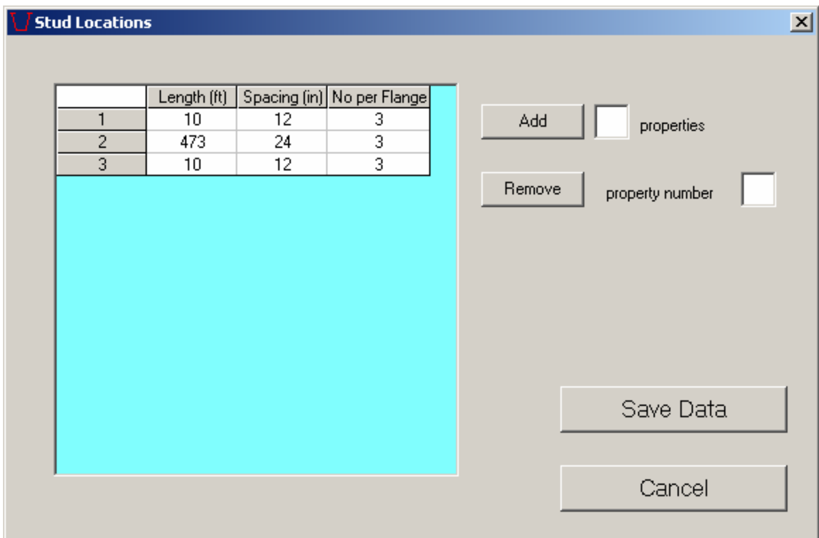


Figure 11: Stud Properties Form

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.

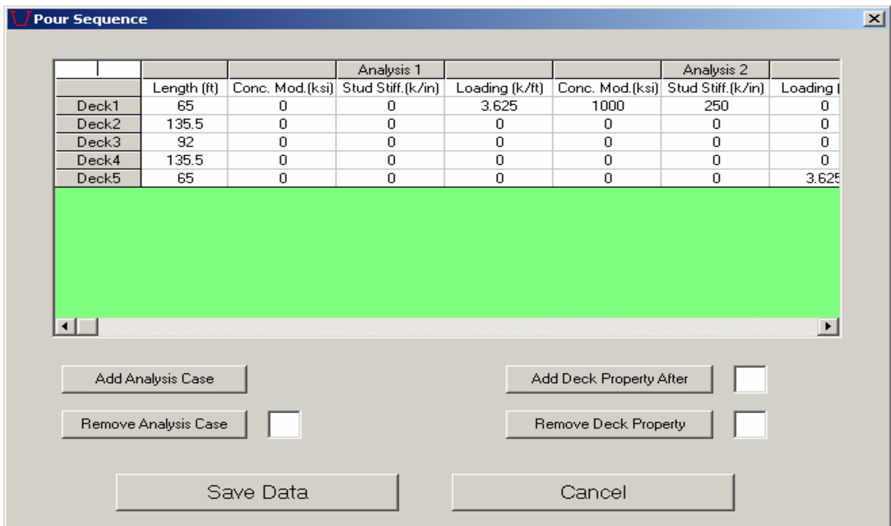
Add Analysis Case: This button is used to add a new analysis case to the table. Three columns for analysis parameters are added to the right of the table each time a new analysis is added. The three new columns are used to specify the concrete stiffness, the stud stiffness, and any loading acting on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

<u>Example Problem</u>: 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 3. Figure 12 shows the pour sequence form together with the input data.


Pour Sequence

			Analysis 1			Analysis 2	
	Length (ft)	Conc. Mod.(ksi)	Stud Stiff.(k/in)	Loading (k/ft)	Conc. Mod.(ksi)	Stud Stiff.(k/in)	Loading (
Deck1	65	0	0	3.625	1000	250	0
Deck2	135.5	0	0	0	0	0	0
Deck3	92	0	0	0	0	0	0
Deck4	135.5	0	0	0	0	0	0
Deck5	65	0	0	0	0	0	3.625

Figure 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 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 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.

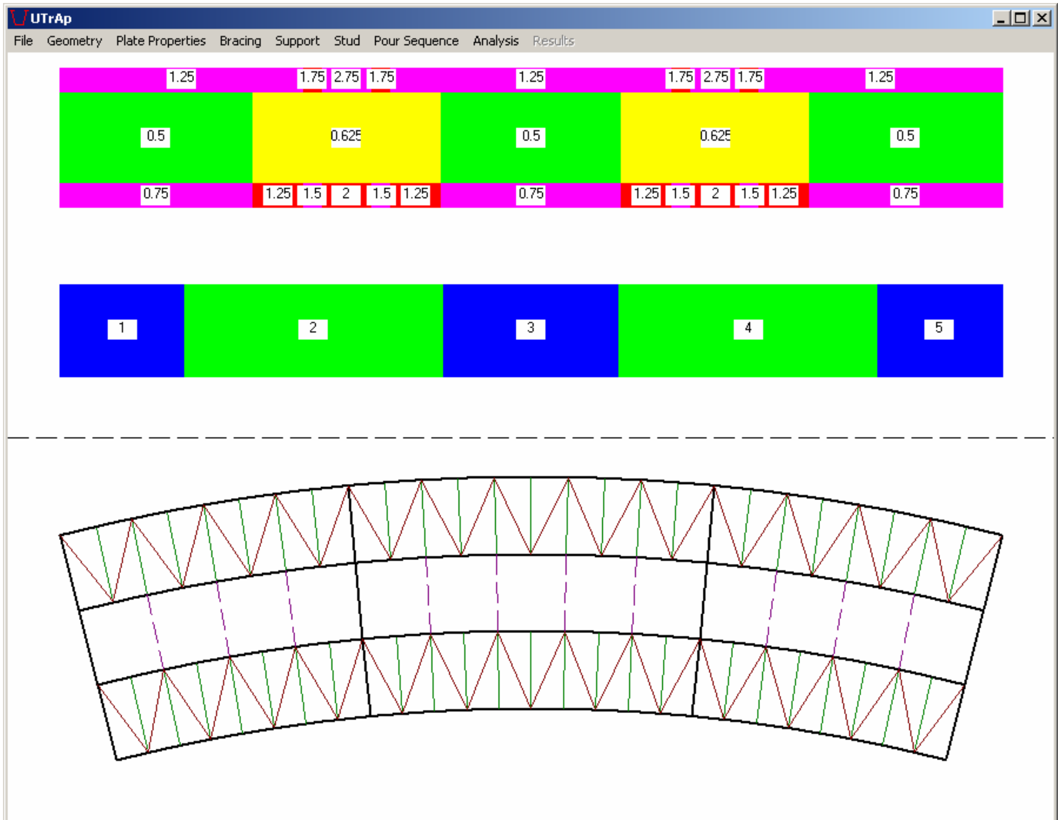


Figure 13: Main form of UTrAp

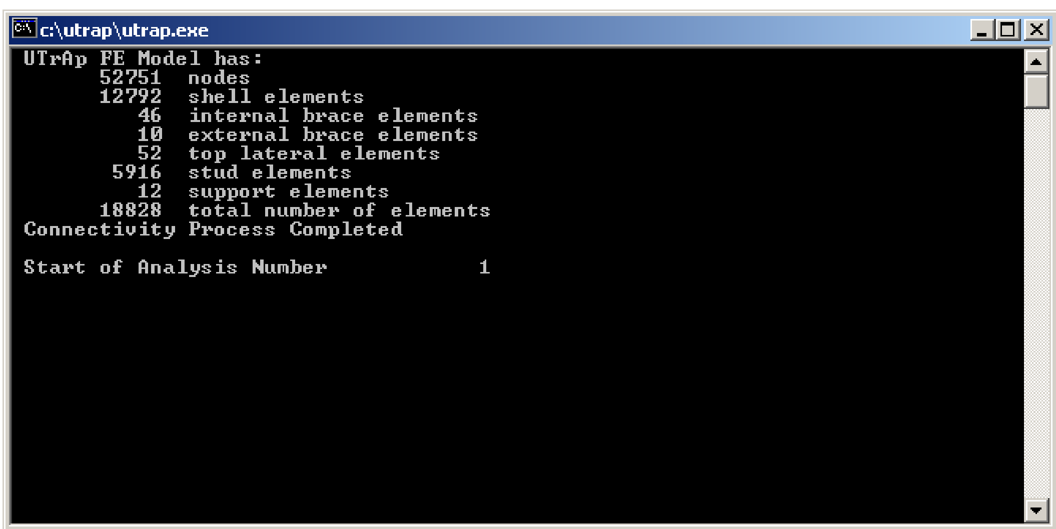


Figure 14: DOS Screen for an Analysis

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 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 15 shows the deflections form with the results for the example bridge.

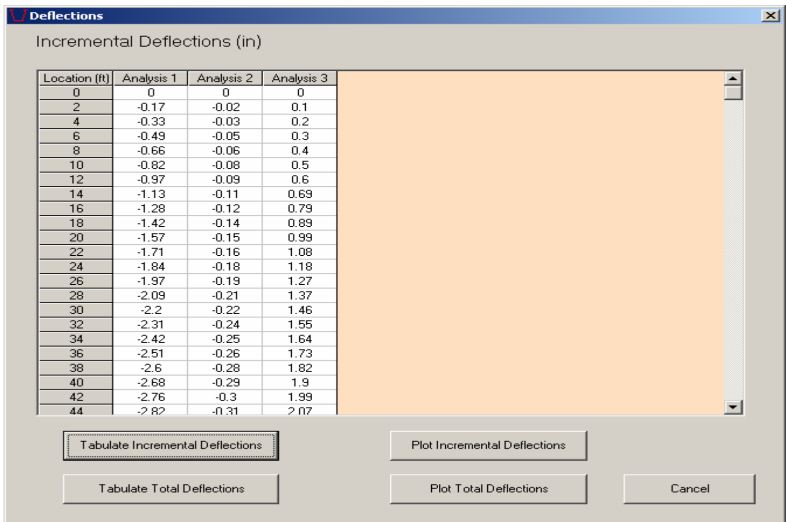


Figure 15: Deflections Form

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 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 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 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.

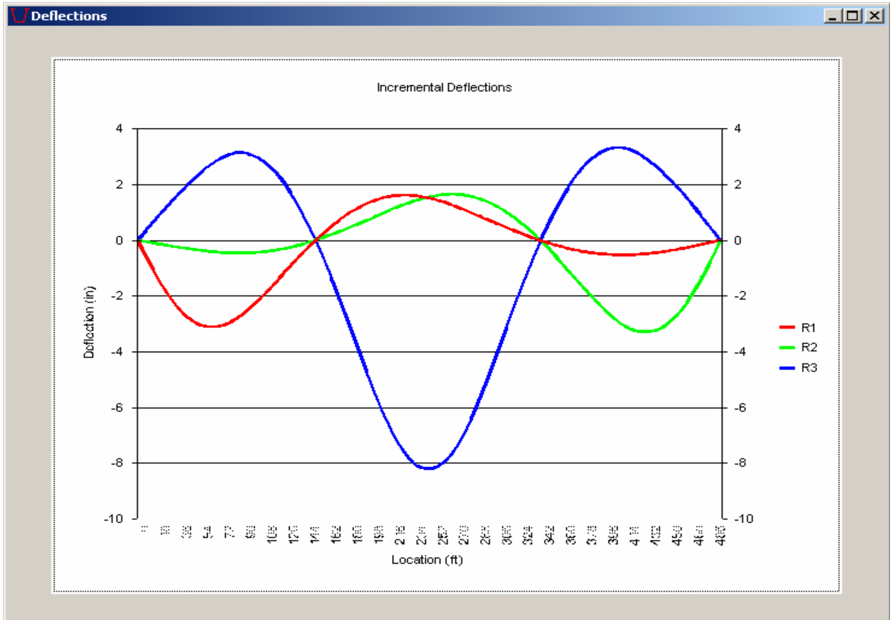
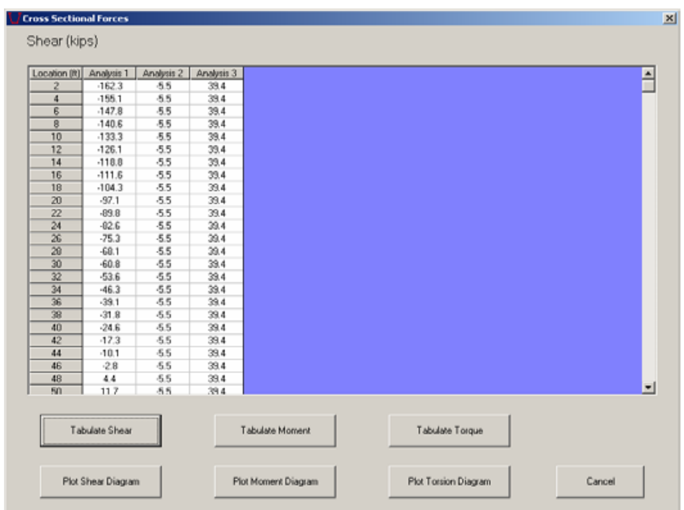


Figure 16: Deflection Diagram



Location (ft)	Analysis 1	Analysis 2	Analysis 3
2	-162.3	-5.5	39.4
4	-155.1	-5.5	39.4
6	-147.8	-5.5	39.4
8	-140.6	-5.5	39.4
10	-133.3	-5.5	39.4
12	-126.1	-5.5	39.4
14	-118.8	-5.5	39.4
16	-111.6	-5.5	39.4
18	-104.3	-5.5	39.4
20	-97.1	-5.5	39.4
22	-89.8	-5.5	39.4
24	-82.6	-5.5	39.4
26	-75.3	-5.5	39.4
28	-68.1	-5.5	39.4
30	-60.8	-5.5	39.4
32	-53.6	-5.5	39.4
34	-46.3	-5.5	39.4
36	-39.1	-5.5	39.4
38	-31.8	-5.5	39.4
40	-24.6	-5.5	39.4
42	-17.3	-5.5	39.4
44	-10.1	-5.5	39.4
46	-2.8	-5.5	39.4
48	4.4	-5.5	39.4
50	11.7	-5.5	39.4

Figure 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 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.

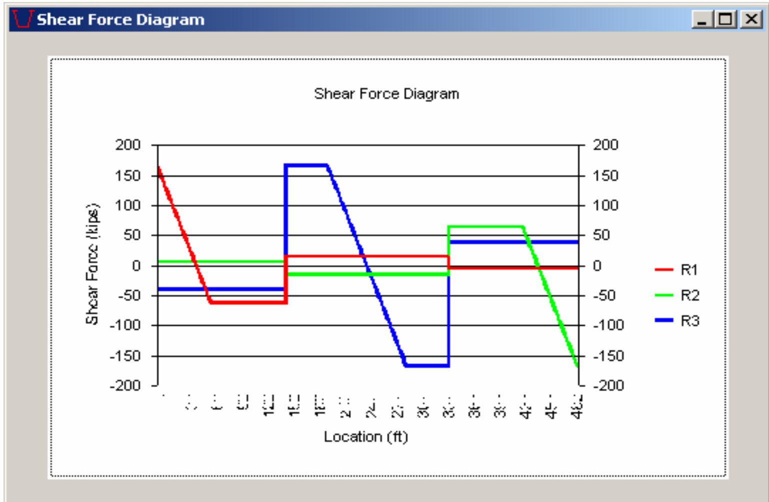


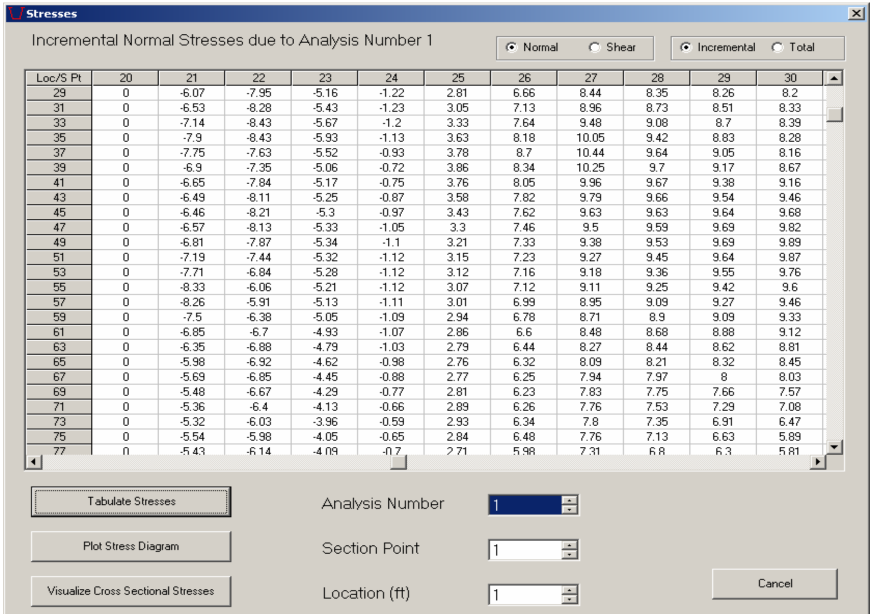
Figure 18: Shear Force Diagram

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 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 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 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.



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 19: Tabulated Cross Sectional Stresses

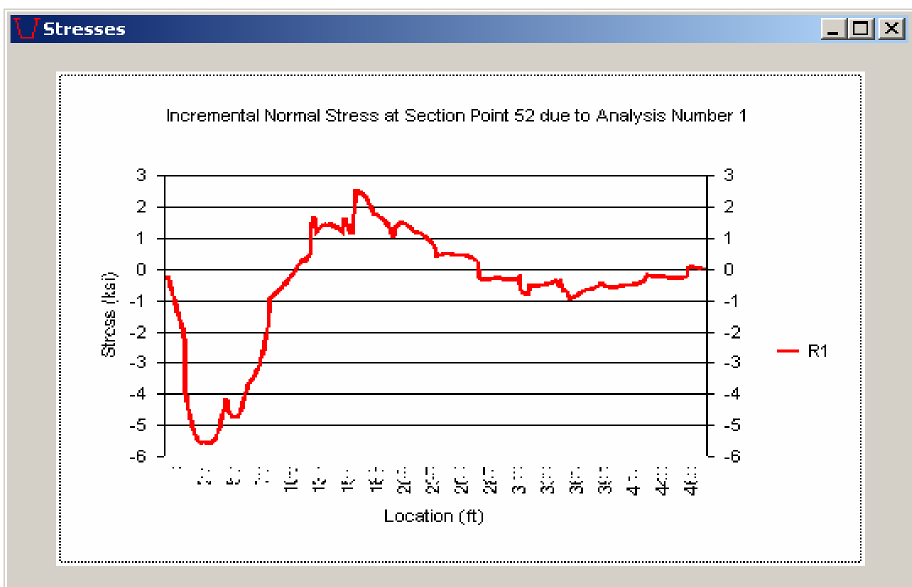


Figure 20: Stress Diagram For Section Point 52

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 Fig. 22 .

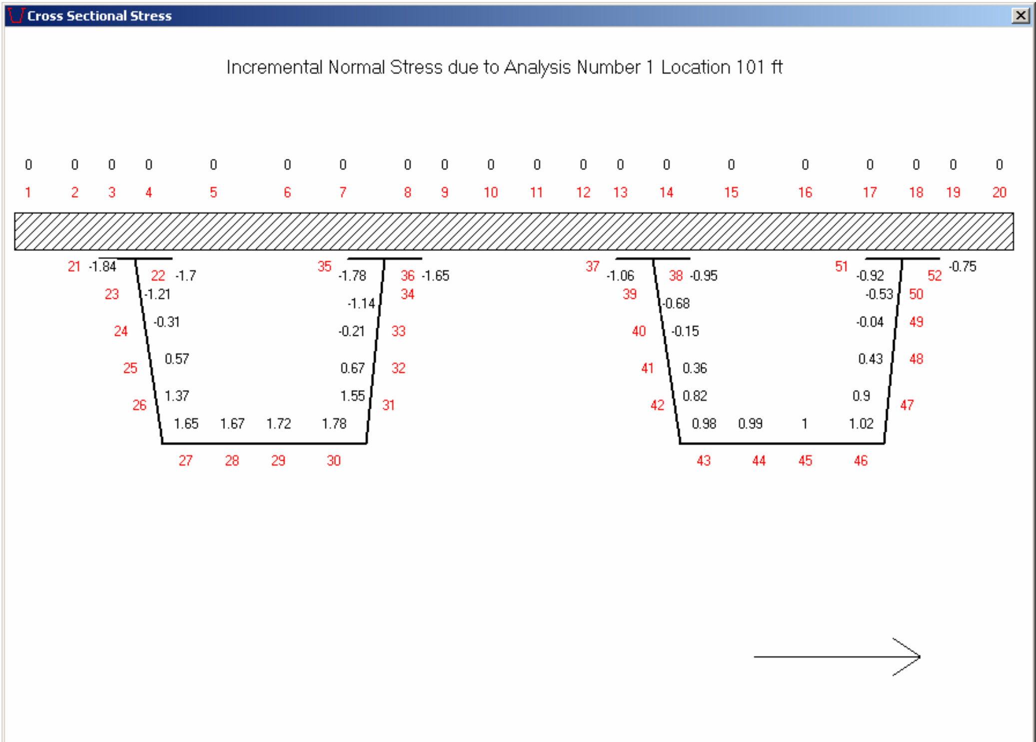
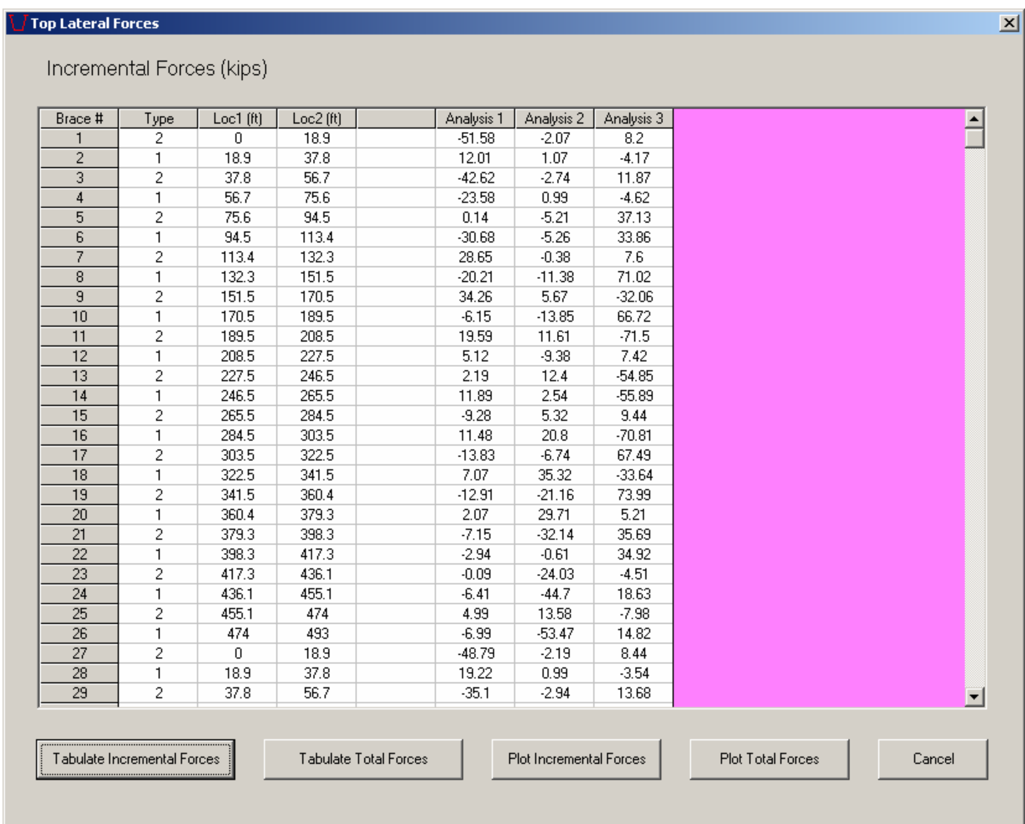


Figure 21: Cross-Sectional Stresses



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

Figure 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 23 shows a typical bar chart of brace forces.

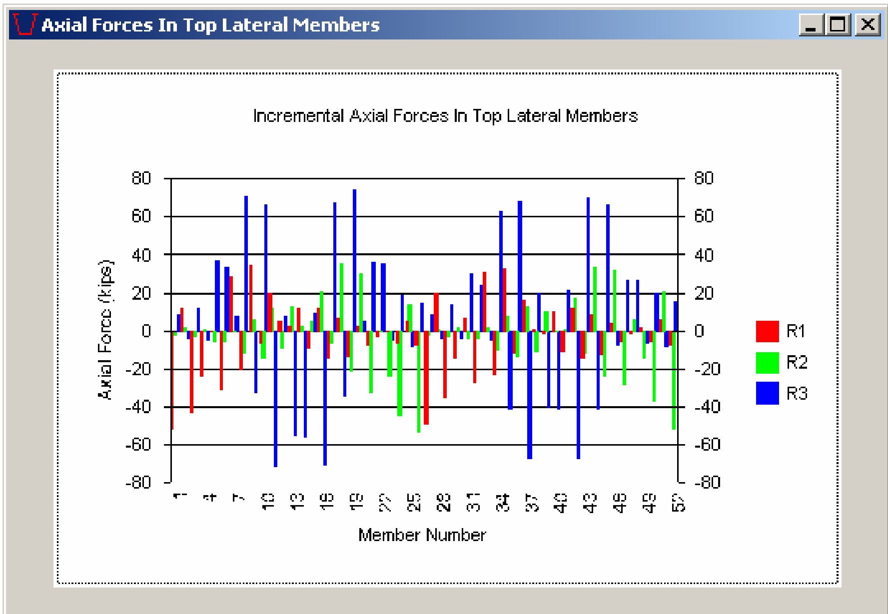


Figure 23: Bar Chart of Top Lateral Forces

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.

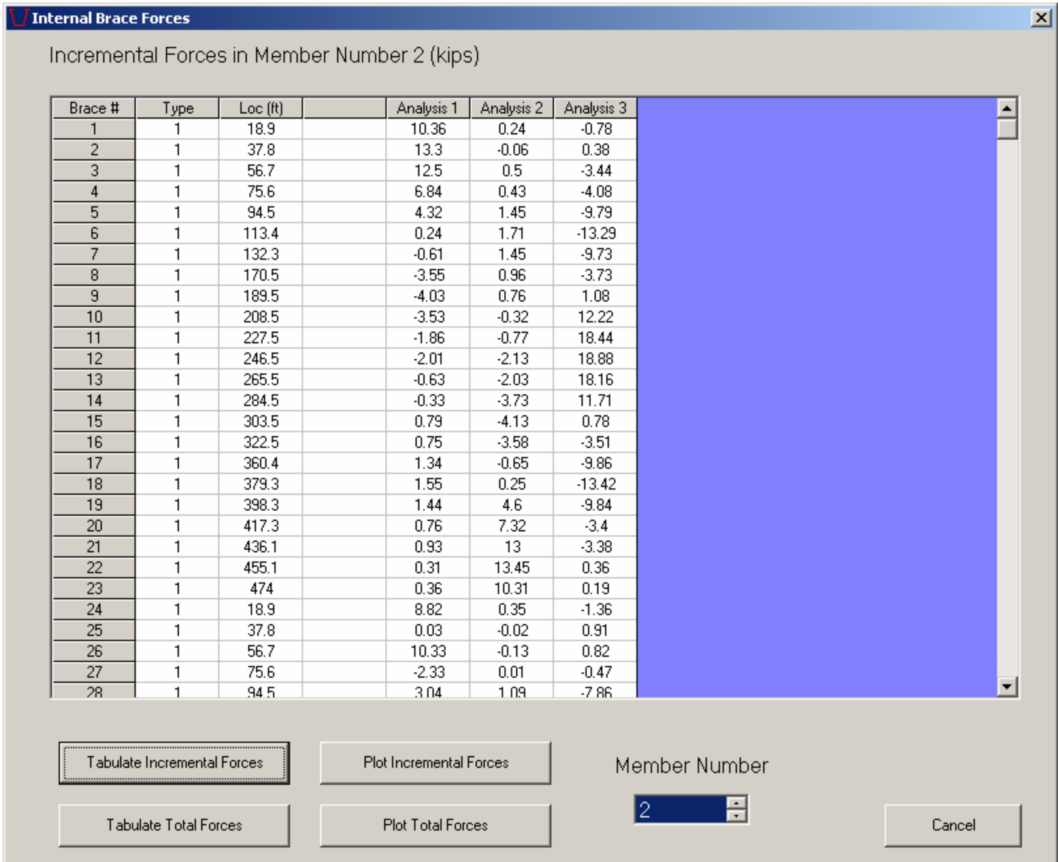
Internal Brace Forces and External Brace Forces Submenus: These submenus are used to display the member axial forces for internal and external braces. Because they have identical properties, both menus will be explained in this section. Axial force values can be tabulated or visualized as a bar graph. 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 (Fig. 6). Figure 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 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.



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

Figure 24: Internal Brace Forces Form

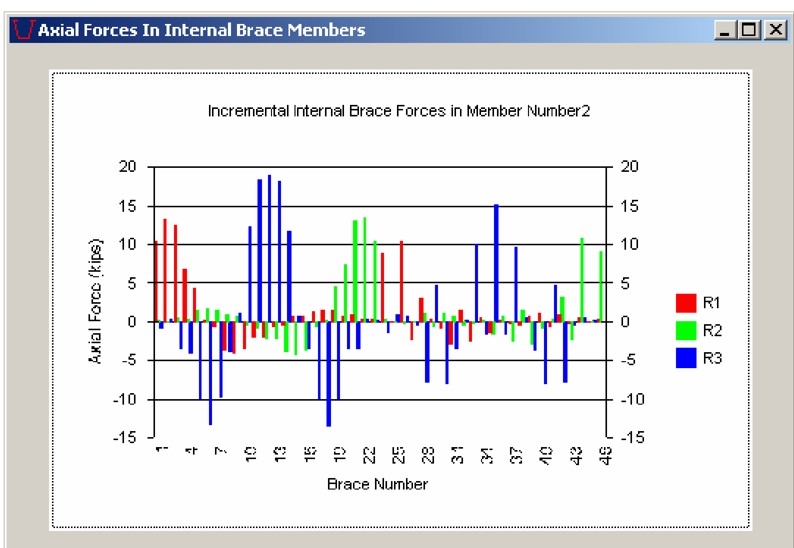
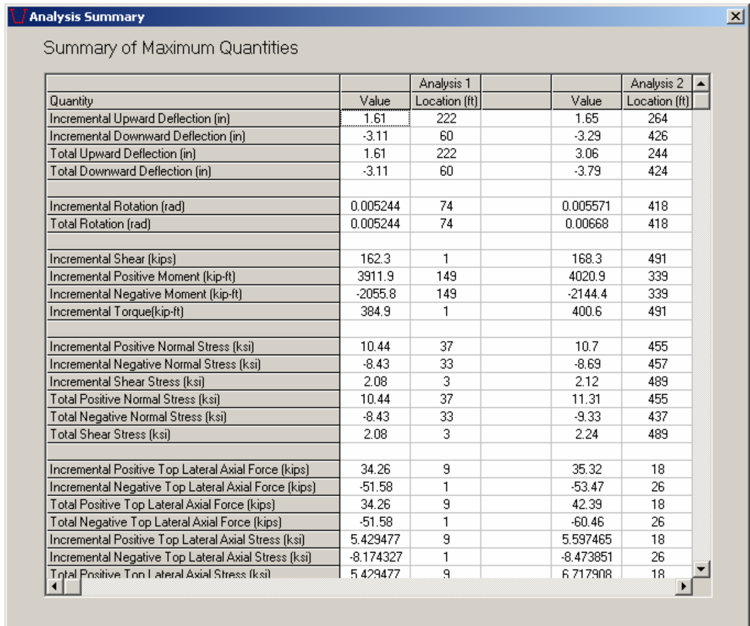


Figure 25: Bar Chart of Internal Brace Forces

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 Fig. 26.


Analysis Summary

Summary of Maximum Quantities

		Analysis 1			Analysis 2
Quantity	Value	Location (ft)		Value	Location (ft)
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 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.