Numerical simulation of crash test for bridge safety barrier

Jiri DROZDA
Civil Engineer
Faculty of Civil Engineering
Czech Technical University
in Prague
jiri.drozda@fsv.cvut.cz

Jiri Drozda, born 1985, received his civil engineering degree from the CTU in Prague in 2011, since then he studies for Ph.D. with focus on non-linear 3D modelling and equipment of bridges.

Jan MAREK
Civil Engineer
Faculty of Civil Engineering
Czech Technical University
in Prague
jan.marek.2@fsv.cvut.cz

Jan Marek, born 1986, received his civil engineering degree from the CTU in Prague in 2011, since then he studies for Ph.D. with focus on modal analysis and steel bridges.

Tomas ROTTER
Associate professor
Faculty of Civil Engineering
Czech Technical University
in Prague
rotter@fsv.cvut.cz

Tomas Rotter, habilitated at the CTU in Prague in 1987. His main area of research is steel bridges, fatigue, brittle fracture. He cooperated on many most important bridges in Czech rep.

Summary

The purpose of this paper is to examine the opportunity of non-linear dynamic impact simulation and to show the possibility of using finite element method (FEM) for developing new designs of safety barrier. Basic tasks of work are to determine a method that leads to creation of validated finite element (FE) model before performing the full scale crash and to perform non-linear dynamic impact simulation on this model in LS-DYNA. This method should help to improve accuracy of FE model that are used for non-linear dynamic impact simulation. Results of more accurate impact simulation will help to reduce necessary costs for developing new safety barrier.

Keywords: safety barrier, experimental modal analysis (EMA), verification of FE modelling, dynamic impact simulation.

1. Introduction

Basic tasks of the work are determination of a method that leads to creation of validated finite element (FE) model before performing the full scale crash and to perform non-linear dynamic impact simulation on this model in LS-DYNA. This method should help to improve accuracy of FE model that are used for non-linear dynamic impact simulation.

The first part of this article deals with the creation of FE model, which the newly-designed safety barrier was used in, and focuses on application of an experimental modal analysis (EMA). The FE model has been created in ANSYS Workbench and is formed from shell elements. Experimental modal analysis, which was performed on a real pattern, was used for measuring of modal frequencies and shapes. Afterwards perform of the EMA, the FE model is validated by comparing measured modal frequencies with calculated ones. Verification process was tested on newly designed steel bridge crash barrier. Therefore only steel crash barriers are discussed below. The steel bridge crash barriers consist of several major parts, which are guardrail, distance spacer, post (column), base plate and anchoring bolts. After all these parts are joined together, the bridge crash barrier acts upon its purpose.

The second part describes processes that lead to the performance of non-linear dynamic impact simulation on previously validated model. Dynamic impact simulation has been performed by using LS-DYNA explicit solver. Issues such as ensuring vehicle stability during impact and redirection, problem of proper accounting of friction effect and monitoring energy dissipation in whole system are described in this article.

Last part covers verification of simulation by comparison of results with other validated simulation.

https://doi.org/10.2749/222137814814027819