THREE-DIMENSIONAL FINITE ELEMENT SOFTWARE FOR STEEL BRIDGE ERECTION AND CONSTRUCTION

Introduction and Background

The critical stage for the stability of many bridges often occurs during construction when the loading and support conditions of the bridge are widely variable. Historically, the majority of bridge failures have occurred during construction due to the unpredictable behavior of the structural system from the perspective of loading, support conditions and stability. The safety of the erection scheme is dependent upon the evaluation of the behavior with several critical stages involving the partially erected structure when limited bracing may be available. In addition, a great deal of uncertainty on the structural behavior can occur during the concrete deck placement in which the steel girder alone must support the full construction load and the system is susceptible to a variety of potential instabilities. While most commercial software programs focus on the analysis of completed bridges in their fully-composite state, the authors are aware of no commercial analysis programs that are currently available to model accurately in a fast way partially erected steel I-girder bridges. Curved bridges in particular are difficult to model, and the combined bending and torsion that occurs due to the geometry can result in a flexible structural system in cases with a partial erected system. In the development of a suitable erection and concrete deck placement scheme, obtaining a good prediction of the deformations and strength of the system at the various construction stages can be critical to the success of bridge construction. A structural analysis providing this valuable deformation and strength prediction will often result in the elimination of problematic conditions during construction and can therefore avoid costly construction delays or unsafe conditions for the construction personnel and travelling public.

Most current commercial programs that are used in practice for the analysis and design of curved and skewed I-girder bridges (such as MDX or DESCUS) perform a refined grid analysis. Zureick (1) acknowledged that the most accurate means to analyze complex bridge geometries is with the use of 3D shell models and although it was believed that such models would eventually replace these simplified models, the complexity involved in creating 3D models as well as the post-processing required to comprehend the results from such a model have greatly hampered wide-spread use. UT Bridge aims at filling this gap. An improved modeling approach to conventional grid models was considered by Chang (2), using thin-walled beam theory. A program for simulating the construction of curved steel I-girders was proposed and used in NCHRP Report 725 "Guidelines for Analysis Methods and Construction Engineering of Curved and Skewed Steel Girder Bridges" (3). However, the large size of the input files has also limited the use of such models. While historically the use of 3D modelling techniques were hampered by limited computational resources, technological improvements have resulted in the ability for simple laptop and desktop computers to carry out very sophisticated analyses. However the bottleneck that is slowing the use of 3D modelling in routine analyses is actually the necessary time for users to become proficient in the modelling techniques as well as the required time to create these detailed models and analyze the results.

In an effort to fill the void of available analysis options for designers and erection engineers, the program UT Bridge was developed (Stith, 2010) through funding provided by the Texas Department of Transportation (TxDOT). The software allows the user to efficiently develop a 3D model of steel I-girder bridges for the analysis during erection or deck placement. While such models can be generated in several general-purpose finite element programs that are commercially available, the creation of such models can take several days or weeks even for the most experienced analyst. The UT Bridge software consists of three modules that include a pre-processor, processor, and a post-processor. The user defines the basic bridge geometry through a series of input forms in the pre-processor (as described in Figure 1), which was developed using a Visual Basic (VB) interface. The user can select either an eigenvalue analysis or a first-order structural analysis. Once the geometry of the bridge is defined, the pre-processor also allows the user to specify the erection scenario or deck placement scheme considering temporary supports such as shore towers or holding cranes. When the user is satisfied with the geometry and erection/deck placement scenario, the input file is sent to the FORTRAN-based processor that is based upon the pre-processor to carry out the structural analysis. Upon completion of the analysis, the user can then open and review the results in the C++ based post-processor.

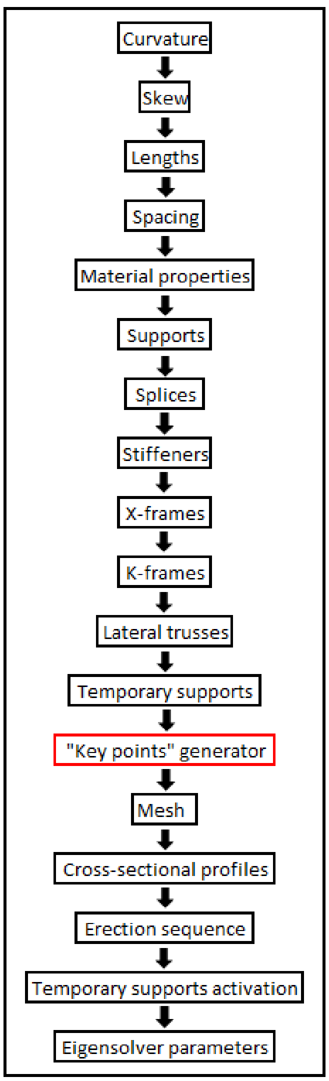

Figure 1 – Pre-processor (user-friendly VB forms) architecture of UT Bridge Version 2.0

The program is currently in Version 1.6 and has been well accepted and used by a number of designers and erection engineers. As with most software packages, through the use, various bugs and limitations in the software have been identified since its release in 2010. Because the software is free, the developers have not had the resources to continually update the software; however a major effort began in 2014 to update and improve the software.

The expected release date of Version 2.0 of the software (henceforth referred to as V2.0 in the remainder of the document) is expected in mid- to late- 2016. Although the basic framework of the software is the same (pre-processor, processor, post-processor), these individual components have been totally changed from Version 1.6 (henceforth referred to as V1.6). In addition to changing the pre- and post-processors, V2.0 also includes new element formulations with improved accuracy, a new solver, and a new eigensolver. V2.0 will result in significant improvements in the accuracy and capabilities of the software. The program provides a 3D model with improved modelling accuracy of curved and skewed I-girders systems using quadratic isoparametric shell elements for all primary steel members (connecting plates and stiffeners included) and truss or beam elements for all brace members, using state-of-the-art solver and eigensolver algorithms.

While UT Bridge is able to perform a deck placement analysis, the scope of this paper is limited to the erection sequence analysis of I-girder systems in V2.0. The goal of this paper is to outline the basic changes to the program and to demonstrate the use of the program. Validation results of V2.0 with commercially available general-purpose finite element programs are provided. In addition, a brief overview of subsequent changes that are under consideration for the software are also provided.

Model Generation

Modeling of the structure in UT Bridge is achieved through a user-friendly interface using a series of VB forms. The geometry is defined using a minimum number of parameters, namely the bridge type (straight or curved), the number of girders, the length of each girder, the girder spacing (which may be non-uniform), the radius of curvature (in case of a curved bridge), and the support skew. Nodes are defined at the mid-plane of the shells. The program uses Cartesian coordinates for straight bridges and cylindrical coordinates for curved bridges (using the center of curvature of the bridge as the origin of the coordinate system), which allows for easy and exact nodes coordinates calculation. The accuracy of the node generation algorithm results in a proper shell orientation, which is critical to the definition of the shells stiffness matrices. Therefore, the nodes are defined in the local isoparametric coordinates and must be transformed to global coordinates. Shell kinematics is also kept accurate, as a set of local orthogonal vectors at each node is required. Separate sets of orthogonal vectors are defined, depending on whether the shell represents part of a flange, a web, a connecting plate or a stiffener. As a single node may be shared by a flange and/or a web and/or a connecting plate finite element, several local sets of vectors may be required for individual nodes.

The finite element that was selected for V2.0 is an isoparametric eight-noded quadratic general shell element such as that depicted in Figure 2. The element uses five degrees of freedom per node (three displacements, two rotations, with no "drilling" degree of freedom). This element is commonly recommended to model thin shells. The element is very similar to the most commonly used elements in many general-purpose FEA programs. The program that is being used to validate Version 2.0 is ABAQUS (5), which has the S8R5 element that is similar to the new element. Comparisons between V2.0 and the ABAQUS models are subsequently presented in this paper. Full integration on the element was adopted, using three integration points in each direction. Integration through the thickness is done using two Gauss points. Overall, a series of eighteen integration points are therefore considered.

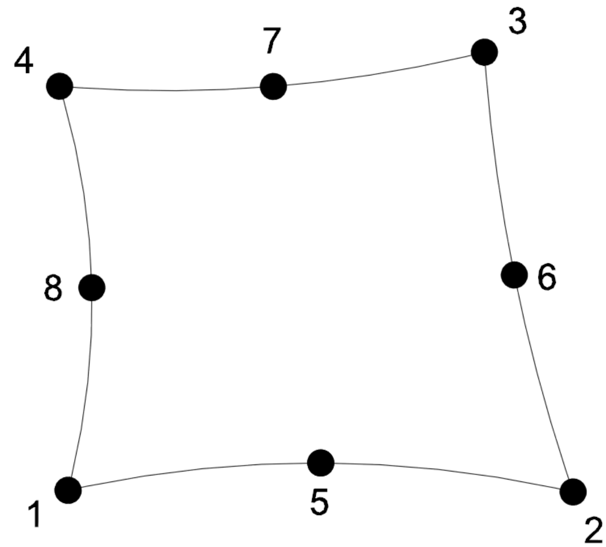

Figure 2 – 8-noded isoparametric finite element

Applying boundary conditions to curved girders requires special care. While the overall stiffness matrix is defined in global Cartesian coordinates (even for curved girders), the restraints at each support, however, are specified in local coordinates as demonstrated in Figure 3. For example, restraining the out-of-plane displacement for a curved girder defining a 90° angle in plane means restraining the x-displacement for a node located at the start of the girder, but restraining the y-displacement for a node located at the end of the girder. Such restraints may be accomplished either numerically, by adding springs at the corresponding degrees of freedom, or through the definition of local degrees of freedom that are then restrained. Adding springs is often not the optimal solution, since a large stiffness value may result in an ill-defined stiffness matrix, while a low stiffness value may lead to insufficient restraint. However, working on local degrees of freedom for the stiffness matrix assembly and solving, and then back transforming the local displacements to overall Cartesian displacements, proves to be the most accurate method of applying boundary conditions to a curved girder.

Specifically, the restrained degrees of freedom are the following: in the case of a single girder (no cross-frame), a pin support requires restraints in the three displacements in the bottom flange to web node, as well as the out-of-plane displacement of the top flange to web node; a roller support requires restraints in the vertical and out-of-plane displacements of the bottom flange to web node, as well as the out-of-plane displacements of the top flange to web node. In the case of multiple girders braced together, the pin or roller restraints are applied only at the bottom flange to web support nodes. By restraining only the translations of only the flange to web nodes, twist can be adequately restrained, but warping is still allowed.

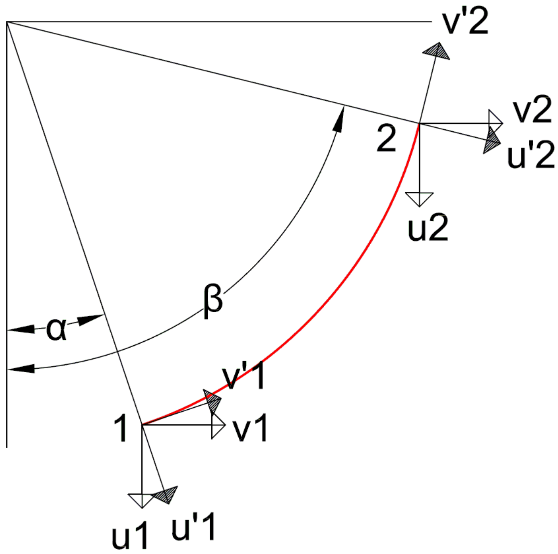

Figure 3 – Global vs. local degrees of freedom

One of the strengths of the UT Bridge software has been the ability to model temporary supports such as shore towers or holding cranes to determine the impact on the behavior of the partially erected system. Shore towers are temporary restraints that physically behave similar to roller supports, restraining the bottom flange to web node vertical and out-of-plane displacements, and the top flange to web node out-of-plane displacement. Therefore, it is assumed that the girder will be braced at the shore tower location. Holding cranes are modeled as an upward reaction at the point of support (on the top flange) – but twist and lateral deformation are not restrained.

At each section along the length of a girder, a series of seventeen nodes are defined, which results in eight elements as depicted in Figure 4. The resulting cross-section consists of two elements for each flange and four for each web. Having two elements per flange was considered enough, especially when using a quadratic shell element. Based on an aspect ratio that is targeted to be close to one, or in the worst case under five, having four elements for the web was considered adequate.

Although UT Bridge V1.6 models the cross frames as tension-only diagonal systems, V2.0 allows the user to specify either compression system x-frames or k-frames (regular or inverted). Another new addition to V2.0 is the ability to specify a lateral truss at the top and/or bottom flange. The x-frames are modeled using three-dimensional truss bars connected to the mid-edge nodes of the outer web shells. The k-frames are connected to the same nodes but use beam elements for the top and bottom chords and truss bars for the diagonals. Finally, the lateral trusses are connected on the flanges exterior nodes and are modeled using truss bars. The truss and beam element stiffness is computed in the local coordinates and then converted to global Cartesian coordinates. Defining braces is easily achieved in the VB forms by specifying their location on the girders. For curved girders, working on cylindrical coordinates, and specifically, on the curvilinear coordinate along the girder, results in quick and accurate modeling.

Meshing of the model is achieved automatically after the call to a separate FORTRAN program that stores a series of “key points” along each girder. Those key points may correspond to splices, supports, stiffeners, or any other location that the user selects based from the bridge framing plans. Exporting framing plans to actual models is therefore made relatively simply. For each girder, once the necessary parameters are defined, the program automatically meshes the model, based on a mesh size specified by the user. Modeling the structure and meshing the model can often be carried out in a matter of minutes. The automatic meshing does not prevent the user from selecting different mesh sizes for different segments. For example, to minimize the size of the stiffness and geometric stiffness matrices, the user may opt to use a relatively coarse mesh. But at segments where the bridge behavior requires more precision, for example around negative moment regions, the user may define a finer mesh.

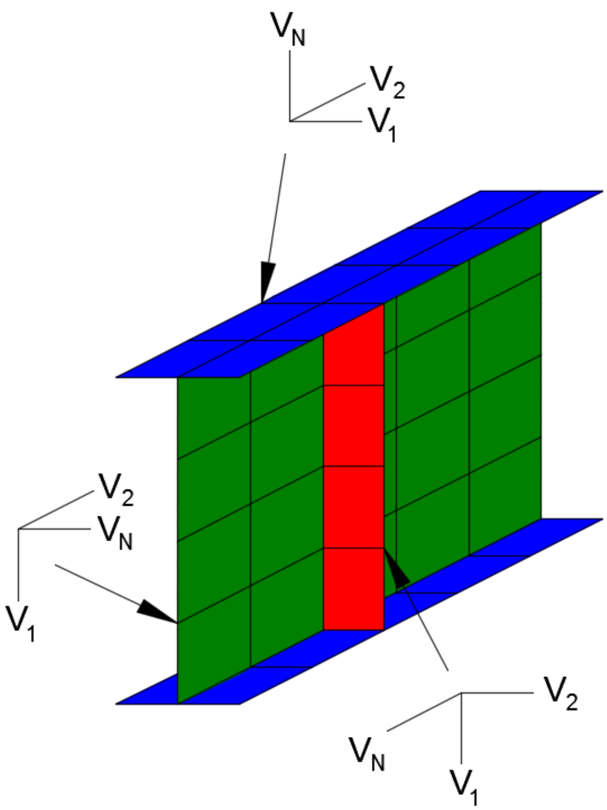

Figure 4 – Mesh and node orientation

Furthermore, by defining different steps and turning on user-specified segments for each girder at each step, UT Bridge is able to perform a full erection sequence analysis. Based on the selected segments, the program automatically activates or deactivates the corresponding nodes, elements, braces, loads and supports. The user is also prompted to specify when to activate and deactivate temporary supports. Altogether, this enables a quick analysis of a structure at different steps along the corresponding erection sequence instead of having to work on a different model for each step.

Processor

A flow chart of the solution process is shown in Figure 5. Defining K as the overall structural stiffness matrix, U the displacement vector, and F the load vector, the equation K*U = F, is solved using the latest PARDISO solver available in the INTEL FORTRAN compiler. The PARDISO subroutine allows for a quick and accurate solution of a set of sparse linear equations. It should be noted that the stiffness matrix, as well as the geometric stiffness matrix, are stored in the Compressed Row Storage (CSR) format. This means that only the non-zero matrix coefficients are stored, which in the case of the highly sparse matrices defined in structural mechanics represent less than 1% of all the matrix coefficients for simple models, or even less than 0.1% or 0.01% for large or very large models. This allows for optimized memory storage and faster solving.

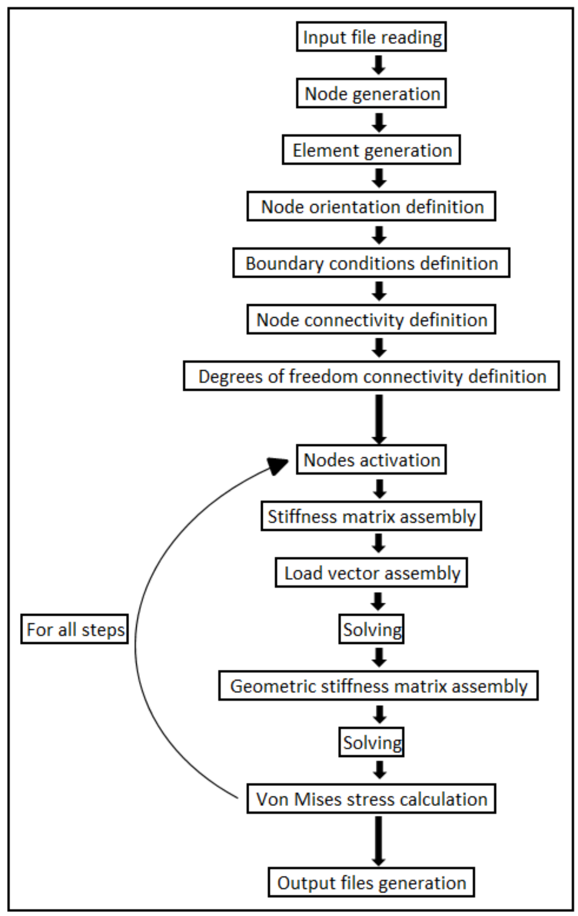

Figure 5 – Processor architecture

Once the linear elastic solution is found, the Von Mises stress invariant is calculated at each integration point of all the shells. The values at the integration points are extrapolated to the shell nodes, using the shape functions of a nine-noded isoparametric quadratic element, as there are nine integration points per plane through the shell thickness. Finally, the values at the nodes are averaged to achieve a smooth stress field. As bending and torsional stresses dictate the behavior of the girders, it was chosen to extrapolate the Von Mises stress invariant through the shell thickness, based on the line defined by the stress values at the two integration points. For the top and bottom flanges, the stress is displayed at the respective upper and lower surface, whereas for the webs and the stiffeners, the stress is displayed at the mid-plane. Finally, to comply with the shell finite element theory, which assumes that there is no normal stress orthogonal to the shell surface, a total of three actual stress fields are represented: one for the flanges, one for the webs, and one the stiffeners and connecting plates. For example, for a node that is shared between a web element and a stiffener element around mid-span of a girder, two values are actually stored and displayed: one for the web, large due to the in-plane bending stress, and one for the stiffener, low as the in-plane bending stress for the web actually results in a zero out-of-plane stress for the stiffener. Typically, only the shear stress components contribute to the Von Mises stress invariant for stiffeners and connecting plates.

For buckling analyses, once the static solution is obtained, the structural geometric stiffness matrix is assembled and the program performs an eigenvalue buckling analysis of the structure. This means that the equation $(K + \lambda * K_g) * X = 0$ is solved, where K_g is the geometric stiffness matrix, X an eigenvector and λ an eigenvalue. In the field of structural stability, the geometric stiffness matrix is computed from the stresses derived from the linear elastic static solution, an eigenvalue is a multiplier on the applied loads that leads to buckling of the structure, as defined by a bifurcation from the linear elastic solution, and an eigenvector is the deformed shape of the structure at that point. In V2.0, the FEAST eigensolver available in the Intel Math Kernel Library (MKL) was selected. The FEAST eigensolver was developed in the late 2000s at the University of Massachusetts (6). As stated by its author, it “takes its inspiration from a density-matrix representation and a contour-representation technique” originally developed in the field of quantum mechanics. This technique enables the quick solution of almost any eigenvalue problem, particularly in the case of symmetric real sparse matrices, which is the case in structural mechanics. Each node is connected to a maximum of ten other nodes in an overall model that contains thousands of nodes, while the stiffness and geometric stiffness matrices are symmetric due to the virtual work principle.

For the purpose of finding critical buckling modes, the user is prompted to enter a search interval within which all the modes will be encountered. For the sake of keeping a limited number of output files, a maximum of five modes is retained for each step, but this may be increased in the future. The eigensolver is able to capture either global or local (flange or web) buckling modes.

Examples

An important step in program development is the validation of the program. A number of systems have been evaluated to date to ensure that proper modelling decisions have been made in the development of V2.0. For the purpose of validating the program, a series of benchmark problems have thus far been considered. The systems consist of full bridges with both straight and curved geometry. Cases with and without support skew have been modeled. For validation purposes companion models have been created using the general-purpose FEA program ABAQUS. Comparisons have been made with numerical comparisons of deflections, stresses, eigenvalues, etc. Some of these numerical comparisons are presented later in this paper in a table. Comparisons have also been made with the graphical output using contour plots from the UT Bridge V2.0 and ABAQUS. The modeling parameters in ABAQUS were selected to mirror as much as possible those assumed in UT Bridge. In particular, the ABAQUS S8R5 general shell was adopted since the element that has been incorporated into UT Bridge is based upon the same formulation. One of the few modelling differences between V2.0 and ABAQUS are how the cross-frames are actually attached to the girders. Since selecting the mid-edge nodes of quadratic elements is not possible in ABAQUS, an additional number of four elements were generated along the webs in order to maintain the same stiffness for the cross-frames, which results in a slight flexibility introduced into the connection.

A number of screen captures are presented on the following pages that show comparisons between the UT Bridge program viewer and the ABAQUS model. Significant changes have been incorporated into the V2.0 post-processor compared to V1.6. The graphics of the viewer is very similar to the ABAQUS output. As with the other forms, the viewer is encoded in Visual Basic. The model itself is displayed using an algorithm encoded in OpenGL, which is with DirectX one of the two main standards used in three-dimensional graphics display. However, as OpenGL is specifically targeted at addressing C++ based codes, the Open Toolkit (OpenTK) library was added to the VB code in order to be able to use some of the already defined OpenGL functions. Generation of the model, its geometry, displaced shape, buckled shape, and Von Mises stress distribution, however, is achieved through a “manual” specific 3D rendering algorithm that allows for control of advanced parameters. These parameters include the magnification factor of the displaced shape and mode shapes, the rainbow pattern used to represent the magnitude of those displacements, the offset between the shells edges and their interior surface (in order to avoid the so-called “stitching” effect that may affect the overall quality of the rendering). The output files required for the rendering are generated at the end of the main FORTRAN calculations and are formatted similarly to the well-known .obj format encountered in three-dimensional graphics display. The viewer form allows for a rapid display of the model, boundary conditions, temporary supports, displaced shape, buckling eigenvalues, mode shapes and Von Mises stress distribution for each step of the erection sequence. One of the major improvements between V1.6 and V2.0 is that the Post-Processor of UT Bridge no longer requires opening separate program modules, which was required in the past. This allows the user to move around the program and switch between input screens and screens showing the model geometry much quicker.

Curved Girder Example

In order to validate the accuracy of V2.0 with ABAQUS, a highly curved prismatic girder was selected. The girder has a radius of curvature of 26.5' and a span of 41.67' long. The respective flange and web dimensions are 12.0"x0.75" and 60.0"x0.75". The tight radius of curvature was selected to validate the program and is not meant to reflect the curvature of actual road or railway bridges. However, such geometries may not be uncommon for other applications such as a pedestrian bridge. As the girder defines a 90° angle in plan, this benchmark problem can be potentially considered as a worst case scenario as far as checking boundary conditions. Figures 6 and 7 show the displacement contours in the x- and y-directions with maximum values of 7.602". An interesting aspect of this problem is that the x- and y- displacements are symmetric due to the symmetry of the problem. The contour of the maximum resultant deformation from V2.0 is shown in Figure 8 with a maximum value of 21.175". The corresponding contour of the maximum resultant displacement from ABAQUS for this problem is shown in Figure 9, with a maximum value of 21.210”, which is almost identical. It should be noted that there is a slight difference between the contour from V2.0 and ABAQUS. Whereas V2.0 displays a continuous (rainbow) contour, the contour graphs from ABAQUS are discretized.

Although an eigenvalue buckling analysis is not very meaningful for a girder with significant horizontal curvature, the analysis was carried out in the two programs. The eigenvectors (buckled shapes) for the first mode from the V2.0 and ABAQUS models are shown in Figures 10 and 11, respectively. As noted in the captions, there is excellent agreement between the corresponding eigenvalues for the two programs. Similar agreement was also achieved for higher modes.

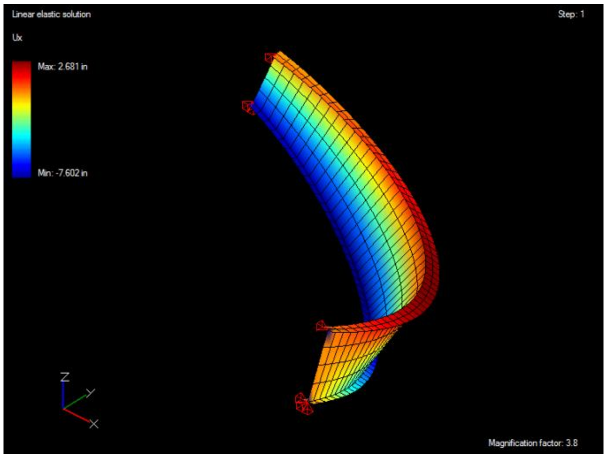

Figure 6 – Curved girder deformed shape (UT Bridge, $U_{x,max}$ = -7.602")

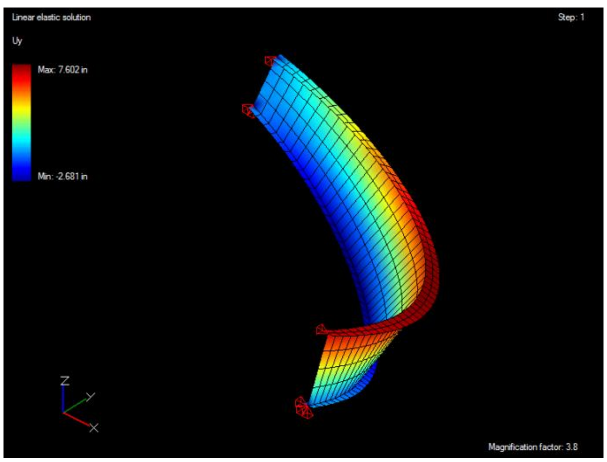

Figure 7 – Curved girder deformed shape (UT Bridge, $U_{y,max}$ = 7.602")

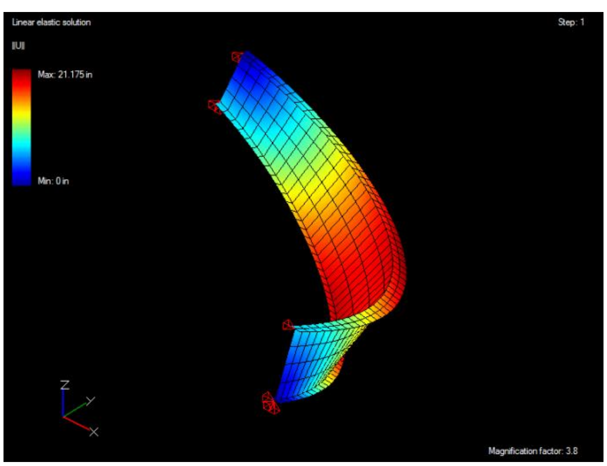

Figure 8 – Curved girder deformed shape (UT Bridge, U_{max} = 21.175")

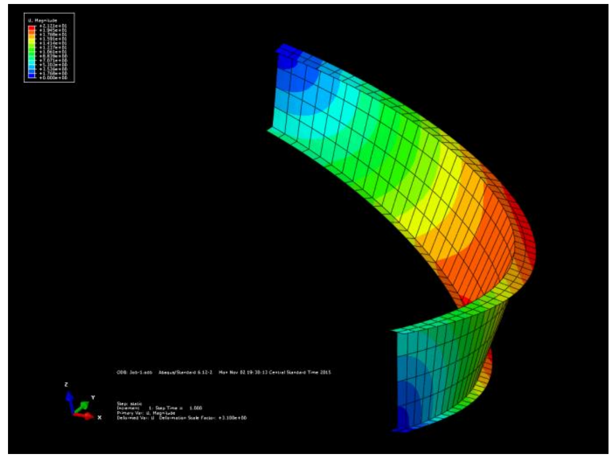

Figure 9 – Curved girder deformed shape (ABAQUS, U_{max} = 21.21")

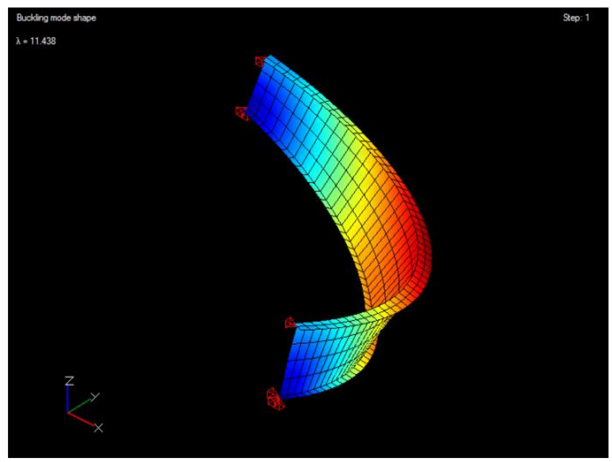

Figure 10 – Curved girder 1st mode shape (UT Bridge, λ = 11.438)

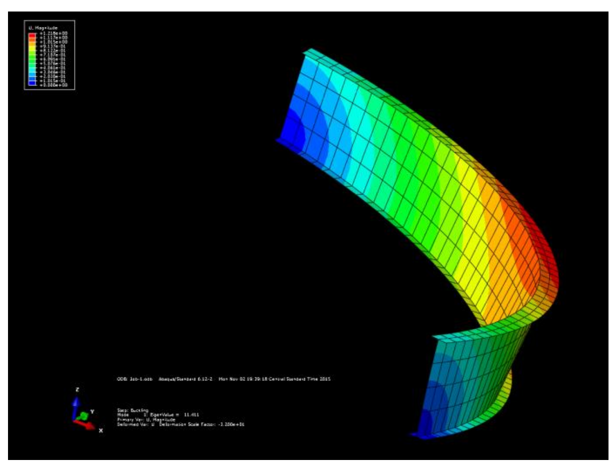

Figure 11 – Curved girder 1st mode shape (ABAQUS, λ = 11.411)

Skewed bridge

Similar to the relatively severe case of the horizontally curved girder from the last example, an equally severe case was also considered for a straight bridge with skewed supports. A plan view of the girder system is shown in Figure 12.

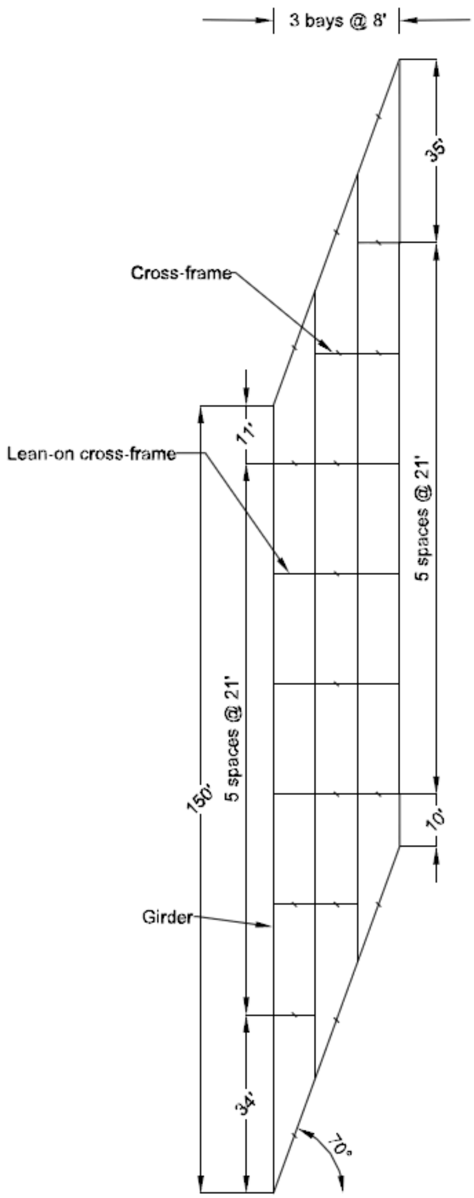

Figure 12 – Skewed bridge framing plan

The structure is a straight, four-girder system with a relatively extreme support skew angle of 70°. The bridge had 12 intermediate cross frames and 6 locations where "lean-on" bracing was used where only the top and bottom struts were provided. The girders were prismatic with top flange dimensions of 18"x0.945", bottom flange dimensions 20"x2.165", and web dimensions 56.1"x0.5625". As noted above, a framing plan taking advantage of lean-on bracing is considered. Four steps are defined for the erection sequence, with one full girder lifted at each step. Nearly exact correlation was achieved between the V2.0 and ABAQUS models for each stage of the erection. Figures 13 and 14 show a comparison of the contour graph of the result displacement for V2.0 and ABAQUS. The maximum value from each analysis was 2.8".

The results from the eigenvalue buckling analyses from each stage were also nearly exact between the two programs. The eigenvectors (buckled shapes) from the two programs are shown in Figures 15 and 16, with the corresponding eigenvalues being 2.6 and 2.5.

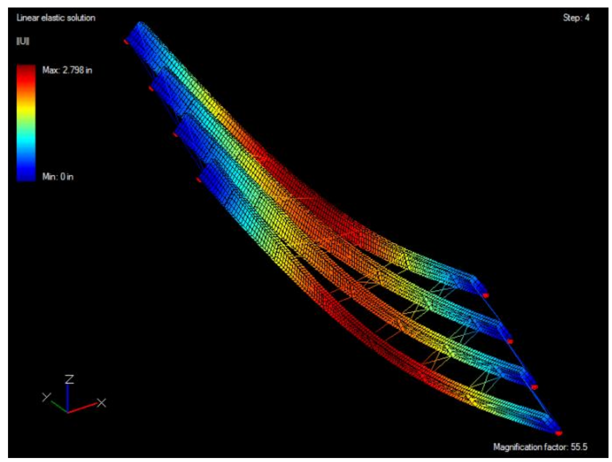

Figure 13 – Skewed bridge deformed shape, step4 (UT Bridge, U_{max} = 2.8")

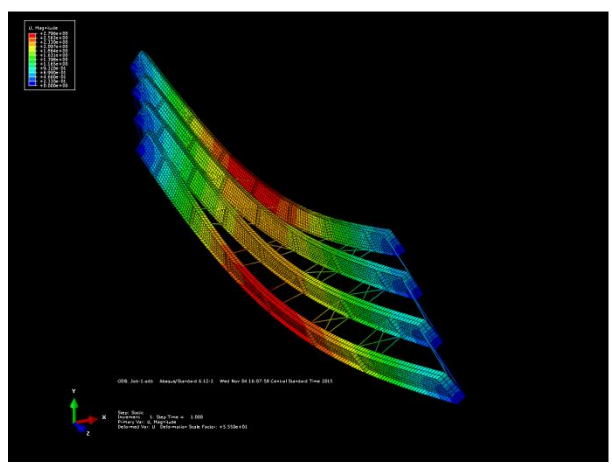

Figure 14– Skewed bridge deformed shape, step4 (ABAQUS, U_{max} = 2.8")

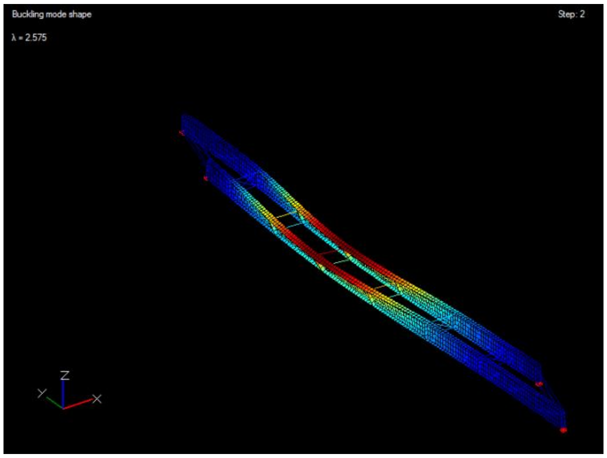

Figure 15 – Skewed bridge 1st mode shape, step 2 (UT Bridge, λ = 2.6)

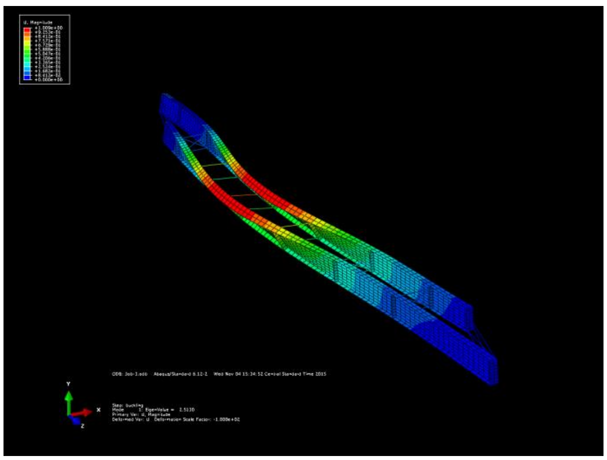

Figure 16 – Skewed bridge 1st mode shape, step 2 (ABAQUS, λ = 2.5)

Curved bridge

An additional validation analysis was carried out on the four-girder system depicted in Figure 17. The two-span bridge had a radius of curvature of the inner girder equal to 288' and was braced with 63 cross-frames oriented in a radial manner. The cross-section was kept uniform for the sake of faster modelling in ABAQUS. The cross-sectional dimensions were equal to 28"x2.5" for both the top and bottom flanges, and 114"x1.25" for the web. Four steps were defined in the erection sequence, with two girders lifted at each step. Temporary supports were modeled for steps 1, 2 and 3 to prevent excessive deflections. Good correlation was achieved between both programs at all steps of the erection. Contours of the maximum displacement in the fully erected structures are shown in Figures 18 and 19 with the maximum displacement of 4.1" in V2.0 and 4.3" in ABAQUS.

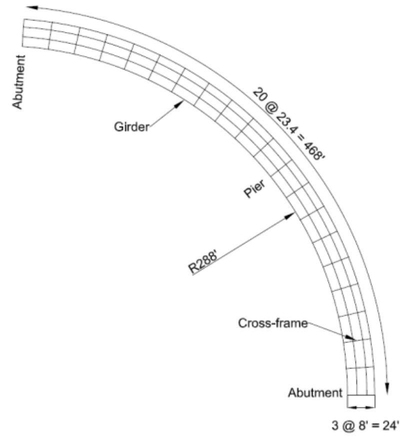

Figure 17 – Curved bridge framing plan

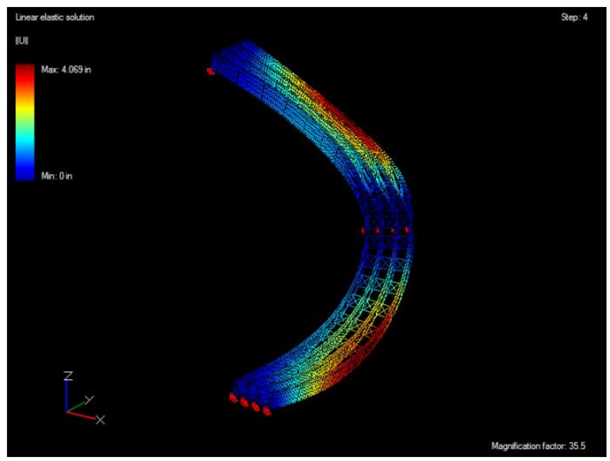

Figure 18 – Curved bridge deformed shape, step 4 (UT Bridge, U_{max} = 4.1")

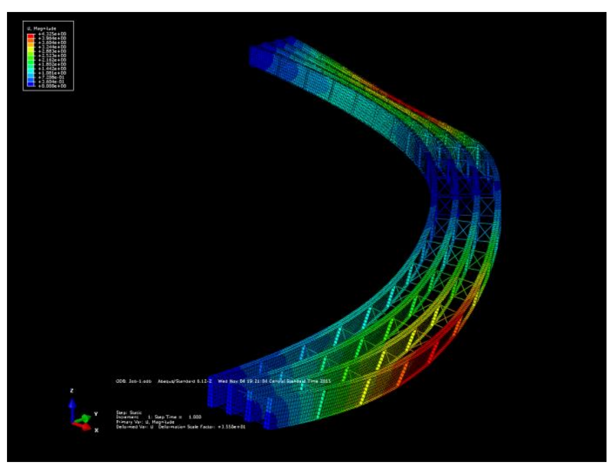

Figure 19 – Curved bridge deformed shape, step 4 (ABAQUS, U_{max} = 4.3")

Validation

The erection conditions of three bridges with relatively severe geometries were analyzed using UT Bridge and ABAQUS. The geometries included severed curvature as well as significant support skew. In all of the analyses, the UT Bridge and ABAQUS models had excellent correlation in the results. A summary of the displacements and eigenvalues are provided in Table 1. Although severe geometrical conditions were modeled, the maximum difference between UT Bridge and a widely used general purpose FEA program was approximately 6%, while for the majority of the results, the percent difference was less than 1%. The differences are likely due to the fact that UT Bridge performs a full integration versus the reduced integration in ABAQUS. Another potential source of the differences is the reduced number of shell elements for the webs for UT Bridge. However, even for the extreme geometries, UT Bridge V2.0 has excellent agreement with the ABAQUS solutions.

	Step		UT Bridge	ABAQUS	Δ
Highly curved girder	1	U_x	-7.602"	-7.614"	**-0.2 %**
		U_y	7.602"	7.614"	**-0.2 %**
		U_z	-19.675"	-19.70"	**-0.1 %**
		U_{max}	21.175"	21.21"	**-0.2 %**
		λ_1	11.438	11.411	**0.2 %**
		λ_2	58.277	56.398	**3.3 %**
Skewed bridge	1	U_{max}	2.427"	2.430"	**-0.1 %**
	1	λ_1	1.121	1.117	**0.4 %**
	2	U_{max}	2.498"	2.501"	**< -0.1 %**
	2	λ_1	2.575	2.513	**2.5 %**
	3	U_{max}	2.651"	2.661"	**-0.4 %**
	4	U_{max}	2.798"	2.796"	**< 0.1 %**
Curved bridge	1	U_{max}	0.183"	0.185"	**-1.1 %**
	2	U_{max}	0.173"	0.175"	**-1.1 %**
	3	U_{max}	0.301"	0.311"	**-3.2 %**
	4	U_{max}	4.069"	4.325"	**-5.9 %**

Table 1 – Results summary

Potential Modifications Under Consideration

Although V2.0 is already very promising, the authors are considering further improvements, which would include the following features:

- The ability to perform a large displacement second-order analysis
- Modeling of dapped ends and/or tapered girders
- Modeling of other types of bracing, such as diaphragms
- Display of moment and shear diagrams.

Summary and Conclusions

A 3D finite element program able to capture the linear elastic behavior and the stability of straight, curved, or skewed steel I-girder bridges was presented. The program was validated with a series of tests on ABAQUS, using similar modeling assumptions. The user-friendliness and accuracy of the UT Bridge program make it a reliable tool for designers and erectors. Modeling time in UT Bridge is significantly reduced. Even very complex structural systems can be quickly modeled.

References

(1) Zureick, A. & Naqib, R., 1999, "Horizontally Curved Steel I-Girders State-of-the-Art Analysis Methods", Journal of Bridge Engineering, vol. 4, no. 1, pp. 38-47.

(2) Chang, C. 2006, *Construction simulation of curved steel I-girder bridges*, PhD Thesis, Georgia Institute of Technology, Atlanta, Georgia.

(3) NCHRP Report 725 "Guidelines for Analysis Methods and Construction Engineering of Curved and Skewed Steel Girder Bridges", 2012, Transportation Research Board.

(4) Stith, J. 2010, *Predicting the behavior of horizontally curved I-girders during construction*, PhD Thesis, The University of Texas at Austin, Austin, Texas.

(5) ABAQUS Version 6.12, Dassault Systèmes Simulia Corporation, 2012.

(6) Polizzi, E., 2009, "Density-matrix-based algorithm for solving eigenvalue problems", Physical Review B, vol. 79, no. 11.