Response of a Large Span Stay Cable Bridge to Blast Loading

Cezary Bojanowski[1]
Marcin Balcerzak[2]

1) Transportation Research and Analysis Computing Center
Energy System Division
Argonne National Laboratory
9700 S. Cass Avenue
Argonne, IL 60439-4828, USA
cbojanowski@anl.gov

2) Warsaw University of Technology
Department of Civil Engineering
Al. Armii Ludowej 16
00-637 Warsaw, Poland
marcin.balcerzak@wp.pl

Abstract

The computational analysis of engineering structures under blast loads faces three fundamental problems: (i) reliable prediction of blast loads imposed on structures, (ii) correct representation of material behavior, and (iii) global analysis of large scale structures. Despite the recent developments in Finite Element (FE) codes like LS-DYNA® and advancements in computational power, addressing all of these issues in a single simulation is not a straightforward task.

In this paper, LS-DYNA capabilities were utilized to simulate the transient global response of a long span cable stayed bridge subjected to blast loading over the deck and to evaluate localized damage to the deck structure. Described in detail is the development of a global FE model of the Bill Emerson Memorial Bridge – a cable-stayed bridge crossing over the Mississippi River near Cape Girardeau, Missouri. The global model takes into account the structural details of the deck, support columns and the pretension in the stay cables. It was partially validated by comparing the calculated natural frequencies with those previously extracted by the Missouri Department of Transportation from data recorded during the 2005 earthquake of M4.1 on a Richter scale (Assessment of the Bill Emerson Memorial Bridge, Report No. OR08-003, September 2007). A detailed model of the central section of the deck was developed to simulate localized damage. Boundary conditions on the detailed model were applied through a sub-modeling technique based on the analysis of the global simplified model.

The results show that a detonation of explosives of a typical size of passenger car and van bomb on a traffic lane in the mid span of the deck is not likely to cause a collapse of the bridge. The vibrations in the stay cables do not lead to yielding of the steel in the strands. The simulation of the local damage shows that – for the chosen van location – the blast may perforate the deck and deform the cross beam. The extent of the damage, however, depends greatly on the assumed erosion criteria.

Keywords: Cable stayed bridge, blast loading, cable pre-tensioning, cable vibration, sub-modeling

Introduction

The computational analyst faces several major problems when modeling large civil engineering structures under blast loading:

(i)   Reliable prediction of blast loads imposed on structures,

(ii)   Correct representation of material behavior, and

(iii)   Global analysis of large-scale structures

The first two can be now handled as a result of the advancement of computer codes such as LS-DYNA and their ability to represent complex material behaviors and loading patterns. However, they are dependent on the quality of the FE model assessed in the verification and validation process of its response. The third item on the above list is mainly related to the available computational resources and also to the level of detail that is to be captured in the structure's response. The following work attempts to simulate the response of a large span cable-stayed bridge to a blast load applied to the top surface of the deck. The simulations were performed on two levels of detail – global analysis of the bridge response on a simplified global model and local analysis of the deck damage performed on a detailed model of a center section of the deck.

The global simplified FE model of the Bill Emerson Memorial Bridge was previously developed to study the cable vibrations under different types of loading [1]. In this work it was adapted to simulate the response of the deck to the blast load. The Bill Emerson Memorial Bridge is a cable-stayed bridge connecting Missouri's Route 34 and Route 74 with Illinois Route 146 across the Mississippi River. A photograph of the bridge and its location on the map is shown in Figure 1a and 1b, respectively. The bridge has a total length of 1206 m (3956 ft). It consists of one 350.6 m (1150 ft) long main span, two 142.7 m (468 ft) long side spans, and one 570 m (1870 ft) long approach span on the Illinois side. Due to flexible connection between the approach and the main bridge, the approach was not modeled here. Carrying two-way traffic, the bridge has four 3.66 m (12 ft) wide vehicular lanes plus two narrower shoulders. The total width of the bridge deck is 29.3 m (96 ft).

Figure 1: Bill Emerson Memorial Bridge

To study bridge vibrations, it is sometimes required to simulate a period of several seconds or more of a real time, which is considered to be a long-lasting event for an explicit Finite Element analysis. For this type of analysis, the time step needs to be long enough and the number of elements needs to be kept reasonable, so that the computations aren't too expensive. This usually requires a reduction in the details of the model and appropriate modeling of the relevant elements of the real bridge. However, for analysis of the blast loading, overly simplified models can lead to significant underestimation of structural damage. The multistage approach with models of different complexity level to some extent addresses all these issues.

Development of the Global Finite Element Bridge Model

In the global model, solid elements with one integration point were used to model the towers. The concrete in the towers was simulated with a linear elastic type of material (MAT_001) as the towers were not subject to blast loading. The bridge deck was composed of two longitudinal steel girders located on the sides, transverse floor beams, and precast concrete slabs. A concrete barrier was located in the center of the bridge, and two railings and additional concrete barriers were included in the design along the edges of the deck. The deck slab was built of two layers of reinforced concrete: a 279 mm thick bottom part and a 76 mm thick upper surface (355 mm total). In the global FE model the deck was represented with one layer of shell elements. A user defined integration scheme through the thickness of shells was used to account for the multiple materials and composite structure of the slab. The cross beams under the deck slab and the main steel girders running along the edges of the deck were modeled using shell elements with a piecewise linear plasticity material model (MAT_024). The cross beams were connected to the girders in the model using spot-welds. The deck slab was connected to the beams using rigid links representing dowels. The total number of elements in the model was nearly 250,000 out of which around 172,000 were shell elements. The smallest shell and solid elements had their edge size around 0.12 m. This resulted in a stability time step of 2.45 x 10^{-5} s for the explicit calculations.
Figure 2 (top) shows the lateral extent of the bridge and (bottom) its perspective view.

Modeling Cables

In the Bill Emerson Memorial Bridge, each cable consists of from 19 to 54 separate strands [2]; each strand is 15.7 mm in diameter. Overall there are 128 cables with 32 different cross sections (mass per unit length) and pre-tensioning forces. The multiple strands were modeled in the FE model with one line of beam elements (with element formulation 6 and cable material MAT_071) with equivalent properties of the bundle of strands. Depending on the location of the cable in the structure the prestressing force varied from 1118 kN to 8624 kN. An example of such input for one of the cables is shown in Table 1. Since inappropriate application of prestressing in cable elements is causing instability of the model, great caution has to be exercised in its initialization. The procedure is described in the analysis section.

Table 1: Simplified LS-DYNA input for MAT_071 (units system kg, m, s, N, Pa)

ID	RO	E	LCID	F0	TMAXF0	TRAMP	IREAD
1	7850	2.10E+11	0	7.910E+6	10.0	0.50	

Note: Refer to the LS-DYNA User's Manual [3] for explanation of input parameters.

Figure 2: FE model of Bill Emerson Memorial Bridge

Concrete Material Model for Shell Elements

Material MAT_172 – concrete model for shell and Huges-Liu beam elements based on the Eurocode 2 [4] model was used to represent the concrete in the global model. This material can represent a smeared combination of concrete and reinforcement, concrete only or steel only. The model includes concrete cracking in tension and crushing in compression, and reinforcement yield, hardening and failure. It was developed for structures under thermal loading, but the strain rate effects are not accounted for. Thus, the strength of the concrete was increased to account for the strength increase at higher strain rates by using Dynamic Increase Factors (DIF) based on the CEB Model Code [5] and [6]. A series of benchmark simulations were performed and compared to experiments [7] for blast loading on a square concrete reinforced slab as shown in Figure 3. The purpose of these simulations was to adjust the properties of the MAT_172 so the FE model

based on the shell elements closely represents the response of the plate modeled with solid elements with Winfrith material MAT_084 and the experiment. In the experiment the 1000 x 1000 x 70 mm square plates were made of a concrete with 42 MPa of an average compressive strength. The charge was placed at the stand-off distance of 3.0 m above the plate's center. The steel bar yield and ultimate strength were measured to be 480 MPa and 600 MPa respectively [7]. The used charge was equivalent to 27.34 kg of TNT.
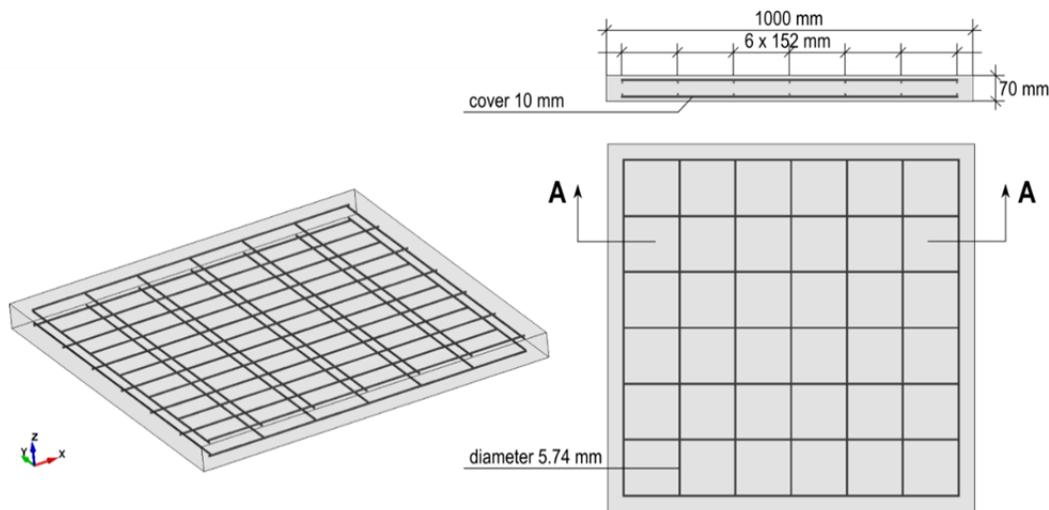

Figure 3: Geometry of the RC plate used in benchmark simulations [7]

Table 2: Simplified LS-DYNA input for MAT_172 (units system metric tonne, mm, s, MPa)

ID	RO	FC	FT	TYPEC	UNITC	ECUTEN	FCC6
1	2.277e-009	100.8	29.85	3	1.0	0.0025	
ESOFT	LCHAR	MU	TAUMXF	TAUMXC	ECRAGG	AGGSZ	UNITL
12000		0.4		1.161		19.00	1.00
YMREINF	PRRINF	SUREINF	TYPER	FRACRX	FRACRY	LCRSU	LCALPS
AOPT	ET36	PRT36	ECUT36				
0.00	3.068e+004	0.16	0.003				

Table 3: Simplified LS-DYNA input for MAT_084 (units system metric tonne, mm, s, MPa)

ID	RO	TM	PR	UCS	UTS	FE	ASIZE
1	2.277e-009	3.068e+004	0.16	42.00	3.77	0.08654	19.00
E	YS	EH	UELONG	RATE	CONM	CONL	CONT
				0.0	-4.0		

The final set of properties assumed for concrete modeled with MAT_172 is presented in Table 2. Properties used by MAT_084 are shown in Table 3. The reinforcing steel was simulated using the MAT_024 piecewise linear plasticity model. The shell section was defined with user defined SHELL_INTEGRATION combining MAT_172 and MAT_024.

For the initial values of compression DIF (CDIF) and tension DIF (TDIF) computed based on [5] and [6], the response of the plate was stiffer than the other computed case and the experiment. Thus, another set of DIFs, decreased by 20% from the estimations from CEB and Malvar formulas for $\dot{\varepsilon}_s$=300, was assumed to give more conservative results on the benchmark problems. CDIF=2.4 and TDIF=7.9 were used in the subsequent models. Table 4 shows comparison of the results between the simulations and the experiment.

Table 4: Test and simulation results comparison

Analyzed Case	Reflected Pressure (kPa)	Reflected Impulse (kPa*ms)	Arrival Time (ms)	Plate Max. Deflection (mm)
Experiment 1	5528	1954	-	13.12
Experiment 2	5712	2412	1.50	9.53
MAT_084 (mesh 10 mm)	5409	1757	1.52	11.68
MAT_084 (mesh 5 mm)	5421	1757	1.52	11.92
MAT_172 (mesh 10 mm)	5241	1733	1.56	12.41
MAT_172 (mesh 5 mm)	5234	1732	1.56	12.52

Figure 4 shows cracks as found by the simulations and visible cracks in the experiments. Two other material models for solid elements were also considered (MAT_072r3 and MAT_159) but the most robust turned out to be material MAT_084 which was further used in the detailed model. Automatic generation of the material parameters in MAT_084 is based on the input of the most common properties and extensive empirical data coded into the material model. The crack database is generated for MAT_084 automatically when option "*q=crack_database_name*" appears in the submission command.
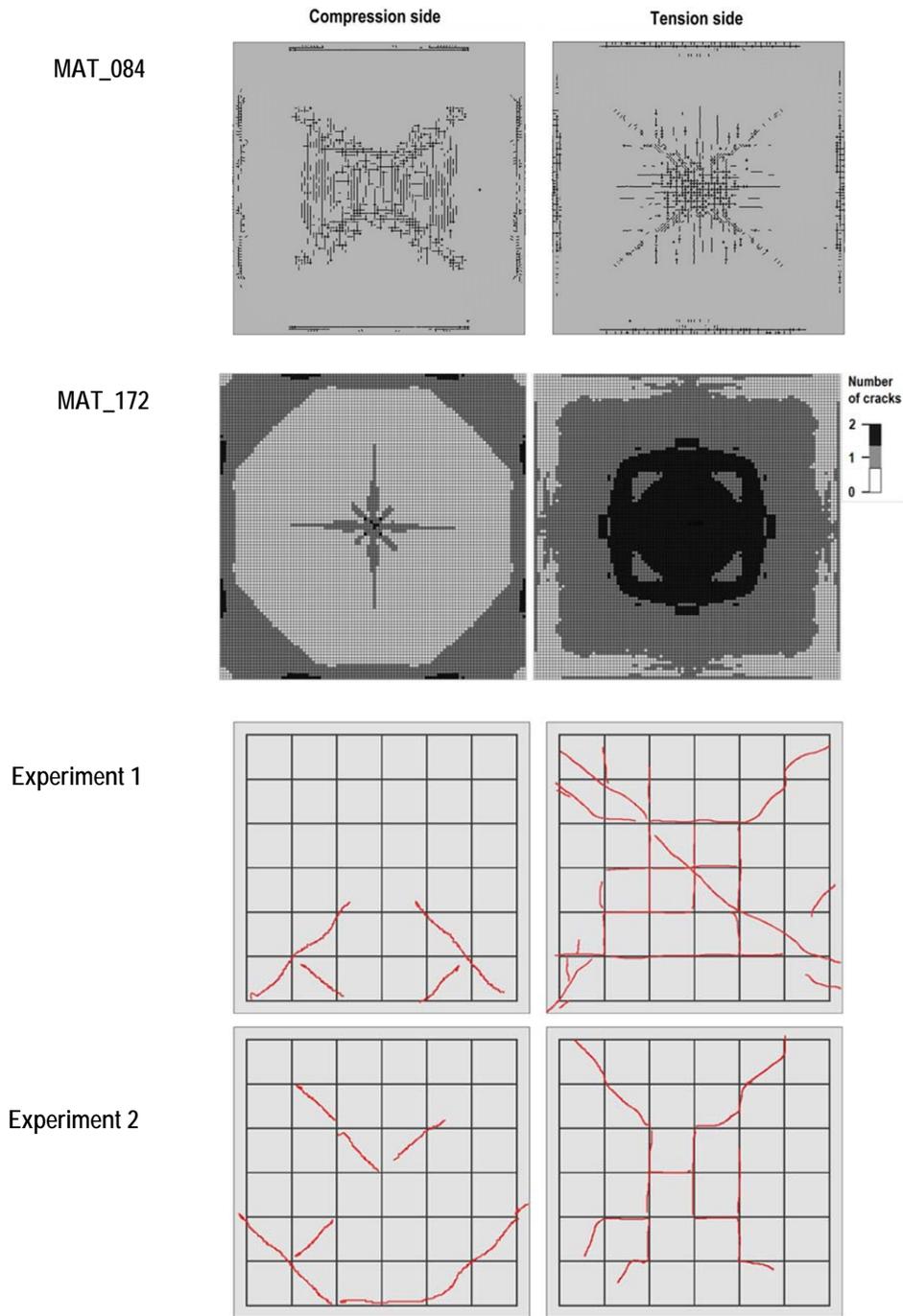
Figure 4: Cracks in the plate

Validation of the Global Bridge Model

The elastic response of the whole bridge was validated through comparison of experimental data for the natural frequencies and the free vibration mode shapes. Such data for the Bill Emerson Bridge was available in an earlier published report – [2].
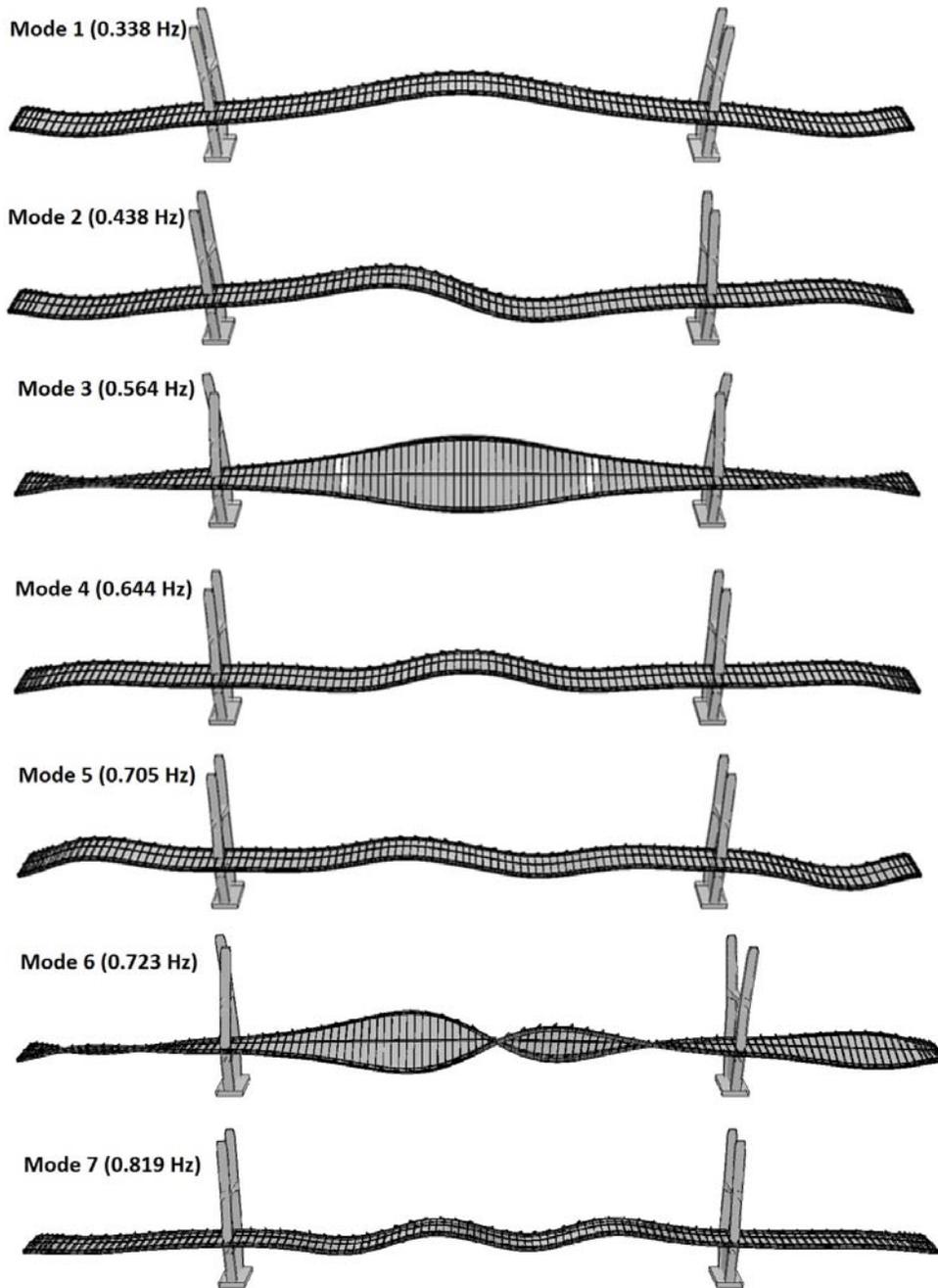
Mode 1 (0.338 Hz)

Mode 2 (0.438 Hz)

Mode 3 (0.564 Hz)

Mode 4 (0.644 Hz)

Mode 5 (0.705 Hz)

Mode 6 (0.723 Hz)

Mode 7 (0.819 Hz)

Figure 5: First seven global mode shapes for FE model of the Bill Emerson Memorial Bridge

The first seven global natural frequencies and mode shapes were extracted from LS-DYNA simulation (see Figure 5). The LS-DYNA beam element with formulation 6 used for modeling cables is unsuitable for eigenvalue analysis. Since it has no rotational stiffness, LS-DYNA produces erroneous eigenvalues and eigenmodes for these elements. In this case, beam elements with formulation 2 were used on the top of cable elements to represent the grout securing the cables in the real bridge and add rotational stiffness in the nodes of the model. This allows for more appropriate eigenvalue analysis. There are multiple local mode shapes at the range 0-1 Hz where the base global mode shapes occur. Here only the global ones (involving deck and tower

deformation) were of interest and only those were presented in Figure 5. The comparison between experimental data and the numerical values for the lowest natural frequencies is shown in Table 5.

Table 5: Comparison of natural frequencies of the FE model with experimental values

Mode number (type)	Experiment	FE model	Relative Error (%)
1 (bending)	0.338	0.338	0.00
2 (bending)	0.438	0.438	0.00
3 (torsional)	0.588	0.564	4.08
4 (bending)	0.650	0.644	0.09
5 (bending)	0.713	0.705	1.12
6 (torsional)	0.775	0.723	6.71
7 (bending)	0.825	0.819	0.73

Blast Loading on the Bridge
Stage I: Initialization of the Stresses in the Global FE Model

Initialization of the models of structures with pre-tensioned (or post-tensioned) members requires special care. The real bridge deck was built with segments, which were separately post-tensioned. The stay-cables were subsequently attached to the segments and also post-tensioned. Achieving the same state of loading in the FE model as in the real structure is difficult and would require repeating the same steps as in constructing the real bridge. Here the initialization of the structure was simplified and the entire model was initialized at once. In the first half second of this simulation a gradually increasing pretensioning force was applied to the cables using the TRAMP parameter in the MAT_071. It was accompanied by large global damping (that lasted two seconds) to prevent excessive shortening of the cables due to pretensioning and in effect to avoid error termination. The initial value of damping constant was adjusted till the simulation became stable. Then reduced damping was applied to allow the structure to achieve the equilibrium state. The pretensioning force was kept constant for ten seconds using the TMAXF0 parameter (as shown in Table 1). At the time 8 s, the damping was reduced to the natural level (5% of critical damping). The system damping coefficient history is shown in Figure 6. The gravity loading was present throughout the entire simulation at the constant level. The deflection of the girder at the mid-span resulting from such initialization process is shown in Figure 7. The deck reached the equilibrium state after about 6 s of simulation time.
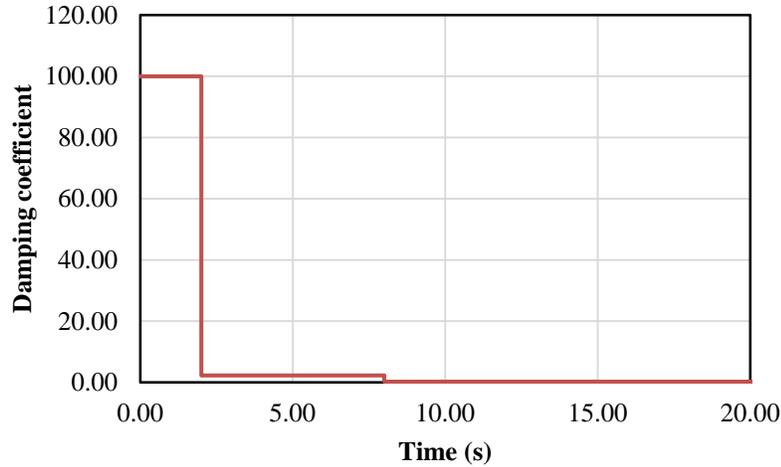
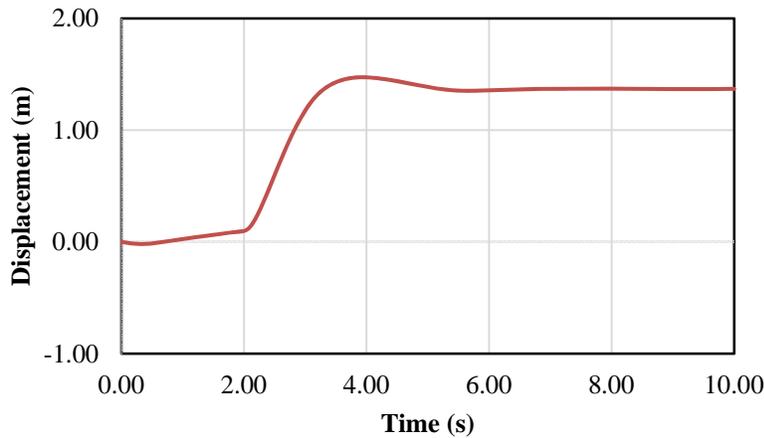**Figure 6:** History of prescribed system damping in the simulation

Figure 7: Displacement of the center of the bridge deck during the initialization phase

Stage II: Blast Loading on the Deck of the Global FE Model

The deformed geometry, stresses, and plastic strains from the initialized model was saved to the "*dynain*" file using command INTERFACE_SPRINGBACK_LSDYNA. In the next computation stage the blast load was applied to the deck using the LOAD_BLAST_ENHANCED keyword. A single "*dynain*" file was reused in all analyzed cases. The load from a passenger size and van size bomb were generated above the deck. The blast load was not applied directly to the cables, however, cases where the cables were detached from the deck were also analyzed. Figure 8 shows the bridge model with highlighted cables that were removed in the analysis. Table 4 lists all the analyzed cases for the two load sizes, number of failed cables and location of the charge.

Figure 8: FE model with highlighted cables removed in the different cases
(left) case 2 and 5 (right) case 3 and 6

Table 6: List of analyzed cases in the stage II

Case	Charge size	Number of removed cables	Location of the charge
1	Small vehicle bomb	0	Middle lane
2	Small vehicle bomb	2	Shoulder lane
3	Small vehicle bomb	4	Shoulder lane
4	Van bomb	0	Middle lane
5	Van bomb	2	Shoulder lane
6	Van bomb	4	Shoulder lane

Figures 9 and 10 show displacement of the middle of the girder in all analyzed cases. The largest deflection was noted for the cases 3 and 6 with 4 removed cables and was about 1.2 m and 1.4 m for car and van size bomb respectively. The maximum deflection occurred in a time less than 1 s. Subsequently the amplitude of the vibrations was decreasing. The largest strains and local damage of the deck however occurred in time less than 0.05 s, which is the topic of the last analysis stage. The largest forces in the cables were registered in the cables closest to the blast (see Figure 11 and 12). The increase of the forces in the cables did not cause the cable steel to yield, although in the worst cases it increased by about 50 %.
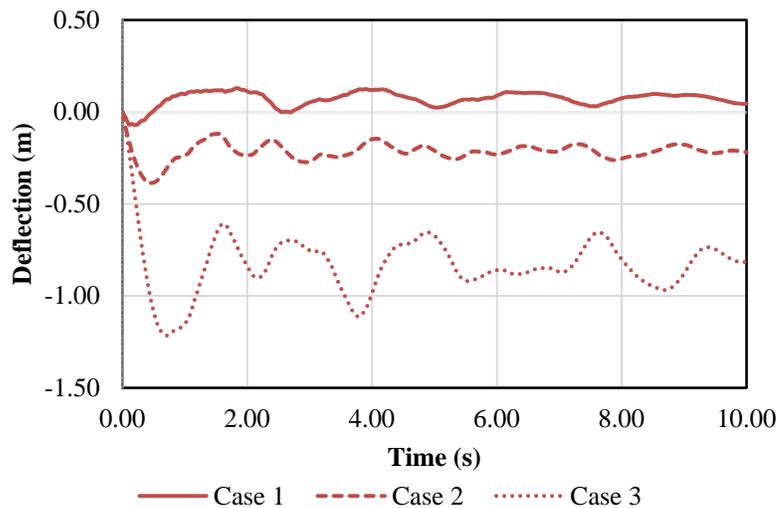

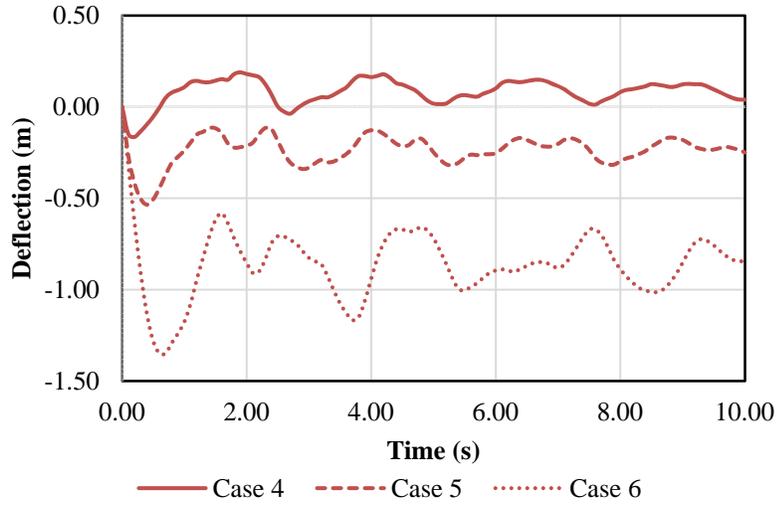**Figure 9:** Vibration of the bridge deck's mid span due to car bomb loading

Figure 10: Vibration of the bridge deck's mid span due to van bomb loading
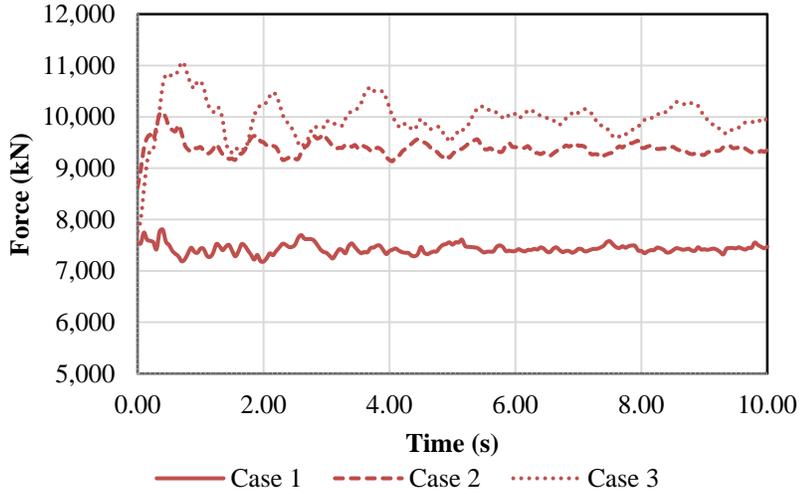

Figure 11: Force history in the stay cable closest to the blast in case 1, 2, and 3
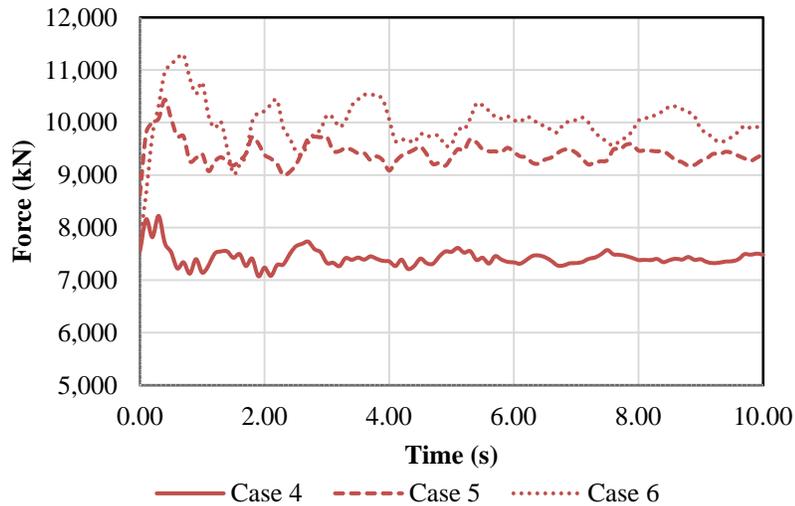

Figure 12: Force history in the stay cable closest to the blast in case 4, 5, and 6

Stage III: Blast Loading on the Detailed Model of the Bridge Section

From the simplified analysis, it was concluded that for the analyzed cases of blast loading the damage can primarily occur locally on the deck – i.e. the induced vibrations will not cause the stay-cables to yield or the girders to fail. The global model as such can be used to estimate an overall behavior of the bridge. However, the localized damage has to be estimated in a more detailed model. The INTERFACE_COMPONENT_NODES keyword (with submission command "*z=interface_segment_file_save*") was used to save the displacements of the edges of the center segment of the deck in the global model simulations. The INTERFACE_LINKING_NODE_SET keyword was used afterward to prescribe these to the boundaries of the detailed model (and submission command "*l=interface_segment_file*"). The segment of interest is highlighted in Figure 13.
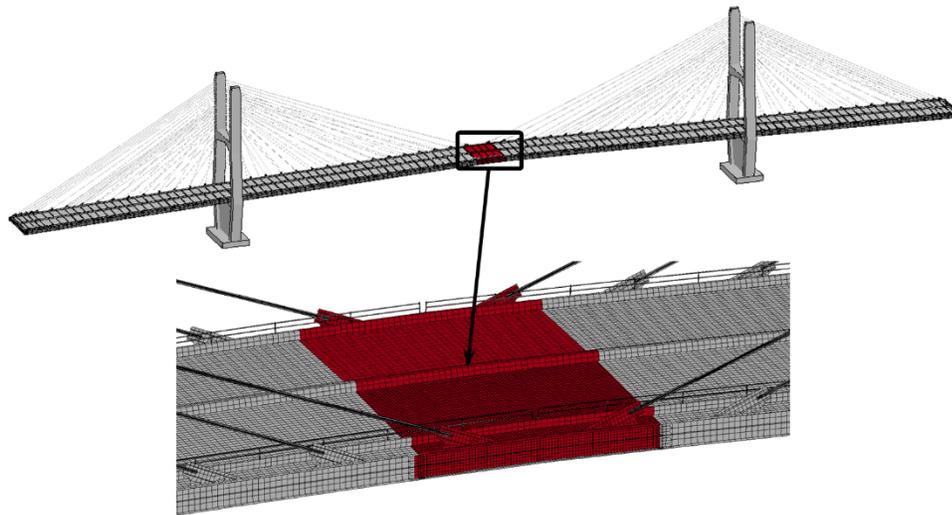

Figure 13: Simplified global model with highlighted section that was modeled in detail at Stage III
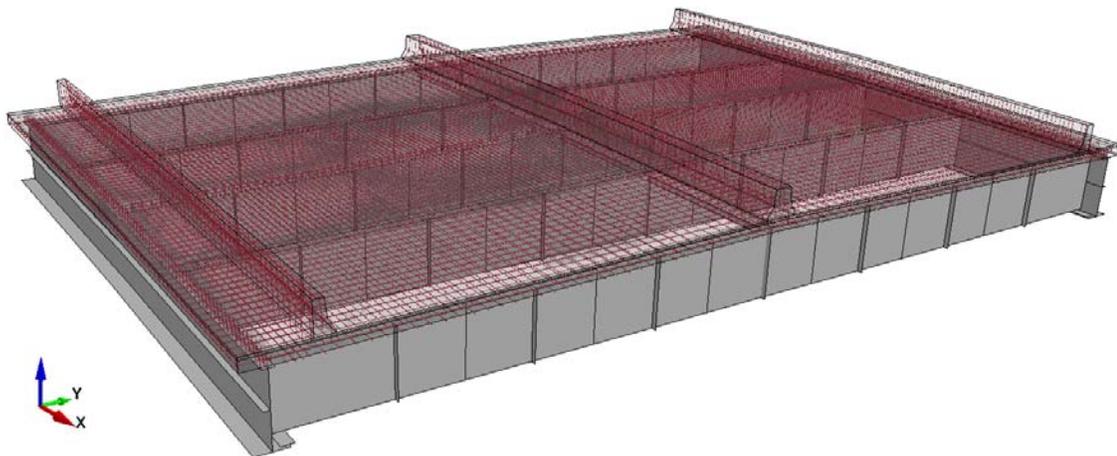

Figure 14: Detailed FE model of the central segment of the bridge

This time the deck was built out of seven layers of solid elements. Attempts were made to model the concrete with three different material models (MAT_072R3, MAT_084 and MAT_159). MAT_084, as pointed out earlier, was the most robust and was used here (see Table 7). Material properties were assumed for concrete with f'c = 46 MPa. Erosion of concrete solid elements was included

in the model by using MAT_ADD_EROSION (with maximum effective strain at failure = 0.1). To remedy the fact that with the eroded solids the pressure applied to them is also removed, the optional command LOAD_ERODING_PART_SET can be used to allow for propagation of the load through the eroded elements (available only for LOAD_BLAST, not for LOAD_BLAST_ENHANCED). The steel rebars were modeled using beam elements with MAT_024 (see Table 8). The steel in the girders and the floor beams was modeled using MAT_015. Assumed properties for that steel are shown in Table 9. The input used for this material is shown in Table 8. The reinforcement was embedded in the concrete elements using CONSTRAINED_LAGRANGE_IN_SOLID with constrained coupling type. The detailed model, shown in Figure 14, consisted of approximately 1,578,000 elements (1,452,000 solids, 50,000 shells and 76,000 beams).

Table 7: Simplified LS-DYNA input for MAT_084 in detailed simulation (units system kg, m, s, N, Pa)

ID	RO	TM	PR	UCS	UTS	FE	ASIZE
1	2275	3.2E+10	0.16	4.6E+7	3.937E+6	92.23	0.019
E	YS	EH	UELONG	RATE	CONM	CONL	CONT
0	0	0	0	0	1	1	1

Table 8: Simplified LS-DYNA input for MAT_024 (units system kg, m, s, N, Pa)

ID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
1	7850	2.100E+11	0.3	2.89E+8	0.0	0.15	1.0
C	P	LCSS	LCSR	VP			
40.4	5.0	1	0.0	0.0			

Table 9: Simplified LS-DYNA input for MAT_015 (units system kg, m, s, N, Pa)

ID	RO	G	E	PR	DTF	VP	RATEOP
1	7850	7.690E+10	2.100E+11	0.3			
A	B	N	C	M	TM	TR	EPSO
2.860E+8	5.0E+8	0.228	0.0171		1773.0	293.0	1
CP	PC	SPALL	IT	D1	D2	D3	D4
486.0							
D5	C2/P	EROD	EFMIN				

The MAT_ADD_EROSION command should be used cautiously. The erosion mechanism is not based on experimental testing of the material but rather on somewhat arbitrary selection of the erosion criteria that assures stability of the solution. Without it, the model terminates with errors indicating a high rate of straining and excessive plastic strains. With the erosion model, mass is vanishing from the system. The selection of the erosion criteria levels has also a great impact on the ultimate damage of the deck. It is of utmost importance to avoid selection of erosion criteria that in the case of erosion of one element triggers erosion of neighboring ones. Thus, with this approach a series of simulations should be performed testing the sensitivity of the model to the varying erosion criteria.

Due to the limitation of the length of the paper, only a couple of results are shown here for the detailed model simulations of Cases 4 and 6. As mentioned earlier the largest strains and damage of the deck occurred in time less than 0.05 s. Figure 15 and 16 show several time instances displaying the damage to the deck with cracks overlaid on the deck surface. The conducted analysis suggests that a car or a van size bomb will mostly cause damage to the deck and barriers and vibrations induced by the load are not likely to cause collapse of the bridge.
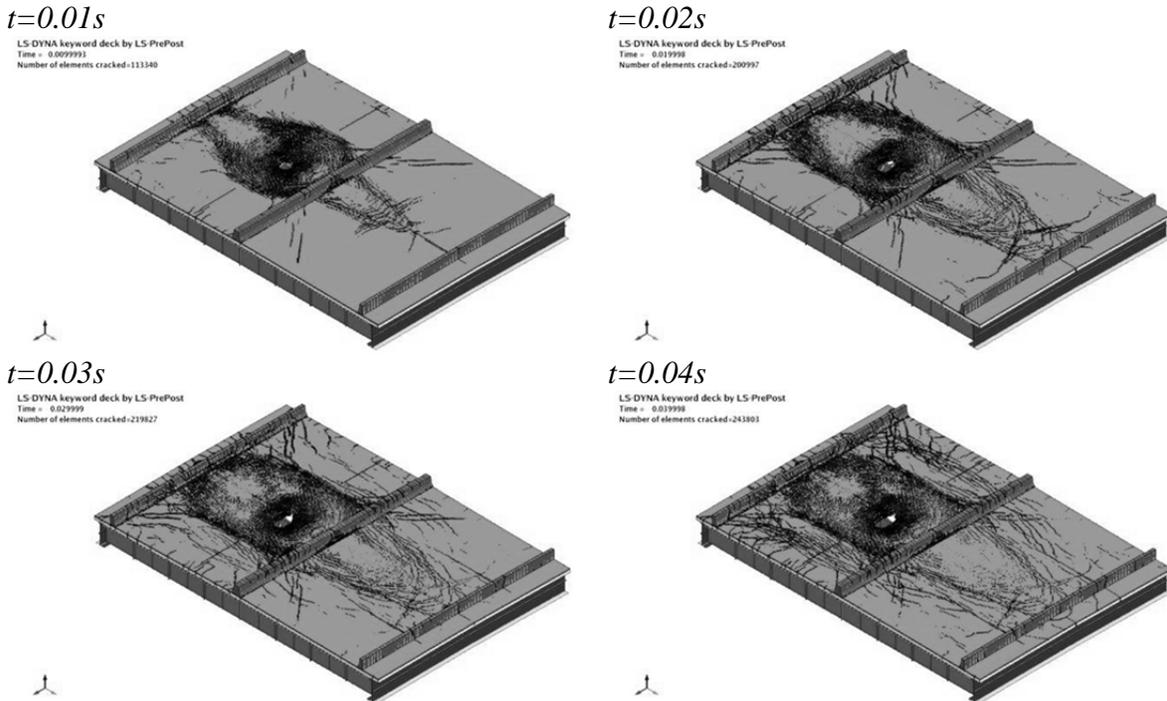
t=0.01s

t=0.02s

t=0.03s

t=0.04s

Figure 15: Damage and cracks in the deck detailed model in the analysis Case 4

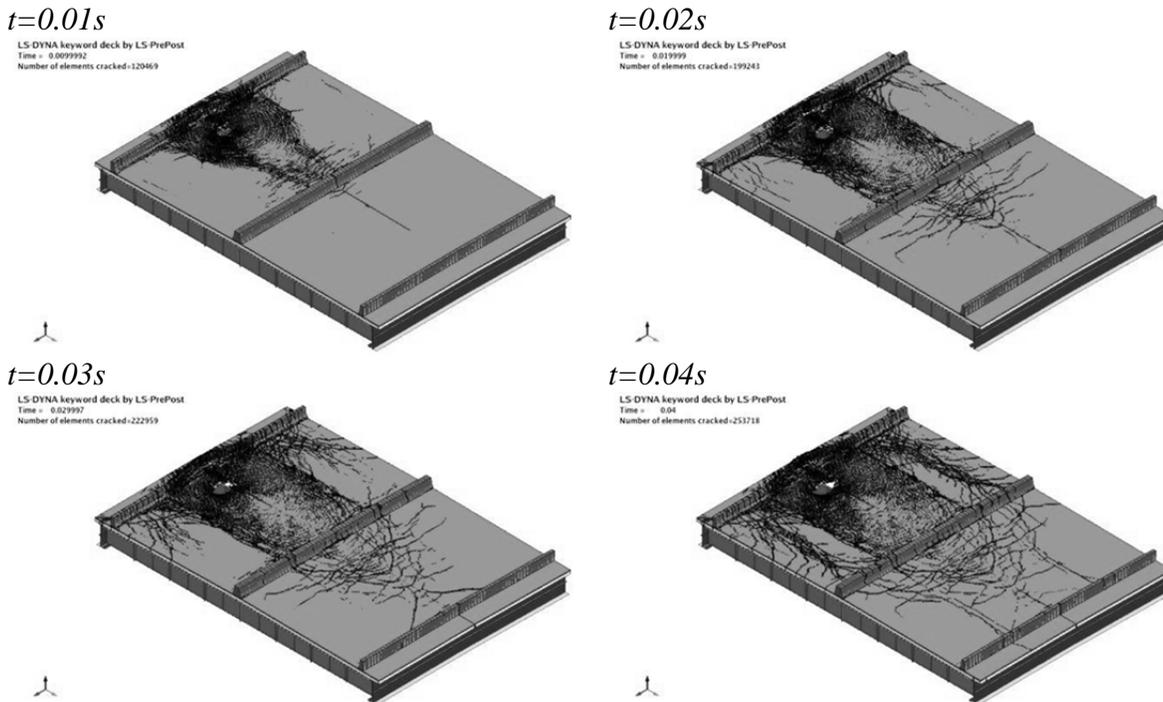*t=0.01s*

t=0.02s

t=0.03s

t=0.04s

Figure 16: Damage and cracks in the deck detailed model in the analysis Case 6

Summary

This paper describes a modeling process for performing blast loading simulations on a large span cable-stayed bridge. The simulation was split into three loading stages: (1) initialization and pre-loading, (2) blast loading on the global simplified bridge model, and (3) blast loading on the detailed model of bridge mid-span section. The paper focuses more on the detailed description of this simulation procedure rather than on specific results. The simplified model provided the description of the overall bridge response after the blast. The detailed model was aimed at estimating localized damage to the deck. Although the initial studies indicate that the bridge would probably survive the blast loading from a typical van bomb detonated on the deck, the detailed description of the deck damage is yet to be determined in a broader set of simulations with several mesh densities, different concrete constitutive materials and different erosion criteria for the concrete and steel. Also, the Arbitrary Lagrangian-Eulerian approach to modeling the blast wave should be used to further gain credibility in the results of the close range explosions.

Acknowledgements

References

[1]   Bojanowski C., Kulak R.F., "*Seismic and Traffic Load Modeling on Cable Stayed Bridge*", Paper No. 11-2553, 90-th Transportation Research Board Annual Meeting, Washington, D.C., January 23-27, 2011

[2]   Chen G., et al., "*Assessment of the Bill Emerson Memorial Cable-stayed Bridge Based on Seismic Instrumentation Data, Missouri Department of Transportation*", Report No. OR08.003, September, 2007

[3]   Hallquist J., "*LS-DYNA Keyword User's Manual Volume I*", Livermore Software Technology Corporation (LSTC), 2014

[4]   European Committee for Standardization "*Eurocode 2: Design of Concrete Structures: Part 1: General Rules and Rules for Buildings*", 1991

[5]   Comité Euro-International du Béton, "*CEB-FIP Model Code 1990*", Redwood Books, Trowbridge, Wiltshire, UK, 1993

[6]   Malvar L.J., Crawford J.E., "*Dynamic Increase Factors for Concrete*", Twenty-Eighth DDESB Seminar, Orlando, FL, 98

[7]   Razaqpur A.G., Tolba A., Contestabile E., "*Blast loading response of reinforced concrete panels reinforced with externally bonded GFRP laminates*", International Journal of Composites, Part B 38 (2007) 535-546