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

Jiří Drozda[1] and Tomáš Rotter[1]

[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.