# Numerical modeling of damage for a tested composite steel-concrete specimen

**Marian Giżejowski**[1], **Lesław Kwaśniewski**[2], **Marcin Balcerzak**[3]*

[1]*Department of Building Structures, Warsaw University of Technology*
*al. Armii Ludowej 16, 00-637 Warsaw, Poland*
*e-mail: m.gizejowski@il.pw.edu.pl*

[2]*Department of Theoretical Mechanics and Pavement Modelling, Warsaw University of Technology*
*al. Armii Ludowej 16, 00-637 Warsaw, Poland*
*e-mail: l.kwasniewski@il.pw.edu.pl*

[3]*Research student at Faculty of Civil Engineering, Warsaw University of Technology*
*al. Armii Ludowej 16, 00-637 Warsaw, Poland*
*e-mail: marcin.balcerzak@wp.pl*

Abstract

This paper presents a finite element development and its verification and validation for a composite steel-concrete specimen subjected to push load up to failure. The considered specimen represents a segment of a typical floor setup, consisting of reinforced concrete slab cast on trapezoidal steel decking and supported by a steel frame subsystem of beams and columns with bolted end-plate beam-to-column joints. The test setup was designed to evaluate the joint behavior in steel-concrete, composite frameworks subjected to extreme loading generated by a rapidly removed column. The presented investigation is a part of the research on potential of progressive collapse of multistory buildings. The main objective of this study is to develop a reliable numerical model for the purpose of extensive parametric study. The virtual tests are performed using the nonlinear explicit dynamic code LS-DYNA® and well developed, verified and validated Finite Element models. The feasibility study is focused on identification of modeling parameters affecting the final results and on development of a plan for hierarchical verification and validation.

*Keywords: concrete, damage, failure, large deformations, numerical analysis, steel structures*

## 1. Introduction

In progressive collapse analysis of structures, the primary objective is to find the way of limiting the potential for disproportional damage and mitigate the effects of structural collapses. After several disastrous building catastrophes, concepts such as progressive collapse and robustness of structures have been reflected in many research investigations into a continuous improvement of design guidelines focused on taking advantage of connection redundancy and resilience, and catenary or compressive arching actions of the structural members. Different design approaches have been developed for reducing or eliminating disproportionate damage. There have been attempts to design stronger members to resist specific abnormal loads or to limit the total damage by an effective redistribution of loads through alternate load paths. For all approaches, the principal question is the extent of damage caused by a local failure initiated by an infrequent event.

Most of the published progressive collapse analyses for entire buildings or their components are based on the alternate load path method with a column removal. The capability of a structure to sustain local damage is evaluated by checking whether the local damage caused by column removal may be absorbed by the deteriorated structural system.

This paper presents a finite element development for a composite steel-concrete specimen subjected to push load up to failure. The considered specimen represents a segment of a typical floor setup, consisting of reinforced concrete slab cast on trapezoidal steel decking and supported by a steel frame subsystem of beams and columns with bolted end-plate beam-to-column joints. The test setup was designed to evaluate the joint behavior in steel-concrete, composite frameworks subjected to extreme loading generated by a removal of the internal column. The presented investigation is a part of the research on potential of progressive collapse of multistory buildings. The main objective of this study is to develop a reliable numerical model for the purpose of extensive parametric study. The virtual tests are performed using the nonlinear explicit dynamic code LS-DYNA® [1] and well developed, verified and validated Finite Element models. The feasibility study is focused on identification of modeling parameters affecting the final results.

## 2. Test setup

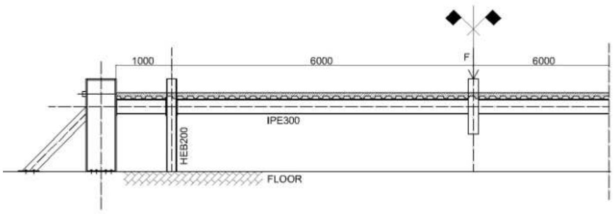


Figure 1: Scheme of investigated specimen

The specimen taken into consideration is a double-span composite frame subsystem presented in Fig.1, and tested experimentally at Rzeszów University of Technology. Steel part of this subsystem consists of HEB200 columns, IPE300 beams made of steel S235JR. Connections are executed using end-plates of thickness 10mm and 4 M20 bolts. The RC slab is made of C25/30 concrete on sheeting Cofraplus 60, with reinforcement of steel BST500S. The slab is connected to the upper flange of beams by shear studs. The width of the slab is

*The study reported in this paper was supported by a grant from the Polish Ministry of Science and Higher Education ‘‘Analysis of the stability and integrity of the multi-storey buildings subjected to exceptional actions’’, grant number 457/N-COST/2009/0.

variable and equals: 0.80m over the support and 1.10m in the middle of the span. At the end of the selected subsystem, beams are connected to a massive element by use of thick end-plates (24mm) and 4 bolts M20. Additionally reinforcement of the slab is fixed at this massive element.

## 3. FE model development

### *3.1. Space discretization*

In the presented in Fig. 2 the FE space discretization scheme, all beams and columns, as well as joint plates and stiffeners, are represented by 3D FE models built of shell elements. The bolts are represented by 1D beam elements. Additionally, there is a global contact defined among all the steel components represented by shell elements, e.g., flush end-plates and column flanges, and eventually shell elements representing the profiled sheeting and concrete of the composite floor slab. The contact is defined using an offset equal to half of the thickness for each interacting metal sheet.

Composite components, such as the reinforced concrete slab cast on the steel deck considered herein, are an additional challenge for numerical modelling. One difficulty is to efficiently merge responses of two materials into one representation. In the presented approach each material is represented separately; a concrete core with solid elements, a metal deck with shell elements, and reinforcement bars with line elements. For large-scale models, this strategy is infeasible today as it would result in a very large number of finite elements.

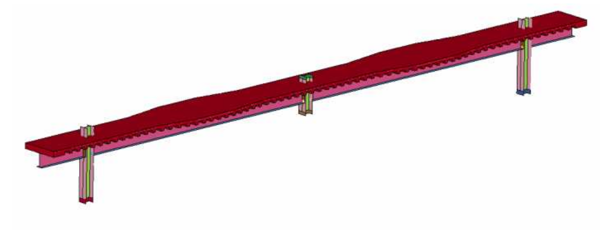

Figure 2: FE model of investigated specimen

### *3.2. Material models*

The collapse performance of composite systems depends strongly on the inelastic properties of the steel components. In the present study, a simple piecewise elastic–plastic model, with linear strain hardening, was applied with the characteristics obtained from the experimental coupon tests. The strain rate effects were neglected in this study.

A sophisticated material model with three invariant formulations for the failure surfaces, the so-called MAT 72R3 model from Ref. [2], was applied in this study for concrete. This material model has the complex description and requires many parameters as the input data however, its implementation in LS-DYNA® is equipped with an internal automatic data generation based on the extensive experimental investigations [2].

## 4. Verification and validation

The predictive capability of the nonlinear dynamic simulations is dependent on the inherent complexity of the method (e.g. contact and failure algorithms), taken assumptions (modeling simplifications) and uncertainties characterizing the input data (e.g. material parameters determining component failure). To improve the validity of the developed FE models and to identify the decisive model parameters a hierarchical verification and validation (V&V) procedure should be considered. While verification uses analytical or highly accurate numerical solutions, validation is based on the comparison of computational results with experimental data.

The main part of verification, which should precede validation, is the study on mesh resolution. Fig. 3 shows as an example of two load paths, for two mesh densities of steel elements in the investigated specimen. Fig. 4 illustrates the distribution of the scaled damage measure in the concrete slab, depicting potential crack pattern.

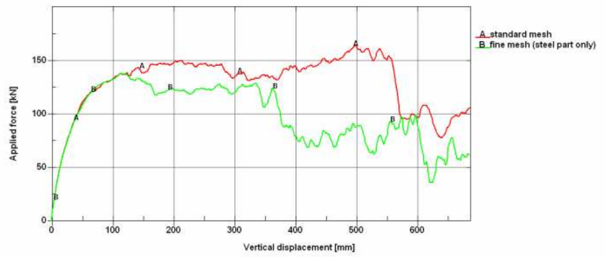


Figure 3: Load versus vertical displacement. Study on mesh resolution

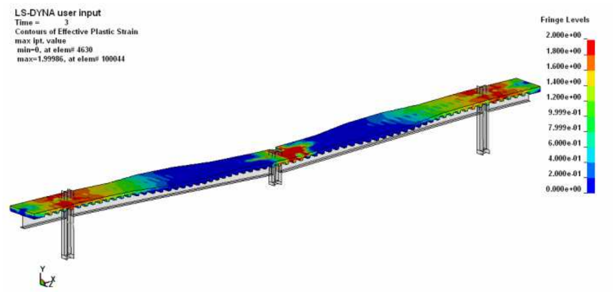


Figure 4: Concentrations of scaled damage measure in the composite slab depicting crack patterns

## 5. Summary

The validation procedure presented here shows that the simplified bolt's representation, in terms of 1D finite elements and the assumed failure strain, affects significantly the moment and rotation capacities of beam-to-column joints and in this way becomes one of the main modeling parameters affecting the global performance. Overall, the most important modeling parameters which can potentially influence the joint performance are mesh density, especially for the flush end-plates, failure strain for the bolts, and the contact algorithm. Since the collapse mechanism may cause locally reverse loading, the correct representation of unloading in the constitutive relationships defining material models is also important.

## References

[1] Hallquist, J.O., *LS-DYNA Keyword Manual*. Version 971. Livermore: Livermore Software Technology Corporation, 2007.

[2] Schwer, L.E., Malvar, L.J., *Simplified concrete modeling with *MAT_CONCRET_DAMAGE_REL3*, *LS-DYNA*, Anwenderforum, Bamberg, 2005.