Simulation of crash tests for high impact levels of a new bridge safety barrier

Jiří Drozda¹ and Tomáš Rotter¹

¹ Czech Technical University in Prague, Faculty of Civil Engineering, Thákurova 7, Prague 6 166 29, Czech Republic

E-mail: jiri.drozda@fsv.cvut.cz, rotter@fsv.cvut.cz

Abstract. The purpose is to show the opportunity of a non-linear dynamic impact simulation and to explain the possibility of using finite element method (FEM) for developing new designs of safety barriers. The main challenge is to determine the means to create and validate the finite element (FE) model. The results of accurate impact simulations can help to reduce necessary costs for developing of a new safety barrier. The introductory part deals with the creation of the FE model, which includes the newly-designed safety barrier and focuses on the application of an experimental modal analysis (EMA). The FE model has been created in ANSYS Workbench and is formed from shell and solid elements. The experimental modal analysis, which was performed on a real pattern, was employed for measuring the modal frequencies and shapes. After performing the EMA, the FE mesh was calibrated after comparing the measured modal frequencies with the calculated ones. The last part describes the process of the numerical non-linear dynamic impact simulation in LS-DYNA. This simulation was validated after comparing the measured ASI index with the calculated ones. The aim of the study is to improve professional public knowledge about dynamic non-linear impact simulations. This should ideally lead to safer, more accurate and profitable designs.

1. Crash barriers
The basic function of the crash barriers is to redirect uncontrollable cars and minimize occupant’s injuries. Nowadays the impact severity, energy dissipation and stiffness of the new designs of crash barriers are evaluated with real crash tests. The requirements for these crash tests are prescribed in the European standard EN 1317 [1] and they have to be fulfilled to gain the certificate for using crash barriers in the European Union. The tests must be performed under different conditions, e.g. using passenger car and lorry, with various impact speeds and impact angles. To reduce the development and testing costs of new crash barrier designs, numerical analysis can be used for pre-evaluation of the crash barrier behavior.

2. Model validation
The process of model validation was described in detail in [2]. Basically validation in the context of roadside safety computer simulations is the process of determining the degree to which a roadside safety computer model is an accurate representation of real world crash tests from the perspective of accurately replicating EN 1317 crash test evaluation parameters, the structural performance of the barrier and the response of the vehicle. Validation always involves comparing a computer simulation or numerical experiment to a physical experiment of some type. The question at the root of the
Validation exercise is whether the simulation replicates the physical experiment and whether it can be used to explore and predict the response of a new or modified roadside hardware in the real world. Validation activities can be performed not only for full-scale crash tests but also for material models, sub-assemblies or components of a full FE model. Ideally, each portion of a large complex model should be validated separately if it is possible to perform a meaningful physical test. This work shows some possibilities to apply Numerical Modal Analysis (NMA) for model validation and mesh calibration. The application of NMA is shown on a prototype design of the bridge crash barrier post.

2.1. Numerical modal analysis
Modal analysis, or a free vibration analysis, is performed to obtain the natural frequencies and mode shapes of a structure. It is a subset of the general equation of motion. In the analysis, the behavior of the structure was assumed to be linear and the response to be harmonic. For solving the damping was not included. ANSYS - Workbench was chosen mainly because the verified FE model was used for the simulation of a crash test. The procedure of verification was based on comparing modal characteristics gained from the FE model and modal characteristics measured on the prototype. The goal of this comparison is to check the appropriate mesh density and to control the geometry 3D model. The first step was to create the 3D geometry, see Figure 1. This shell 3D model was created by using DesignModeler. Parts of the post were divided into subparts, to correspond to the distribution of locations, where excitation forces were applied in the real test. This step is important and necessary for future evaluation and verification of the FE model. It was essential to ensure mesh continuity across subparts of the same plate in order to preserve its real behavior.

The next step was defining material properties and boundary conditions. A mechanical mesh was used with hex dominate elements including mid side nodes. The size of elements was adapted to geometry. Only one boundary condition was defined at the place where the post is bolted to the concrete ledge. The connection was modeled by fixing all degrees of freedom at the edges of bolts holes. The Block Lanczos solver was used to compute the mode shapes.

2.2. Prototype design
A new design of the bridge crash barrier post was created in cooperation with the companies OK-BE and CTU. It consists of two parallel steel plates, which are fillet welded into base plates.

![Figure 1. a) Prototype post; b) Certified post; c) Mesh of prototype post; d) Mesh of certified post.](image-url)
The biggest advantage of this post is that there are no parts, where the dirt can cumulate and the whole construction of the crash barrier can be cleaned by a sweeper-flusher. The serviceability of the bridge crash barrier increases. Another challenge of the designer was to make the post lighter and easy to weld, which makes it more economical. The prototype post was assembled and attached into the massive concrete block by steel bonded anchors. For the comparison between the prototype and the certified post, see Figure 1.

2.3. The measurement

The single point testing was chosen for the experimental modal analysis. It is the most straightforward method of the phase separation techniques. Phase separation techniques rely upon a mathematical assumption that the actual responses are formed from a linear combination of the modes. The forced responses to a known excitation are measured and then the dynamic properties are extracted by means of mathematical curve fitting techniques.

An impact hammer was used to strike the structure at the excitation points. A very short sharp excitation pulse was produced – approximating a Dirac delta function – which has a flat spectrum over a wide frequency range. The amount of energy contained in the impact pulse is small.

![Figure 2. Mounted transducer: a) prototype; b) certified post; Excitation points: c) prototype; d) certified post.](image)

In Figure 2 the mesh of excitation points and the place where the transducer was located are visible. It is important to note, that the reference transducer location must not be in a nodal point of measured natural modes. If the ordinate of a natural mode in a reference point is too small, that natural mode and its corresponding natural frequency cannot be measured. This is one of the reasons for a preliminary NMA, in order to determine the optimal referent transducer placement.

Each excited point was struck 5 times by the impact hammer. Resultant readings were averaged for each point. Both signals, the excitation force and the acceleration, were recorded in the time domain and transformed using Fast Fourier Transform (FFT) to the frequency domain. The Frequency Response Function (FRF) was evaluated from these signals.
2.4. The comparison
The agreement between measured and computed natural mode shapes was evaluated according to MAC values, which were obtained using formula (1) from [3] and [4]:

\[
\text{MAC} \left( \{ \Phi_X \}_i, \{ \Phi_A \}_j \right) = \frac{\left| \{ \Phi_X \}_C \{ \Phi_A^* \}_M \right|^2}{\left| \{ \Phi_X \}_C \{ \Phi_A^* \}_C \right| \left| \{ \Phi_A \}_M \{ \Phi_A^* \}_M \right|}
\]  

(1)

In Table 1 the natural frequencies and the corresponding MAC values, according to Czech technical standards [4] are listed. MAC values near 1.0 refer to perfect match between the compared natural modes, while values near 0.0 refer to perfect orthogonality.

| Meas | Comp | 1  | 2  | 3  | 4  | 5  | 6  | 7  | 8  | 9  | 10 |
|------|------|----|----|----|----|----|----|----|----|----|----|
|      |      | 50.8| 154.0| 237.9| 251.1| 258.7| 275.0| 338.3| 413.4| 431.2| 556.7|
| 1    | 52.963| 0.975| 0.002| 0.045| 0.001| 0.007| 0.001| 0.072| 0.001| 0.036| 0.001|
| 2    | 168.320| 0.037| 0.896| 0.029| 0.066| 0.012| 0.014| 0.035| 0.000| 0.002| 0.018|
| 3    | 279.583| 0.007| 0.006| 0.007| 0.008| 0.913| 0.035| 0.017| 0.09| 0.000|
| 4    | 299.926| 0.010| 0.060| 0.014| 0.336| 0.017| 0.893| 0.025| 0.069| 0.007| 0.008|
| 5    | 331.327| 0.037| 0.153| 0.527| 0.081| 0.003| 0.079| 0.511| 0.013| 0.008| 0.032|
| 6    | 447.346| 0.006| 0.001| 0.005| 0.07| 0.035| 0.251| 0.002| 0.834| 0.078| 0.053|
| 7    | 465.166| 0.046| 0.001| 0.004| 0.005| 0.208| 0.008| 0.054| 0.021| 0.959| 0.004|
| 8    | 596.518| 0.002| 0.018| 0.004| 0.008| 0.006| 0.014| 0.032| 0.05| 0.045| 0.733|

Natural mode shapes were obtained from FRFs using polynomial curve fitting method. The comparison of the prototype’s first 4 measured and computed mode shapes is visible in Figure 3.

![Figure 3. Natural Mode Shapes: a) 1st mode; b) 2nd mode; c) 3rd mode; d) 4th mode.](image-url)

3. Finite element simulation of a crash test
The FE simulation of crash tests requires special consideration about modeling parts such as vehicles, vehicle restrain systems and contacts between them. In this part, the primary focus is on the modeling and the performing of dynamic analysis of a bridge safety barrier. Due to large dynamic deformations,
It is suitable to employ explicit time integration schemes like LS-DYNA (LSTC, 2002). The FE analysis consists of these main steps: creation of the geometry model and discretization, specification of the material constitutive law and element types for LS-DYNA, performing the calculation and evaluation of results. By the creation of the FE model it is necessary to keep in mind that the calculation time is highly dependent on the size of the mesh, which determines the minimum computational time step, the definition of the material’s failure criteria and the selection of an impact contact algorithm [5].

3.1. Vehicles models
Two crash scenarios according to EN 1317 were simulated. The first one, called TB 11, involves a small passenger car (900 kg). The geometrical details of this vehicle are similar to Ford Fiesta for which the FE model Geo Metro Passenger Sedan could be used. This model was developed through the process of reverse engineering at the National Crash Analysis Center (NCAC) of The George Washington University (GWU) [6], (http://www.ncac.gwu.edu) see Figure 4. It contains 193 200 elements. The model includes fully functional capabilities of the suspension and steering subsystems and the whole model was validated by a frontal wall impact test.

Figure 4. Model of passenger vehicle Geo Metro Passenger Sedan.

Figure 5. Model of lorry Rigid HGV.

The second simulated impact test is TB42 which covers an impact scenario with a small lorry (10000 kg). The model of this vehicle was developed by Politecnico de Milano (POMI). The geometry of this vehicle model is very similar to the geometry of the lorries which are used for full-scale tests in Europe. This model contains real models of tires, frontal and rare axel and also suspension and steering subsystems. The whole model contains 254 000 elements, see Figure. 4.

3.2. Safety barrier
The two FE models of bridge safety barrier used in this study are: the certified type ZSH4 [7] Figure 6, and the newly developed type Figure 7. Each model includes 30 m length of safety barrier. In both models solid elements have been used for the creation of the bridge ledge and shell elements have been used for steel sheets of safety barriers.

Figure 6. Certified type of vehicle parapet ZSH4.

Figure 7. Newly developed type of vehicle parapet.
The solid elements defined were of type 1, which is a sub-integrated element type with constant stress. With the proper setup of the hourglass stabilization algorithms, this element type has been proved to be very efficient and accurate for crash simulations. The shell elements of steel sheets were considered to be of type 16. This is a fully integrated element type, which delivers accurate results with the proper hourglass control definition. In the case of the newly developed type of bridge safety barrier the accurate modeling of bolts was achieved through beam elements. The beam elements were used due to, that the failure of bolts could be simulated. The connection between beam elements and other steel sheet parts was modeled using a contact algorithm. For this purpose, the bolt nuts and heads were modeled with shell elements, which were included in the contact definition between steel sheets and bolt nut or head.

4. Discussion of results and conclusion
A full-scale crash test was performed for a new design of a vehicle parapet. The guardrail was mounted on the standard concrete ledge with a distance of 1500 mm between posts. The full-scale crash test was performed for the scenario TB1 according to EN 1317. The results of this real crash test were compared with the results of the numerical simulation. Acceleration Severity Index (ASI), a sequence of plots and contact length was used for comparing the results and estimate how good is the representation of the real world crash tests. Firstly, the computational and experimental ASI index histories were compared as shown in Figure 8. The ASI index for the tested system does not exceed the limiting value in any case. The highest value (computational 1.5; measured 1.4), was observed when the guardrail hits the post. Secondly, the 100 km/h and 20 degree impact resulted in an acceptable performance as illustrated in Figure 9. The vehicle was redirected and the contact length was 5.92 m. A similar behaviour was noted in the finite element simulations as shown on the left side of Figure 9.

The usage of computational model provided comparable results to experimental measurements and can thus be used for the computational evaluation of other road safety barriers in order to avoid numerous expensive full-scale crash tests. The tests have also shown that the new safety barrier assures controllable crash energy absorption which in turn increases the safety of vehicle occupants.

![Figure 8. Comparison of ASI.](image-url)
Figure 9. Full-scale test and FE simulation of a 900-kg car.
Acknowledgements
This work was funded and administered by the Faculty of Civil Engineering, Czech Technical University in Prague (CTU). Additional funding was provided by Technology Agency of the Czech Republic (TAČR) within the projects TAČR no. 01031225 and TE01020168 - CESTI.

References
[1] European Committee for Standardization (CEN) 2010 Silniční záchytné systémy - Část 2: Svodidla - Funkční třídy, kritéria přijatelnosti nárazových zkoušek a zkušební metody /Road Restraint Systems – Part 2: Performance classes, impact test acceptance criteria and test methods for safety barriers including vehicle parapets/ (ČSN EN 1317-2)

[2] Ray MH, Plaxico CA and Anghileri M 2010 Procedures for verification and validation of computer simulations used for roadside safety applications Final Report for NCHRP Projekt 22-24

[3] Český normalizační onstitut /Czech Standards Institute/ 1984 Dynamické zkoušky stavebních konstrukcí /Dynamic tests of building structures/ (ČSN 73 2044)

[4] Český normalizační onstitut /Czech Standards Institute/ 2005 Zatěžovací zkoušky mostů /Load tests of bridges/ (ČSN 73 6209)

[5] Ulker MBC and Rahman MS 2008 Traffic barriers under vehicular impact: from computer simulation to design guidelines Comput-Aid Civ Infrastruct Eng 23 pp 465-80

[6] Opiela K, Kan S and Marzougui D 1997 Handbook of Development & Validation of a Finite Element Model for the 1997 Geo Metro Passenger Sedan

[7] Ministry of Transport of the Czech Republic 2011 Technické podmínky - Ocelové svodidlo ZSH2 (TP 230)