

UTrAp 2.0

Analysis of Steel Box Girders during Construction



developed at

The University of Texas at Austin

by

Eric Williamson

Cem Topkaya

Daniel Popp

Joseph Yura

October 2004

UTrAp 2.0 User's Guide

UTrAp 2.0 is a revised and expanded version of the program UTrAp, developed for the analysis of steel trapezoidal box-girder bridges. The program is designed for analysis of the bridge under construction loading, but can also be used to analyze the completed bridge. The major new feature is the capability to perform buckling analyses. Various other improvements have also been made, including the ability to add web stiffeners to the model and the inclusion of self-weight loading. In addition, numerous changes have been made to the user interface to increase the user-friendliness.

INSTALLATION

UTrAp 2.0 is offered as a free download. After downloading the compressed file from the UTrAp 2.0 webpage, the user must extract the UTrAp 2.0.msi file. Executing this file will open the Windows Installer. The Microsoft .NET Framework is required to run UTrAp 2.0, which is written in the .NET programming language. The framework can be obtained using Windows Update, or directly from the Microsoft website at

http://msdn.microsoft.com/netframework/downloads/framework1_1/

If this link is outdated, a quick search on the Microsoft home page should locate the proper link. After the .NET Framework is installed properly, the UTrAp 2.0 installation can proceed. The Windows Installer will ask the user to specify the program directory location, which by default is Program Files\UTrAp 2.0. Clicking "Install" will complete the installation process.

In order to run UTrAp, the user must obtain a license file, which is available at no charge. Directions for obtaining the file are found on the program download site. After the file is received, it must be placed in the program folder where UTrAp 2.0.exe is located. Also, the user must have permission to create and modify files in order to run the program, which creates text files for input and output.

UTrAp 2.0 is entirely backwards compatible with the original UTrAp program. Any input file (*.inp) created with the original program can be opened by UTrAp 2.0; however, after the file is saved by the new version, it will not open in the original version. All features of the original UTrAp are included in the updated version, so uninstalling the original program is recommended after installing UTrAp 2.0.

CAPABILITIES AND LIMITATIONS

UTrAp 2.0 is intended to perform both linear analyses and linearized buckling analyses of straight or curved steel trapezoidal box-girder bridges under construction loading. The program is able to model the partially-composite behavior due to concrete curing during the bridge deck pours. UTrAp 2.0 is limited to elastic analyses, and does not account for nonlinear material behavior. There are no limits on stresses in the girder, which may therefore exceed the yield stress. In order for the UTrAp analysis to be valid, care must be taken by the designer to ensure that the bridge remains elastic under the given loading.

In addition, buckling of individual bracing members is not captured by the UTrAp buckling analysis. Bracing members, which in reality are typically channels or angles, are modeled as truss members that carry only axial force. This simplification prevents the buckling of individual truss members, which are defined by a straight line between two points. The designer must ensure that each individual brace member will not buckle under its maximum expected load, obtained directly in UTrAp.

The bridge deck is modeled as a solid slab attached directly to the top flanges with shear studs. Concrete haunches are not considered.

Straight versus Curved Girders

Section 4.6.1.2 of the AASHTO LRFD Bridge Specifications pertains to structures curved in plan. Subsection 1 states:

Segments of horizontally curved superstructures with torsionally stiff closed sections whose central angle subtended by a curved span or portion thereof is less than 12.0° may be analyzed as if the segment were straight.

Of note in this provision is that the superstructure must be torsionally stiff, as in the case of a composite trapezoidal girder. The UTrAp program, however, analyzes the girders during the concrete pour, when the sections are only partially closed, with a stiffness that could be one or two orders of magnitude less than that of the completed section. Accordingly, UTrAp 2.0 analyzes sections with any curvature, however small, as curved sections. Only in the case of a perfectly straight bridge will the program treat the girder as straight.

GRAPHICAL USER INTERFACE

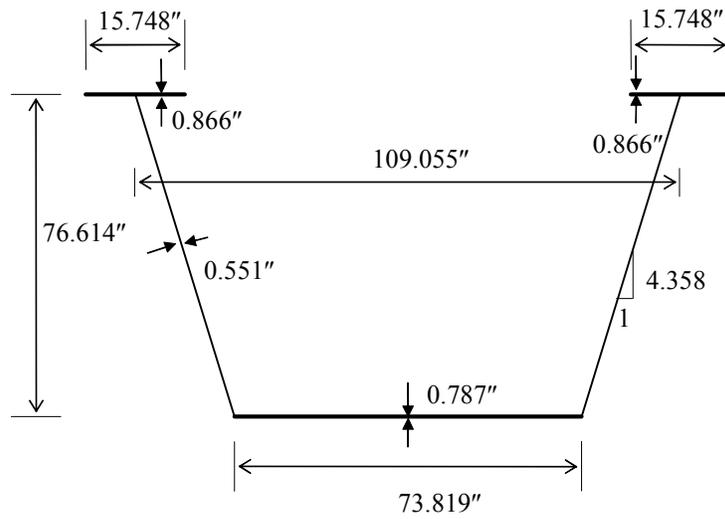
In UTrAp 2.0, bridge data is input by using the GUI (graphical user interface), and analyses are performed by a separate module invoked by the GUI. Load input is in kips, cross-section dimensions are in units of inches, and all other bridge dimensions are in feet. Results are viewed within the user interface, and may also be exported to a Microsoft Excel file. Presented here is a guide to the user interface, adapted from the original UTrAp user's guide. As features are explained, an example bridge is analyzed to illustrate the program's use.

Example Problem Definition

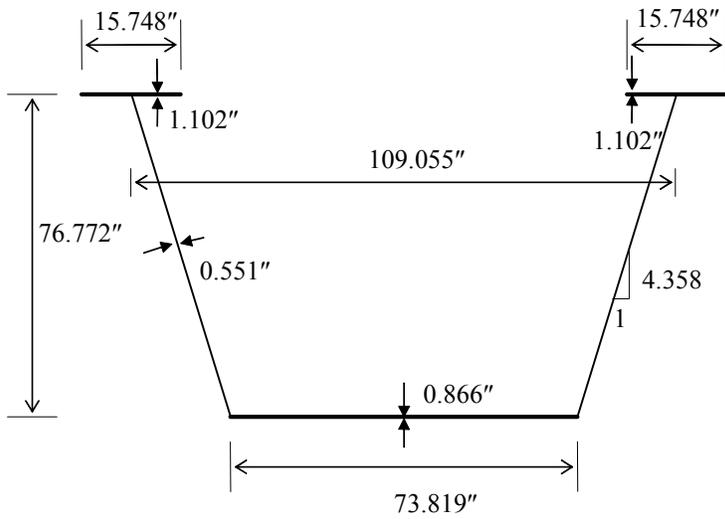
The pedestrian bridge near Marcy, New York, that collapsed when the deck pour reached midspan is chosen to illustrate the use of the UTrAp 2.0 user interface. Figure 1 shows the section dimensions of the Marcy bridge. Dimensions were originally specified in metric units and are converted to inches in the figure. The web thickness is constant over the length, while the top and bottom flanges each have a greater thickness at midspan. The bottom flange thickness changes 36 feet from each support, and the top flange thickness is increased 51 feet from each support. The span of the bridge is 170 feet.

Only internal diaphragms and struts are used in the Marcy bridge; no top lateral bracing is present. There are nine diaphragm spaces along the length of the girder, resulting in an internal diaphragm spacing of about 19 feet. Struts are placed at third points between internal diaphragms. All bracing members are L3x3x $\frac{3}{8}$ angles with a cross-sectional area of 2.11 in².

The concrete deck is 14 feet wide and 7.5 inches thick, with a dead weight including the haunches of 1.562 kips/ft. Additional dead weight from the permanent metal deck forms and the diaphragms adds 0.086 kips/ft. The total load applied to the girder is therefore 1.65 kips/ft, not including the self-weight of the girder, which is handled internally in UTrAp 2.0.



End Section Dimensions



Midspan Section Dimensions

Figure 1 Section dimensions of Marcy pedestrian bridge

UTrAp 2.0 Menus

The graphical user interface of UTrAp has nine menus. This section describes each of these menus in detail and provides information regarding how data is supplied to UTrAp for the analysis of trapezoidal box-girder bridges. In addition, specific information needed to analyze the example bridge described above is provided.

File Menu: This menu is used for data management and has four submenus. Project files are stored and retrieved using the following submenus:

New Project: Select this option to start a new bridge project. If a project is active and has been changed, the program will ask if it should be saved before creating a new project.

Open Project: This submenu opens an existing project. The UTrAp input project files have an extension of *.inp. When this submenu is invoked, an open file box will appear which is used to select the existing project file. If a project is active and has been changed, the program will prompt for save.

Save Project: This submenu saves 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 this submenu is invoked, a save file box will appear which is used to name or rename the project file. As with any program, it is recommended that projects be saved on a regular basis while using UTrAp.

Exit: This submenu is used to exit the program. If the data in the current project has been changed, a confirmation box will offer a chance to save the project upon exiting.

Tip: *Shortcut keys are available for frequently used menu options, and are shown next to the options on the menus. For example, pressing Ctrl + O is the same as clicking "File" then "Open Project".*

Geometry Menu: Choosing this menu opens the form used to input the dimensions of the bridge. Values can be typed in the boxes provided. A graphical representation of the cross section is displayed on the Geometric Properties form. After entering the required data, the user must press the Save Data button in order for the values to be stored in memory. Pressing the Save Data button stores the data and closes the form. If the user does not want to save the values, the Cancel button should be pressed. This process for saving data applies to the other data input forms as well.

Tip: Since the web depth is defined as the distance between flange centroids, it will change with changes in the flange thicknesses. The user should input an average or a weighted average value, or use the web depth at mid-span. Also, all lengths are measured along the center of the bridge.

Example Problem: A new project is formed by choosing “New Project” from the File menu. The dimensions of the Marcy bridge are input into the Geometric Properties form, as shown in Figure 2.

Geometric Properties

Project Name: Marcy Pedestrian Bridge

Number of Girders: 1 2

Geometry: Straight Curved

Length of Bridge: 170 feet

Radius of Curvature: _____

Girder Offset: _____

Cross Section Dimensions

Web Depth (between flange centroids) (in): 76.772

Width of Bottom Flange (in): 73.819

Top Width (in): 109.055

Top Flange Width (in): 15.748

Width of Deck (in): 168

Thickness of Concrete Deck (in): 7.5

Save Data Cancel

Figure 2 Geometric Properties form

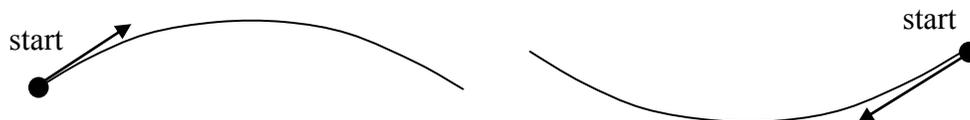


Figure 3 Definition of Bridge Start

Plate Properties Menu: This menu choice opens the Plate Properties form, which has three separate tabs for entering the web, bottom flange, and top flange properties. The length and corresponding thickness of each plate must be entered from the start to the end of the bridge. The starting point is illustrated in Figure 3 for two orientations of bridge curvature. Two buttons used to add and remove properties:

Add: This button is used to add plate 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. The number of rows in the table is increased by the number specified.

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

Example Problem: When the Plate Properties form is first displayed in a new project, one property is shown in each tab. The user can enter the total number of additional plate properties in the box next to the Add button before it is pressed. All plate properties are entered in a tabular format. The input for the bottom flange plate properties is given in Figure 4. Similar data are 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.

The screenshot shows a software window titled "Plate Properties" with three tabs: "Web Thickness", "Top Flange Thickness", and "Bottom Flange Thickness". The "Bottom Flange Thickness" tab is active. It contains a table with the following data:

	Length (ft)	Thickness (in)
1	51	0.866
2	68	1.102
3	51	0.866

Below the table is a large orange rectangular area. To the right of the table are two buttons: "Add" followed by a text input field containing "0" and the label "properties"; and "Remove" followed by a text input field containing "1" and the label "property number". At the bottom of the window are two buttons: "Save Data" and "Cancel".

Figure 4 Plate Properties form

Bracing Menu: This selection opens the Bracing Properties form, shown in Figures 7, 8, and 9. The form has four separate tabs for inputting the internal brace, external brace, top lateral brace, and web stiffener properties. The program currently offers only one type of both internal and external braces, which can be seen by clicking on the “Show Brace Types” button at the bottom of each tab. The location and member cross-sectional area are required for internal and external braces. The start location, end location, type, and area are required for the top lateral braces. In UTrAp 2.0, transverse web stiffeners can also be added to the model by specifying their location, width, and thickness. The following buttons are provided for adding and removing braces and stiffeners:

Add: This button inserts braces or stiffeners. The number of rows in the table is increased by the number specified.

Equally Space: This button adds a specified number of braces or stiffeners at equally-spaced intervals between the two location values entered. The location entered in the first box must be smaller than the location in the second box.

Match Internal Brace Locations: This button copies the internal brace locations into the web stiffener tab. Because internal braces are almost always accompanied by web stiffeners, this button simplifies the input process.

Remove: This button removes the brace or stiffener corresponding to the number entered.

Remove All Braces/Stiffeners: This button removes all braces or stiffeners in the tab.

Type: This button is displayed in the internal brace and external brace tabs. Because the current version of UTrAp offers only one type of both internal and external braces, this button is disabled and the braces are automatically set to type 1. The internal and external brace configurations can be seen by using the Show Internal/External Brace Types buttons.

Area: This button is used to assign the same cross-sectional area to all brace members. If all braces do not have the same cross-sectional area, the values for each brace can be entered directly into the table.

Width: This button appears only in the web stiffener tab, and is used to apply a constant width to all web stiffeners. Widths of individual stiffeners can also be specified directly in the table.

Thickness: Similar to the Width button, this button applies a constant thickness to all web stiffeners. Again, individual thicknesses can be entered directly.

Show Internal/External/Top Lateral Brace Types: These buttons are used to display the types of braces that can be specified in the program. When this button is pressed, a form that shows the geometry and types of braces is displayed on the screen. Figure 5 shows the types of internal and external braces supported by the current version of the program. The two types of top lateral braces are shown in Figure 6.

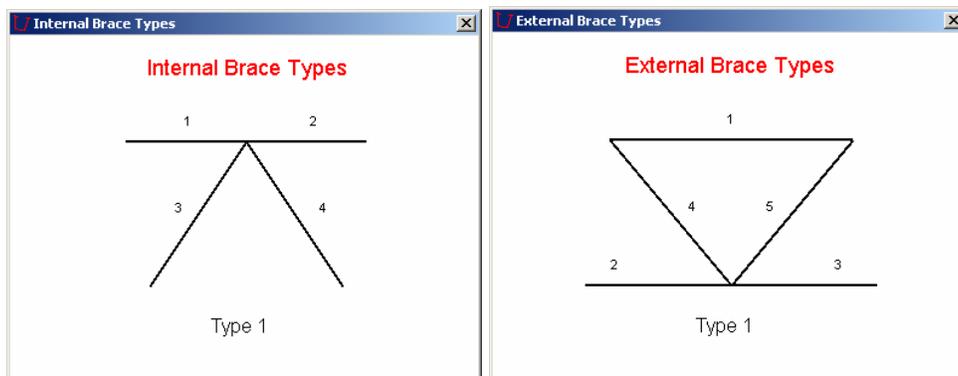


Figure 5 Internal and external brace types

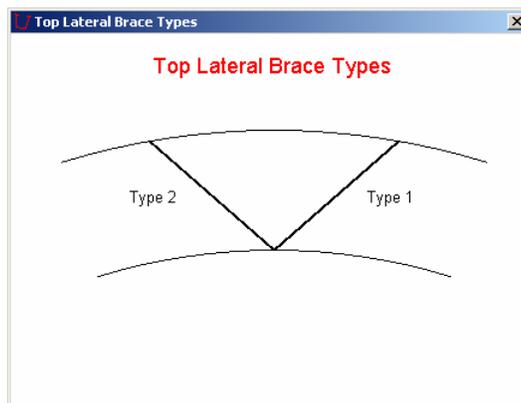


Figure 6 Top lateral brace types

All Type 1: This button, displayed only in the Top Lateral Braces tab, assigns Type 1 to all top lateral braces.

All Type 2: Similar to the previous button, this button assigns Type 2 to all top lateral braces.

Alternating Starting with Type 1: This button, displayed only in the Top Lateral Braces tab, assigns alternating types to consecutive braces. The first brace will be Type 1, the second Type 2, and so on.

Alternating Starting with Type 2: This button accomplishes the opposite of the previous button, starting with Type 2 braces instead of Type 1.

An X-type top lateral system can be handled by the program simply by specifying both Type 1 and Type 2 braces with the same start and end points. For each X-brace, two data lines with the same start and end location would be needed, one with a Type 1 brace and the other with a Type 2 brace. Struts can also be defined within the program. As shown in Figure 5, an internal brace includes a strut (members 1 and 2 in the figure). If a strut exists at location where an internal brace is not present, the user can simply specify the strut as a top lateral brace with identical start and end locations. For struts, either Type 1 or Type 2 braces can be specified. Placing a strut at the same location as an internal crossframe will add both areas, not replace one of the members.

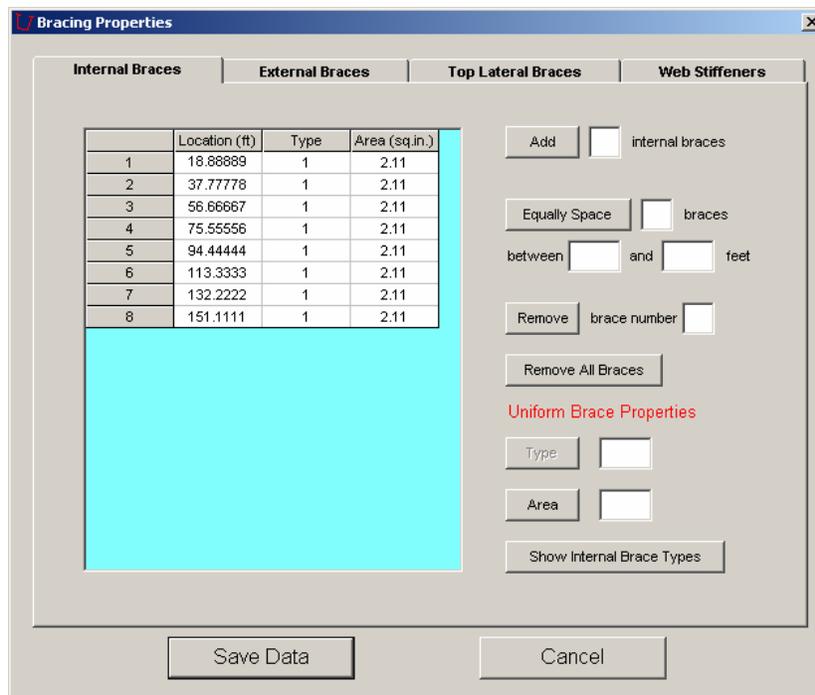


Figure 7 Internal Braces tab

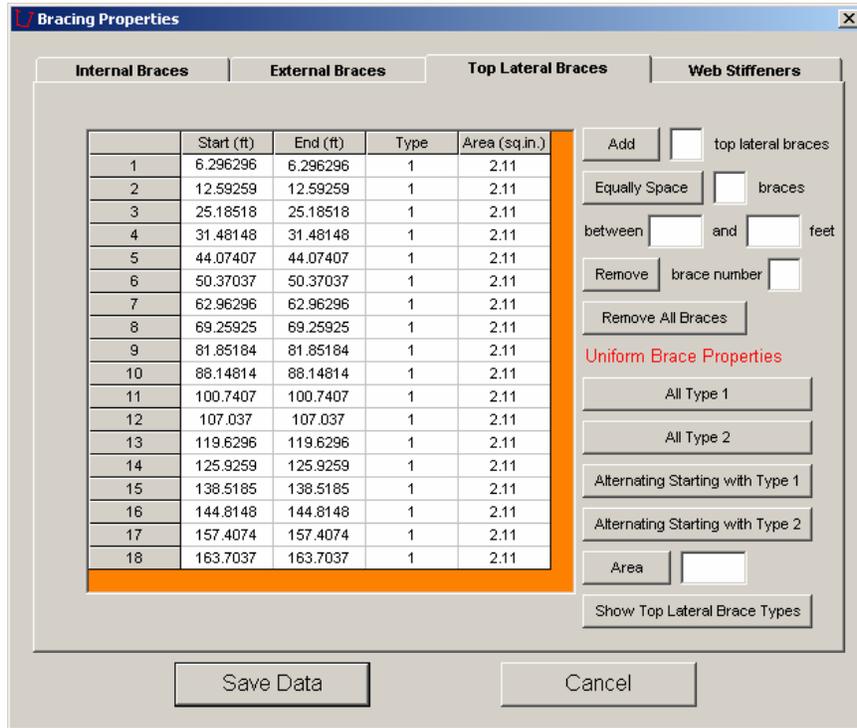


Figure 8 Top Lateral Braces tab

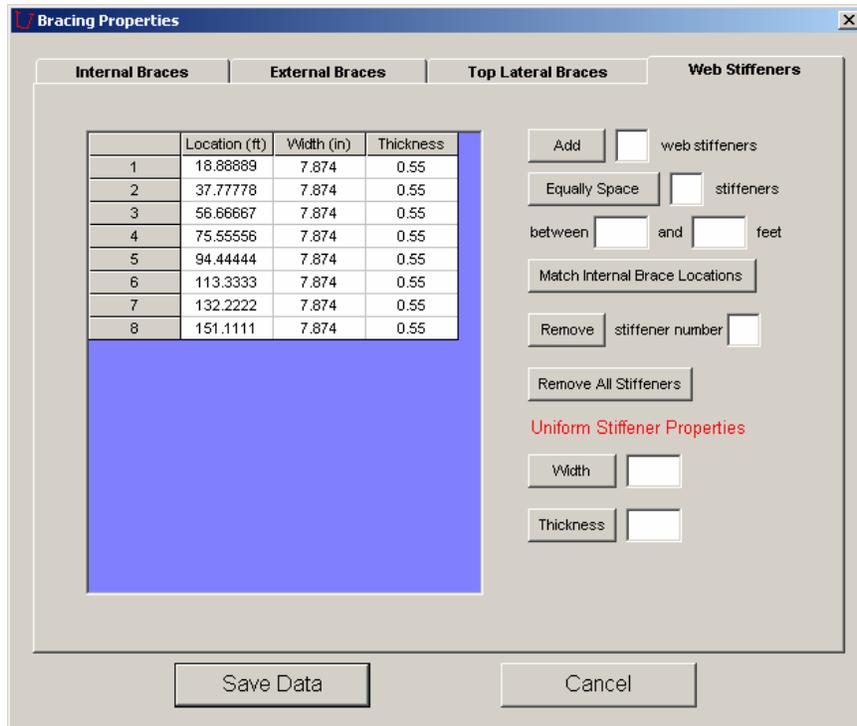


Figure 9 Web Stiffeners tab

Example Problem: Eight internal braces are added using the Equally Space button, followed by eight pairs of web stiffeners added using the Match Internal Brace Locations button. The eighteen strut start locations are entered by first using the Equally Space button with 26 braces and then removing every third brace where internal cross frames are located. The start locations are then copied into the end locations. Areas and dimensions are entered using the Uniform Properties buttons. Figures 7 through 9 show the relevant tabs of the Bracing Properties form with the data for the example problem entered.

Support Menu: This menu is used to input support locations. The program assumes the first support entered is pinned in all directions and the others are restrained against vertical and lateral movement only, allowing them to move longitudinally. The number of rows in the table is controlled by the Add and Remove buttons:

Add: This button is used to add the specified number of supports. The number of rows in the table is increased by the number entered.

Remove: This button is used to remove the specified support number.

Example Problem: Two support properties are automatically shown when opening the Supports form in a new project. The bridge is a simple span, so 0 feet and 170 feet are entered for the support locations. The support at 0 feet will be pinned, although the opposite end could be pinned by simply reversing the entries. Figure 10 shows the Support Locations form with the entered data.

Support No.	Location (ft)
1	0
2	170

Figure 10 Support Locations form

Stud Menu: The spacing of the studs along the length of the bridge and the number of studs per flange are input in this menu. The number of rows in the table is controlled by two Add and Remove buttons:

Add: This button is used to add the specified number of properties.

Remove: This button is used to remove the specified property.

Tip: To make the deck non-composite, enter any generic stud spacing data, then specify stud stiffnesses of zero in the *Pour Sequence* form.

Example Problem: Stud spacing is constant along the length of the Marcy bridge, so only one property is necessary. A stud spacing of 24 inches is entered, and two studs per flange are specified. Figure 11 shows the Stud Properties form with the data entered.

	Length (ft)	Spacing (in)	No per Flange
1	170	24	2

Figure 11 Stud Properties form

Pour Sequence Menu: This menu is used to input the pour sequence parameters in tabular form. The concrete deck can be divided into segments corresponding to each separate pour. Lengths of the deck segments are the same for all analyses. Independent analyses can be run for different loadings and deck segment stiffnesses. For each analysis, properties of the deck segments and loading on the segments are required input. Properties for a deck segment include the concrete stiffness and the stud stiffness. During the pour sequence both the concrete and stud stiffness are zero when the concrete is first poured and the section is non-composite. As the concrete sets, both the concrete

and the studs gain stiffness. It is conservative to assume non-composite behavior throughout the pour sequence. However, substantial stiffness can develop after as little as six hours, significantly reducing the diagonal brace forces (see Topkaya, Yura, and Williamson 2004).

A new feature in UTrAp 2.0 is the ability to include the self-weight of the girder. This is accomplished by checking the "Include Self-Weight" box. Note that only the dead weight of the steel girder itself is added. The load in each segment should account for the weight from the concrete, bracing members, stiffeners, deck forms, and any other loads. The Pour Sequence table is controlled by the following four buttons:

Add Analysis Case: This button adds a new analysis case to the table. Three new columns are added to the right side of the table and are used to specify the concrete stiffness, the stud stiffness, and the load acting on the deck. Loads are treated independently in each analysis case, allowing the results to include both incremental and total values. Thus, in a new analysis case, only the additional load added during that analysis should be specified; the loads from previous analysis cases are still present.

Remove Analysis Case: This button removes the specified analysis case. The three columns related to that analysis are removed from the table.

Add Deck Property: As mentioned, the concrete deck can be divided into different pours, each with unique properties. This button adds a new deck property row to the bottom of the table. The sum of the lengths of the deck properties must equal the total bridge length.

Remove Deck Property: This button removes the specified row of deck properties.

Example Problem: In this problem, the deck is divided into two segments: a 68 ft segment that represents the length of the wet concrete when lateral buckling is expected, and the remaining 102 ft segment with no concrete. The deck segments are added to the table by making use of the Add Deck Property button. The two analysis cases to be performed are added to the table by using the Add Analysis Case button. In the first analysis, the dead weight from the forms and diaphragms of 0.086 kips/ft is added to both segments in addition to the self-weight of the girder. For the second analysis, the weight of the concrete slab and haunches of 1.562 kips/ft is applied only to the 68 ft deck segment. Figure 12 shows the Pour Sequence form with the Marcy bridge data.

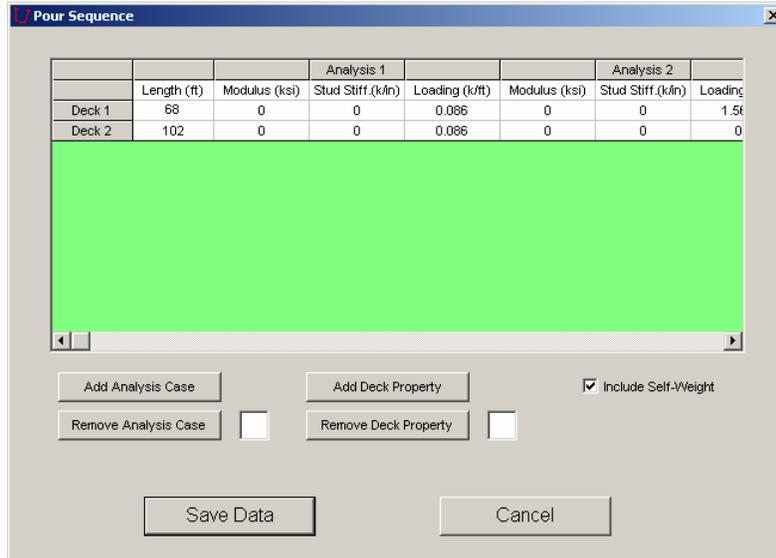


Figure 12 Pour Sequence form



Figure 13 UTrAp 2.0 main window

Before running an analysis, the user can verify the bridge information by looking at the main window, which shows three figures representing the bridge properties. At the top, the plate thicknesses along the length are shown in an elevation view. The middle figure shows the deck lengths. The bottom figure is a plan view of the bridge, showing internal braces, external braces, and top lateral braces. Figure 13 shows the main window after all the information for the Marcy bridge has been provided.

Analysis Menu: The two submenus in this menu call the finite element analysis portion of UTrAp 2.0. When the user invokes either submenu, the program verifies that the bridge properties 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, the analysis module is called to perform the finite element analyses. The analysis module runs in the DOS environment. A user can monitor the progress of the analysis by observing the messages displayed in the DOS window, which closes automatically when the analysis is completed. The two options on the Analysis menu are explained below.

Linear Analysis Submenu: This submenu launches the linear analysis included in the original version of UTrAp. No nonlinear behavior is considered. Figure 14 shows a representative linear analysis screen for the Marcy bridge.

Buckling Analysis Submenu: This submenu first opens a dialog box asking for the number of buckling modes desired. UTrAp 2.0 can provide between one and five buckling modes. The default value is two, which will allow the user to verify that the first two modes are well-spaced. If the first two modes are very similar, additional buckling modes may be necessary to determine the buckling behavior. The user must bear in mind that the first mathematical value may not represent the actual failure mode. For example, the first mode may show local buckling of the webs, while the second mode shows global lateral buckling of the bridge, which may be more indicative of the true failure mode. Post-buckling strength of the webs is utilized in most design specifications, so local web buckling is usually not significant.

Clicking "OK" starts a buckling analysis of the bridge, which will automatically perform the required linear analysis. For each analysis case, the program will find the elastic critical buckling load as a multiple of all applied loads. Figure 15 shows a representative buckling analysis screen for the Marcy bridge.

```

C:\utrap2.exe
UTrap 2.0 FE Model has :
    1 straight girder
    9234 nodes
    2210 shell elements
    8 internal brace elements
    0 external brace elements
    18 top lateral elements
    16 web stiffener elements
    1026 stud elements
    2 support elements
    3280 total elements
Connectivity Process Completed

Start of Analysis Number 1
Assembly of Shell Elements Completed in 4.4 seconds
Assembly of Internal Braces Completed
Assembly of Top Laterals Completed
Assembly of Web Stiffeners Completed
Assembly of Supports Completed
Assembly of Stud Elements Completed
Modification for Support Conditions Completed
Formation of Load Vector Completed

Start the Solution
Solver Step 1 of 6 - Creation Completed
Solver Step 2 of 6 - Definition Completed
Solver Step 3 of 6 - Reordering Completed
Solver Step 4 of 6 - Factorization Completed
Solver Step 5 of 6 - Solution Completed
Solver Step 6 of 6 - Deletion Completed
Solution Completed in 33.8 seconds

Post-processing of Deflections Completed
Post-processing of Cross Sectional Rotations Completed
Post-processing of Cross Sectional Forces Completed in 3.9 seconds
Post-processing of Support Reactions Completed
Post-processing of Internal Brace Forces Completed
Post-processing of Top Lateral Forces Completed

```

Figure 14 Linear Analysis screen

```

C:\utrap2.exe
Assembly of Shell Elements Completed in 4.1 seconds
Assembly of Internal Braces Completed
Assembly of Top Laterals Completed
Assembly of Web Stiffeners Completed
Assembly of Supports Completed
Assembly of Stud Elements Completed
Modification for Support Conditions Completed
Formation of Load Vector Completed

Start the Solution
Solver Step 1 of 6 - Creation Completed
Solver Step 2 of 6 - Definition Completed
Solver Step 3 of 6 - Reordering Completed
Solver Step 4 of 6 - Factorization Completed
Solver Step 5 of 6 - Solution Completed
Solver Step 6 of 6 - Deletion Completed
Solution Completed in 33.7 seconds

Start of Buckling Check for Analysis 1
Assembly of Shell Elements Completed in 4.2 seconds
Assembly of Internal Braces Completed
Assembly of Top Laterals Completed

Start the Eigenproblem Solution
.....
Eigenproblem Solution Completed in 64.5 seconds

```

Figure 15 Buckling Analysis screen

Results Menus

The following submenus are 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.

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 in this section. Deflection values are the vertical deflections of the center of the bottom flange. Rotation values correspond to the rotation of the bottom flange. For twin girder systems, only the deflections and rotations of the outer girder are reported. Both tabulated and graphical output can be displayed. In the tables, deflection and rotation values at two-foot increments along the length of the bridge are shown. The user can request incremental deflection and rotation values for each analysis or cumulative values after each case. The Deflections form and Cross-Sectional Rotations form each have four buttons to control the display of results:

Tabulate Incremental Deflections/Rotations: This button tabulates the incremental deflections or rotations every two feet along the bridge. Values are presented for all analysis cases and are not summed. Figure 16 shows a sample of the deflections for the Marcy bridge.

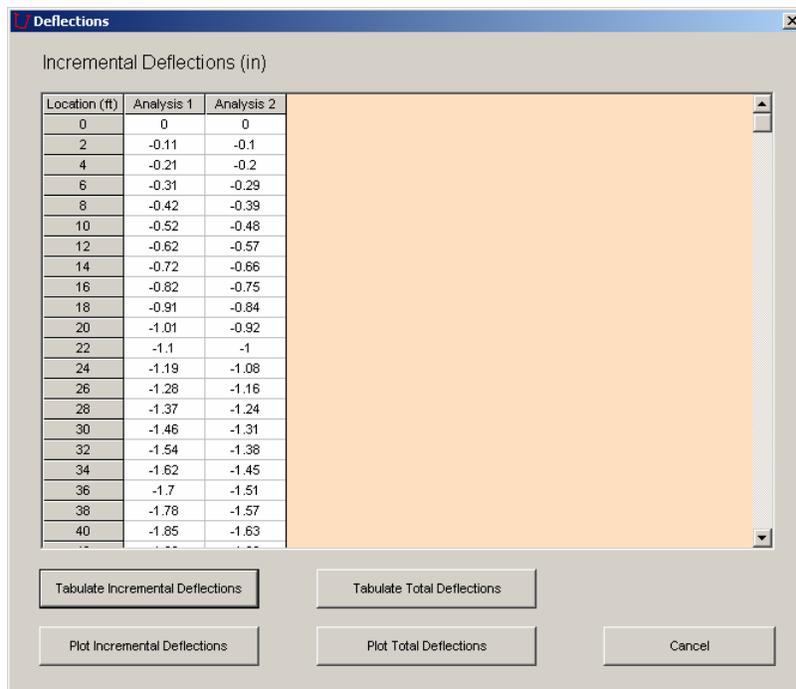


Figure 16 Table of incremental deflections

Tabulate Total Deflections: This button displays the total deflections or rotations every two feet along the bridge in tabular form. Cumulative deflection and rotation values are presented for each analysis. For example, the values in column two would be the summation of deflections or rotations resulting from the first and second analyses.

Plot Incremental Deflections/Rotations: This button displays the incremental deflections or rotations for all analyses on one graph. Figure 17 shows the incremental deflection diagram for the example bridge.

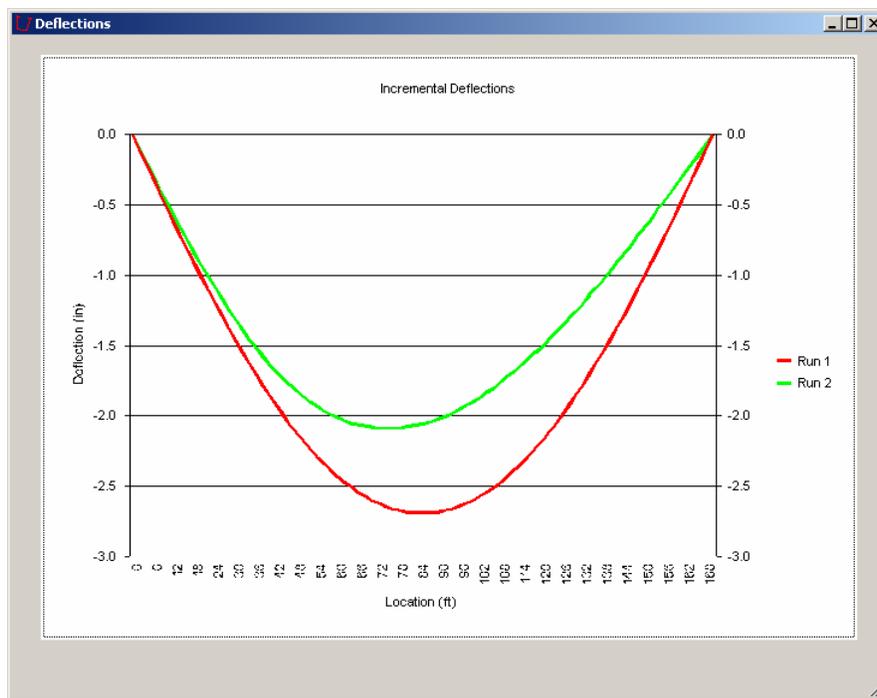


Figure 17 Diagram of incremental deflections

Plot Total Deflections/Rotations: This button displays the cumulative deflection or rotation diagram. Each curve represents the summation of the deflections or rotations from all previous analysis cases.

Cross-Sectional Forces Submenu: This submenu displays the values of the shear, moment, and torsion acting on the bridge at specified cross-sections. Data can be displayed in both tabular and graphical form. Tabulated output consists of shear, moment, and torsion values every two feet along the bridge length. The same values can

be displayed graphically. Radio buttons are used to select between incremental and cumulative values. For twin-girder bridges, quantities are summed for the two girders. The Cross-Sectional Forces form has the following six buttons to control the display of results:

Tabulate Shear: This button tabulates the shear every two feet along the bridge length. Incremental or cumulative values are presented for all analysis cases. Figure 18 shows the Cross-Sectional Forces form with the results for the Marcy bridge.

Tabulate Moment: This button tabulates the internal bending moment at locations every two feet along the bridge length. Incremental or total values are presented for all analysis cases.

Tabulate Torque: This button tabulates the torque every two feet along the bridge length for all analysis cases.

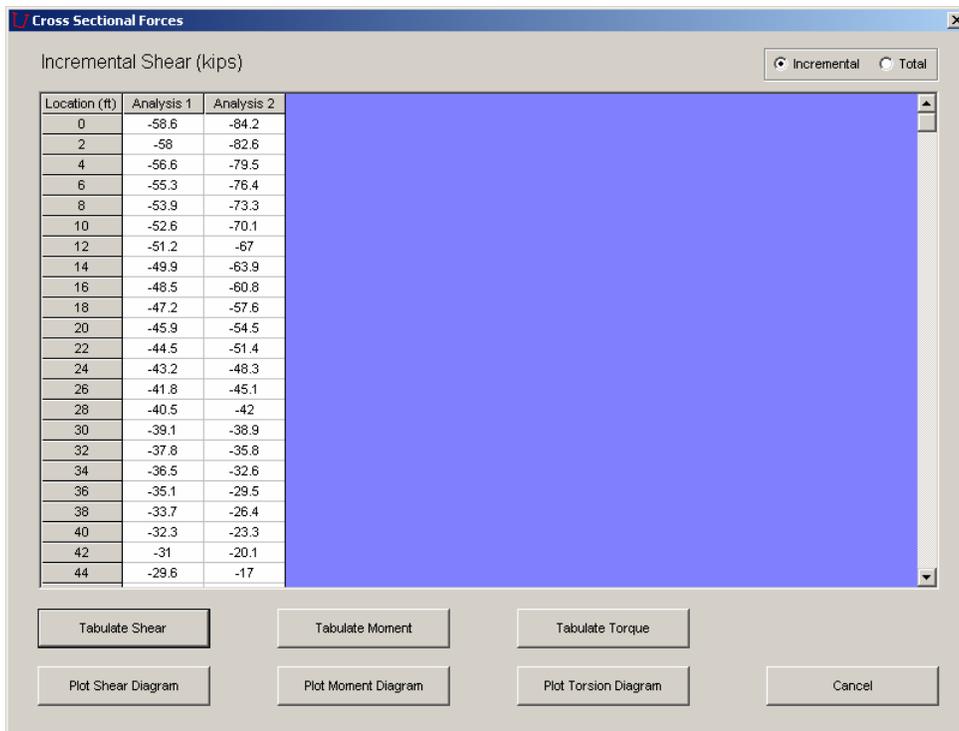


Figure 18 Table of incremental shear values

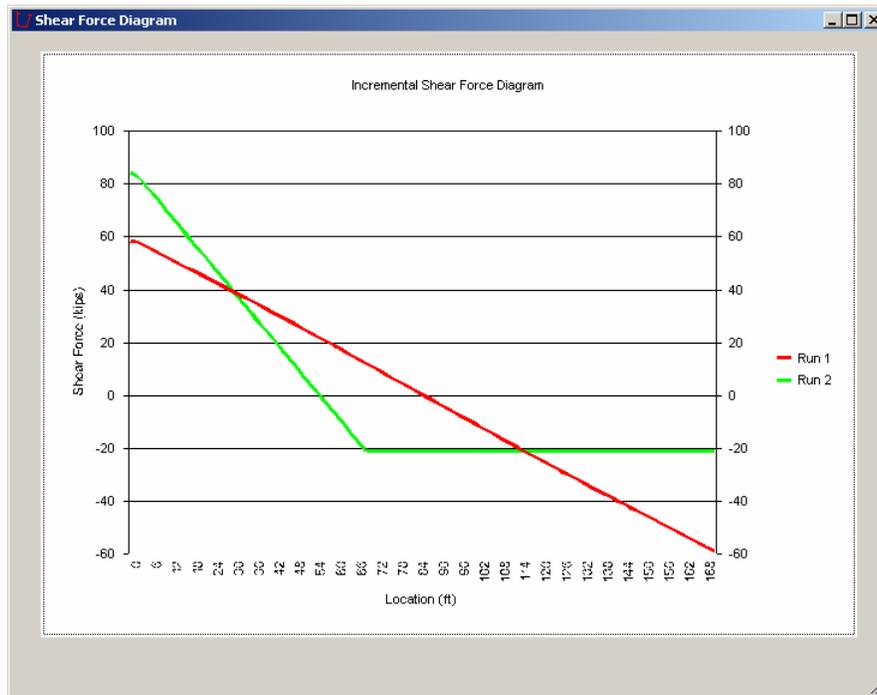


Figure 19 Diagram of incremental shear values

Plot Shear Diagram: This button plots incremental or total shear values for all analyses on one graph. Figure 19 shows the shear diagram for the Marcy bridge.

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

Plot Torsion Diagram: This button shows the incremental or cumulative torsion values for all analyses on one graph.

Stresses Submenu: This submenu is used to visualize the cross-sectional stresses. The analysis module calculates normal and shear stresses at locations on the cross section denoted “section points” every two feet along the bridge length. There are 26 and 52 section points on the cross section for the single and dual girder systems, respectively, corresponding to the central points of the shell finite elements. 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 cumulative format. Radio buttons are placed on the form to select between shear and normal stress and between incremental and cumulative 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. Additionally, this form can be used to display stresses at all section points at a specified cross section. The Stresses form has three buttons that interact with three scroll boxes:

Tabulate Stresses: This button tabulates the stress values along the bridge length at 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 boxes. In addition, total or incremental values can be displayed. Figure 20 shows a stress output table.

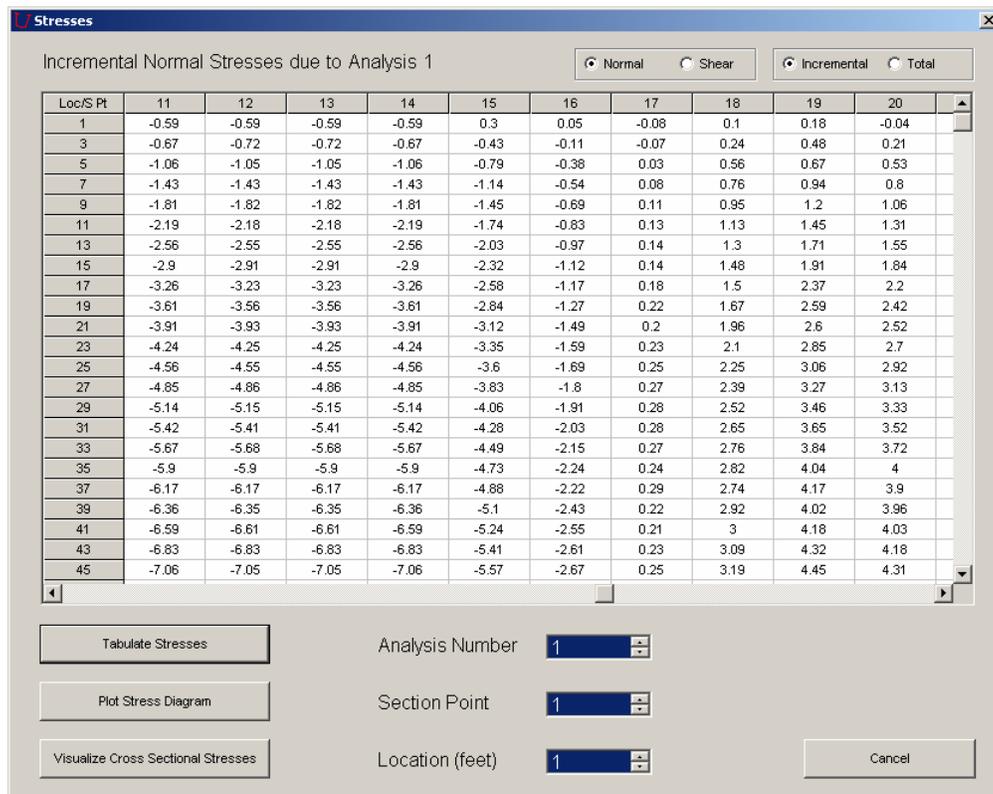


Figure 20 Table of incremental normal stress values

Plot Stress Diagram: This button is used to display the variation of normal or shear stress along the bridge length at a specified section point. The analysis case and section point must be selected using the scroll boxes. Figure 21 shows a plot of incremental normal stress along the bridge length for analysis number 1 at section point 11, located in the left top flange. The discontinuities in the graph are due to changes in the top flange thickness.

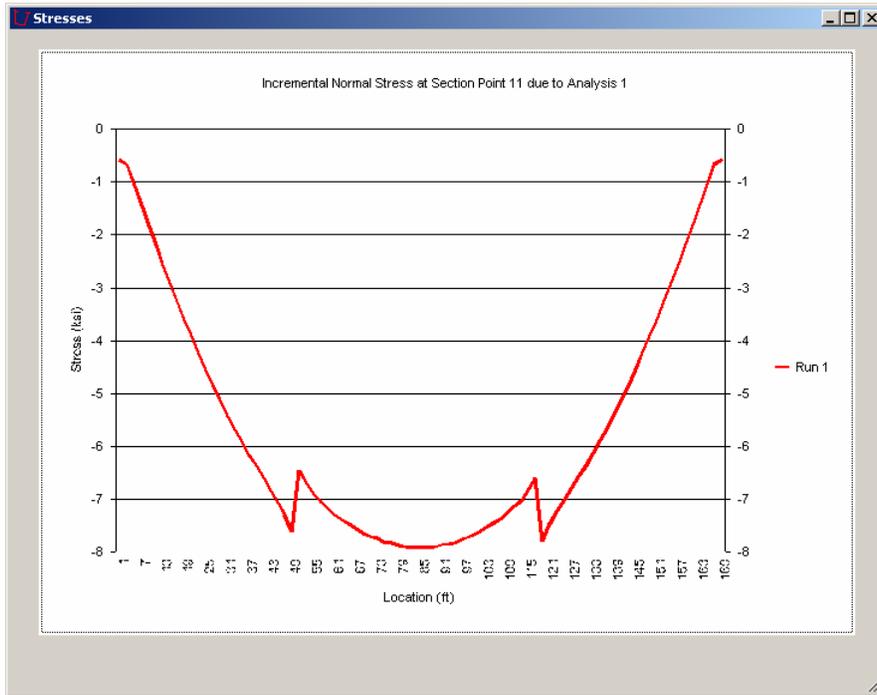


Figure 21 Diagram of incremental normal stresses

Visualize Cross-Sectional Stresses: This button is used to display the stresses at all section points on a specified cross section for an analysis case and location selected using the scroll boxes. Figure 22 shows the incremental normal stress distribution due to the second analysis in a cross section that is at midspan, 85 feet from the start end. Section points and stress values are shown on the diagram. The arrow in the figure points toward the center of radius for the bridge, which for a straight bridge is at a distance of infinity.

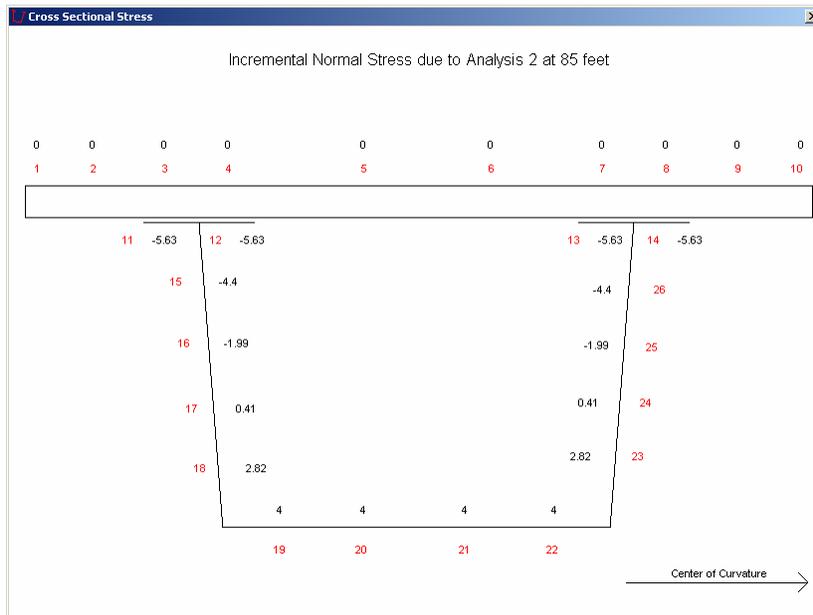


Figure 22 Visualization of incremental normal stresses on cross-section

Support Reactions Submenu: This submenu, new to UTrAp 2.0, is used to display the vertical reactions at the supports, with positive denoting an upward reaction. Radio buttons are used to select incremental or total forces. For twin-girder bridges, both inner and outer girder reactions are displayed. Negative reaction forces denote uplift at that support, a condition which must be considered in bearing design. Figure 23 shows the support reactions for the Marcy bridge.

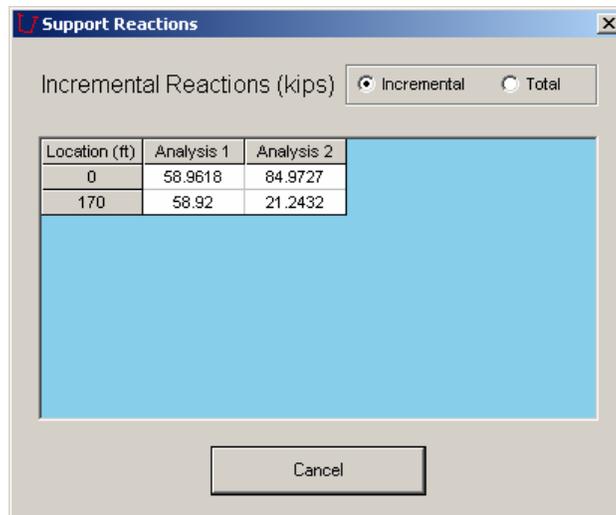


Figure 23 Support Reactions form

Internal Brace Forces/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 displayed on a bar graph. Incremental axial forces due to each analysis or cumulative axial forces after each analysis can be displayed. Four buttons and a scroll box are used to control the output in these forms:

Tabulate Incremental Forces: This button tabulates the forces in brace members due to each analysis case. Results for a certain member of the brace, selected using the scroll box, are displayed. Positive values represent tension forces in the brace members, and negative values represent compression forces. This convention is used throughout the program. The internal and external brace configurations and the corresponding member numbers were presented previously in Figure 5. Figure 24 shows the Internal Brace Forces form with the table of axial force values for member 2 of all internal braces.

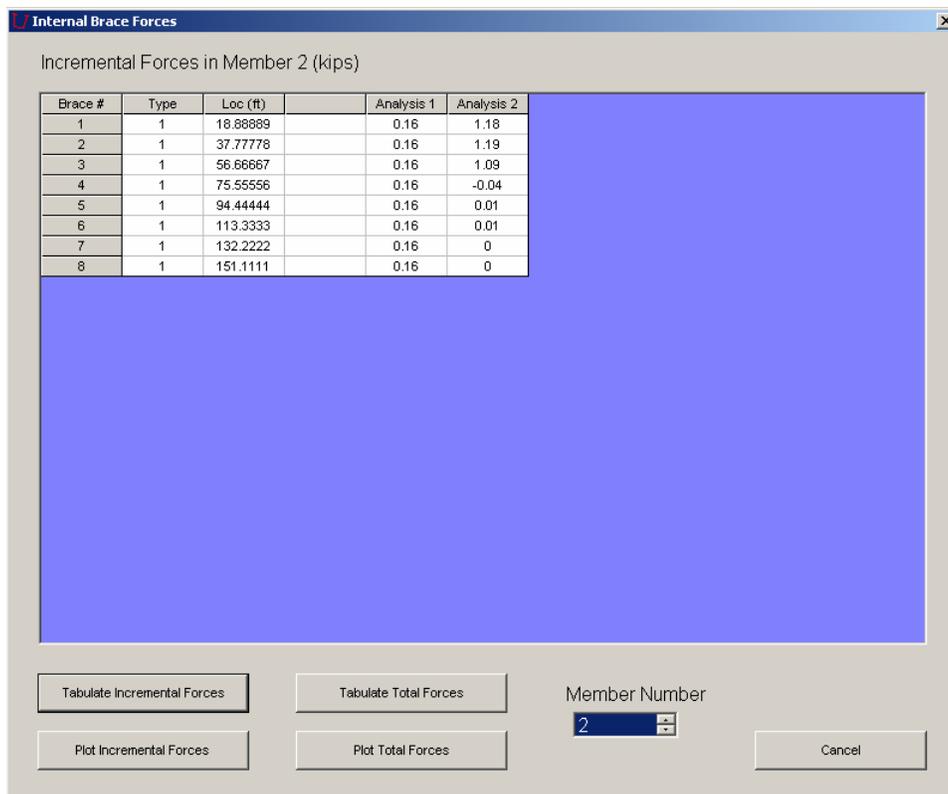


Figure 24 Table of incremental axial forces in internal braces

Tabulate Total Forces: Similar to the Tabulate Incremental Forces button, this button tabulates the cumulative forces after each analysis.

Plot Incremental Forces: This button displays a bar chart of axial forces in the braces for the member number selected using the scroll box. Figure 25 shows a bar chart of incremental axial forces in member two of the internal braces.

Plot Total Forces: This button, similar to the Plot Incremental Forces button, is used to display the cumulative forces after each analysis.

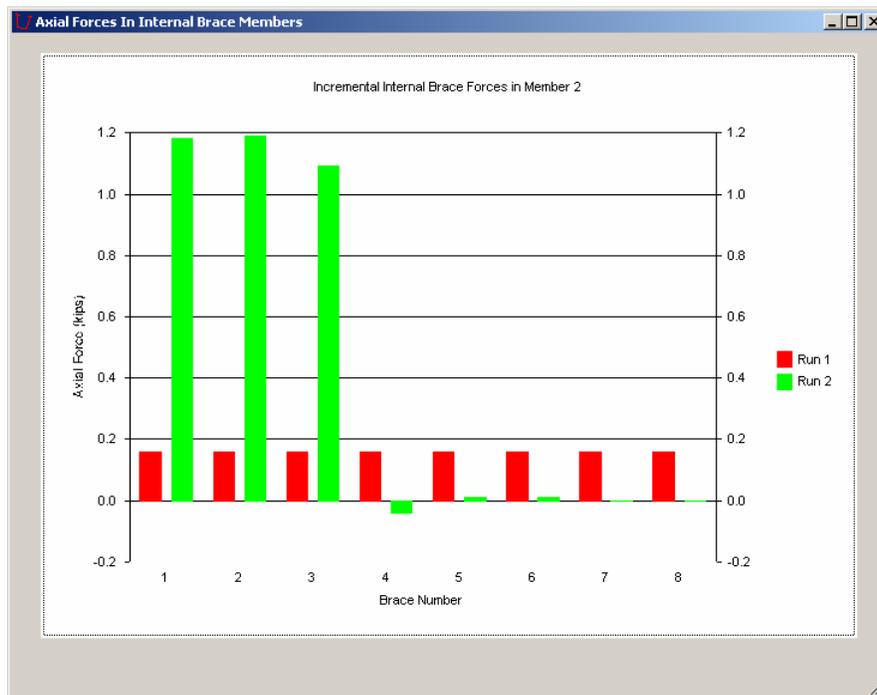


Figure 25 Chart of incremental axial forces in internal braces

Top Lateral Brace Forces Submenu: This submenu displays the forces in the top lateral braces. Values can either be tabulated or visualized as a bar graph. Incremental forces due to each analysis or cumulative forces after each analysis case can be displayed. The four buttons explained below are used to control the output.

Tabulate Incremental Forces: This button tabulates the forces in the top lateral members due to each analysis case.

Tabulate Total Forces: This button tabulates the cumulative forces in the top lateral members after each analysis case.

Plot Incremental Forces: This button plots a bar chart showing the incremental top lateral brace forces due to each analysis.

Plot Total Forces: This button shows a bar chart of the cumulative top lateral brace forces after each analysis.

Export to Excel Submenu: In the original version of UTrAp, the user interface was the only option for viewing the results of an analysis, unless the user wished to interpret the text files containing the results. UTrAp 2.0 allows the user to export the results to Excel. When the Export to Excel submenu is selected, a save file dialog will appear, allowing the user to name the Excel workbook. After clicking "Save", the program creates an Excel file containing the results, with deflections, rotations, etc. on separate worksheets.

The buckled shape results are not exported due to the large amount of data, but if the user is interested, the x- and y-coordinates of the deflected shape along the bridge are contained in the file csc0987.txt, found in the program folder.

Note: The "Export to Excel" feature has only been tested for Windows XP and Excel 2002 and newer. The feature may not work with other operating systems or versions of Excel.

Analysis Summary Submenu: This option displays the maximum values of significant quantities for each analysis, including maximum deflections, moments, shear forces, and axial stresses. The Analysis Summary form is shown in Figure 26. The data contained on this form can be output to Excel format by using the Export to Excel function.

Analysis Summary

Summary of Maximum Quantities

Quantity	Analysis 1		Analysis 2	
	Value	Location (ft)	Value	Location (ft)
Incremental Upward Deflection (in)	0	0	0	0
Incremental Downward Deflection (in)	-2.69	82	-2.09	72
Total Upward Deflection (in)	0	0	0	0
Total Downward Deflection (in)	-2.69	82	-4.76	78
Incremental Rotation (rad)	0	0	0	0
Total Rotation (rad)	0	0	0	0
Incremental Shear (kips)	58.6	0	84.2	0
Incremental Positive Moment (kip-ft)	2544	84	2311.1	54
Incremental Negative Moment (kip-ft)	0	0	0	0
Incremental Torque (kip-ft)	0	0	0	0
Total Shear (kips)	58.6	0	142.8	0
Total Positive Moment (kip-ft)	2544	84	4626	64
Total Negative Moment (kip-ft)	0	0	0	0
Total Torque (kip-ft)	0	0	0	0
Incremental Positive Normal Stress (ksi)	5.81	75	5.16	49
Incremental Negative Normal Stress (ksi)	-7.94	85	-8.3	49
Incremental Shear Stress (ksi)	0.89	169	1.23	3
Total Positive Normal Stress (ksi)	5.81	75	10.27	75
Total Negative Normal Stress (ksi)	-7.94	85	-15.93	49
Total Shear Stress (ksi)	0.89	169	2.09	3
		Brace #		Brace #
Incremental Positive Top Lateral Axial Force (kips)	0.23	7	1.18	3
Incremental Negative Top Lateral Axial Force (kips)	0	0	-0.01	9

Figure 26 Analysis Summary form

Buckling Analysis Output

The preceding discussion of UTrAp 2.0 output has been limited to linear analysis, which is the same as in the original version of UTrAp. The focus in this section is on the output given by the revised version of the program. There are two types of output given by UTrAp 2.0 buckling analyses: the results just prior to buckling and the buckled shape.

Results Just Prior to Buckling

The response of the bridge just before it reaches the critical load is found by simply multiplying the displacement vector from the original applied load by the first eigenvalue. Because the analysis is linear and elastic, the eigenvalue is simply a scale factor increasing each response quantity. For example, if the midspan vertical deflection of a girder is one inch and the eigenvalue is 2.0, the results from the buckling analysis would indicate a midspan deflection of two inches. These results are equivalent to those from an elastic analysis with an applied load equal to the original load multiplied by the first eigenvalue.

This information is useful for two purposes: determining whether the bridge will buckle elastically, and determining the forces and displacements just prior to buckling.

The UTrAp buckling analysis is valid only if the stresses at all points in the girder are below the yield stress of the material. The analysis is not valid if the stresses exceed yield, because linear-elastic material behavior is assumed. If the bridge remains elastic, the forces can be used to check buckling of individual bracing members. The forces in the brace members just prior to buckling are included in the output, and these quantities must be checked to ensure that they are below the buckling loads of the respective individual members.

As the response quantities just prior to buckling are obtained linearly, they are identical to the quantities that would be obtained from a linear analysis using a load that is scaled by the critical eigenvalue. The response quantities can therefore be output using the same Results menu as for elastic analysis, with one critical difference: the incremental results cannot be summed to give the total results because a buckling analysis cannot rely on elastic superposition of successive analyses. The load vector in the buckling analysis is cumulative: each new loading is added to the loadings from previous analyses. The buckling analyses are independent, each being done for all loads on the structure at that time. Each response quantity displayed is therefore equivalent to the cumulative linear response for that analysis multiplied by the eigenvalue for that analysis, and represents the maximum value possible prior to buckling. For this reason, the display of incremental results is disabled for buckling analyses.

The results just prior to buckling may be slightly counter-intuitive upon initial inspection. For the Marcy bridge example, the first analysis case involves only the dead weight of the girder, forms, and diaphragms. The second analysis adds the weight of the wet concrete over the first 68 feet of the span, leading to greater forces and deflections throughout the girder. However, the results just prior to buckling are just the opposite. The first analysis has an eigenvalue of 1.924, which means the response quantities are increased by nearly a factor of two over the linear elastic analysis results. The second analysis case, chosen to result in imminent failure, has an eigenvalue of 1.00, which means the results just prior to buckling are equal to the results from the linear case. Thus, in order for the girder to become unstable for the first analysis, almost twice the load of the pattern specified for this analysis would be required. For the second case, the applied loads increase, and the load factor or eigenvalue associated with buckling is lower than the first case. Though the eigenvalue is lower, the total applied loads are greater than in Analysis 1.

The two results are illustrated in Figure 27. The deflections just prior to buckling in Analysis 1 are greater than those in Analysis 2, implying that the second analysis is the more critical loading case. The bridge under the second loading case will buckle at lower levels of stress than it will under the first loading case. As explained above, the response quantities from each analysis must be individually examined to ensure that the bridge remains elastic.

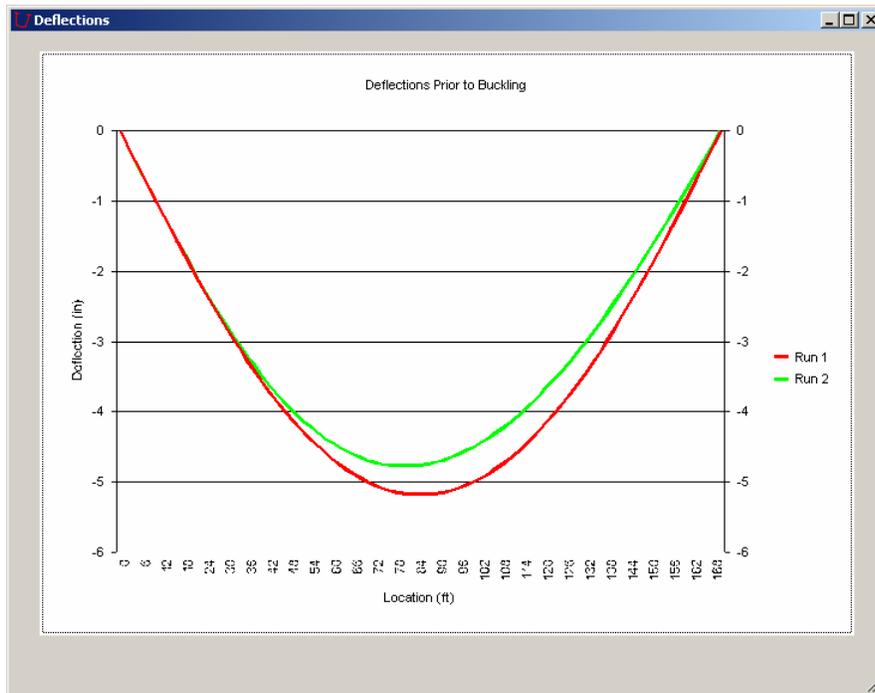


Figure 27 Deflections just prior to buckling

Buckled Shape

The buckled shape is also output by UTrAp 2.0 and can be used to determine the type of buckling failure exhibited by the bridge. The displacements and rotations shown in the buckled shape are obtained from the eigenvector corresponding to a particular eigenvalue, and thus are not related to the linear deformations of the girder. Because the displacements are based on the eigenvector, which has only relative values, no units can be assigned to the deformations. Some experience with the program and knowledge of stability is necessary to be able to interpret the failure modes predicted by the buckled shape.

Buckled Shape Submenu: Within the graphical user interface, the buckled shapes for a given analysis can be seen by using the Buckled Shape submenu. Buckled shapes are shown by two-dimensional cross-sections every two feet along the length of the bridge. The user specifies the analysis number, buckling mode, and location along the bridge by using the scroll boxes on the form. The Buckled Shape form in Figure 28 shows a typical second-mode buckled shape of the Marcy bridge. It can easily be seen that the bridge is experiencing global lateral torsional buckling, which agrees with the observations from the bridge collapse.

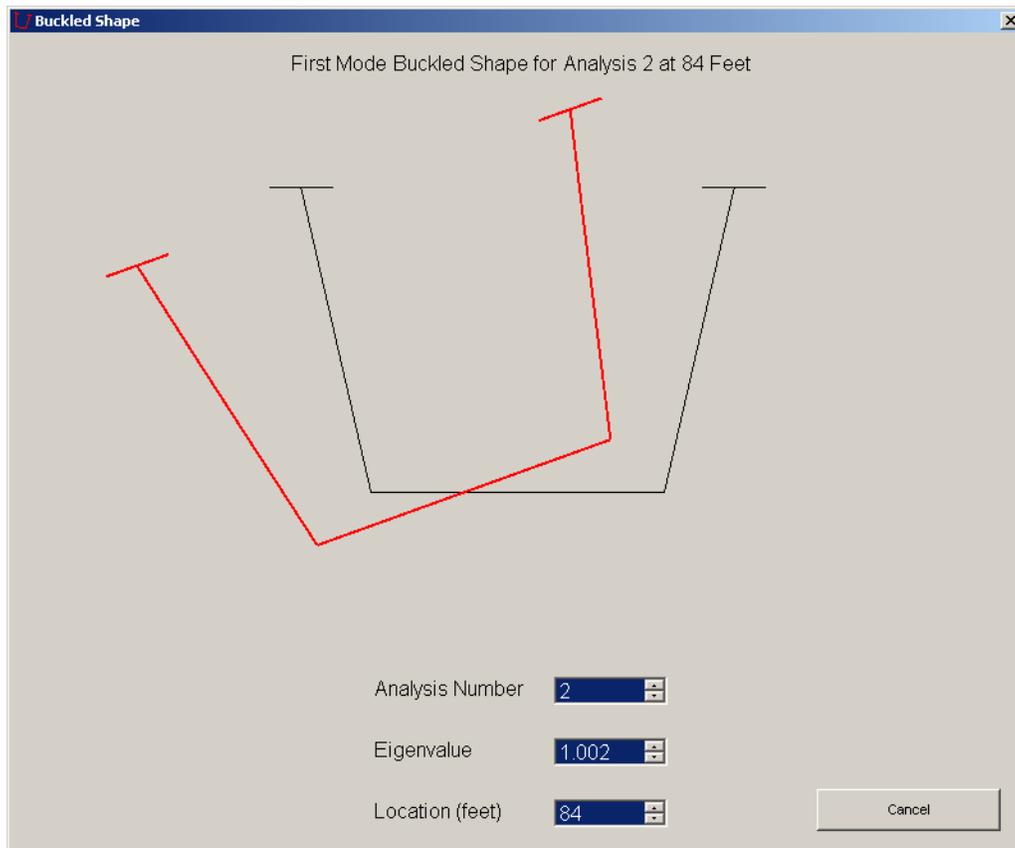


Figure 28 Buckled Shape form

Tip: In the Buckled Shape form shown in Figure 28 above, the user can select the analysis number and eigenvalue desired, then click in the Location scroll box and hold the down arrow. This will show an “animation” of the buckled shape, moving along the length of the bridge. The user will in most cases be able to quickly determine the mode of buckling and the affected areas of the girder.

OTHER ISSUES

UTrAp 2.0 has been developed and tested in Windows XP, and also works with Windows 2000. The complete compatibility of the program with earlier versions of Windows is not guaranteed. A physical memory of at least one gigabyte is recommended, particularly for buckling analyses. If the available physical memory is insufficient, virtual memory will be used, greatly increasing the analysis time. In order to use the Export to Excel feature, the user must have Excel 2002 or newer installed on the computer running UTrAp 2.0.

SUMMARY

This user's guide has presented the capabilities and limitations of UTrAp 2.0 and explained the graphical user interface. Little or no knowledge of finite element modeling is required to obtain accurate results for both linear and buckling analyses of trapezoidal box-girder bridges. The bridge designer will find UTrAp 2.0 a valuable tool for analyzing steel box girder bridges, particularly under construction loading.

REFERENCES

- Popp, D.R. (2004). "UTrap 2.0: Linearized buckling analysis of steel trapezoidal box girders." Master's Thesis, The University of Texas at Austin, Austin, Texas.
- Topkaya, C., Yura, J.A., and Williamson, E.B. (2004). "Composite shear stud strength at early concrete ages," *J. of Structural Engineering, ASCE*, Vol. 130, No. 6, pp. 952-960.
- Topkaya, C. and Williamson, E.B. (2003). "Development of computational software for analysis of curved girders under construction loads." *Computers and Structures*, Vol. 81, pp. 2087-2098.
- Topkaya, C., Yura, J.A., Williamson, E.B., and Frank, K.H. (2002). "UTrap: Finite-element-based Software for the Analysis of Curved Trapezoidal Girders under Construction Loads." Report No. FHWA/TX-03/1898-3. Texas Department of Transportation, Research and Technology Implementation Office, Austin, Texas.