

## FIRE ANALYSIS OF A SIMPLY SUPPORTED STEEL BEAM USING OPENSEES AND ANSYS WORKBENCH

Igor Džolev<sup>1</sup>

Aleksandra Radujković<sup>2</sup>

Meri Cvetkovska<sup>3</sup>

Đorđe Lađinović<sup>4</sup>

Vlastimir Radonjanin<sup>5</sup>

UDK: 624.042.5.072.2 : 519.6

DOI:10.14415/konferencijaGFS 2016.031

*Summary:* If experimental studies are difficult to conduct, the validation of results obtained using numerical procedures can be achieved through comparative analyses performed by different numerical model codes. In this paper, analysis of a simply supported steel beam subjected to fire is performed using open source software OpenSEES and a commercial software ANSYS Workbench. Both material and geometrical nonlinearities are taken into account. Calculation methodology is adopted according to Eurocode standards EN 1991-1-2 and EN 1993-1-2. Results, in terms of the midpoint displacements in time, are compared and discussed.

*Keywords:* OpenSees, Ansys Workbench, heat transfer, structural fire analysis

### 1. INTRODUCTION

Predicting the thermal-mechanical response of the structure subjected to fire is a highly nonlinear problem. Nonlinearity is reflected in the temperature-dependent thermal and mechanical properties of steel, which are influencing the temperature distribution during fire and the degradation of mechanical properties at elevated temperatures. In addition to material nonlinearity, geometric nonlinearity also needs to be considered, since large deformations due to thermal elongations combined with mechanical deformations are expected. A numerical approach, rather than analytical, is in expansion, following the development of computer systems. Specialized software, such as OpenSEES, SAFIR, VULCAN and commercial software ANSYS, ABAQUS have been used and are constantly being developed to simulate the behavior of structures in case of fire.

This paper presents a comparative analysis of a simply supported steel beam subjected to standard fire, conducted using open source software OpenSEES [1] and a commercial

<sup>1</sup> MSc, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: [ldzolev@uns.ac.rs](mailto:ldzolev@uns.ac.rs)

<sup>2</sup> PhD, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: [leksa@uns.ac.rs](mailto:leksa@uns.ac.rs)

<sup>3</sup> PhD, Faculty of Civil Engineering, University “Ss. Cyril and Methodius”, Skopje, Macedonia, e – mail: [cvetkovska@gf.ukim.edu.mk](mailto:cvetkovska@gf.ukim.edu.mk)

<sup>4</sup> PhD, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: [ladjin@uns.ac.rs](mailto:ladjin@uns.ac.rs)

<sup>5</sup> PhD, Faculty of Technical Sciences, University of Novi Sad, Serbia, e – mail: [radonv@uns.ac.rs](mailto:radonv@uns.ac.rs)

software ANSYS Workbench 16.0 [2]. The advanced calculation method is adopted according to Eurocode norms EN 1991-1-2 [3] and EN 1993-1-2 [4].

## 2. OPENSEES THERMAL (OS)

The Open System for Earthquake Engineering Simulation (OpenSEES) is a software framework for simulating the seismic response of structural and geotechnical systems. OpenSEES has been developed as the computational platform for research in performance based earthquake engineering in Pacific Earthquake Engineering Research Centre. This open source object-oriented software framework is focused on providing an advanced computation tool for analysing the non-linear response of structural frames subjected to seismic excitations. In 2009, professor Asif Usmani and his research team at the University of Edinburg have started to work on adding structures in fire modeling capacity in OpenSEES. New classes for defining temperature-time relations were added to the code, and existing classes were modified to account for the effect of thermal load. In current OpenSEES releases, calculation of thermal action is limited to thermo-mechanical analysis of planar (2D) frames. In [5], an extension to the open source software framework OpenSEES to accommodate with thermo-mechanical analysis of structure in fire is presented. The modifications in OpenSEES involve element, material and load classes as well as interface between them which are based on its existing class hierarchy. New thermal load classes were created to define the temperature distribution in structural members, existing material classes were modified to include temperature dependent properties according to Eurocodes, and new functions were also added into the existing beam-column element to apply temperature related loads. Figure 1 shows the new classes that have been added within the current OpenSEES framework, where the original OpenSEES classes are marked by the grey boxes. Figure 2 presents the calculation algorithm.

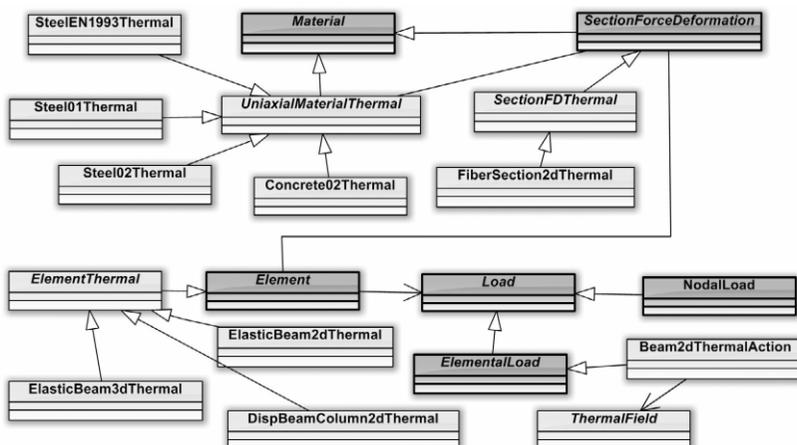


Fig. 1. New classes added to the OpenSEES for performing thermo-mechanical analyses [5]

New thermal load class Beam2dThermalAction was created to store the temperature distribution through the depth of the section defined by coordinates and corresponding temperature. Temperature of each fibre in the section will be determined by the interpolation of the temperature at the nearest coordinate point according to its location. Thermal action is defined based on 9, 5, or 2 temperature points and two parameters, temperatures ( $T_i$ ) and location ( $y_i$ ) which are assigned to each temperature point, Figure 3. Temperature could be constant or time dependent.

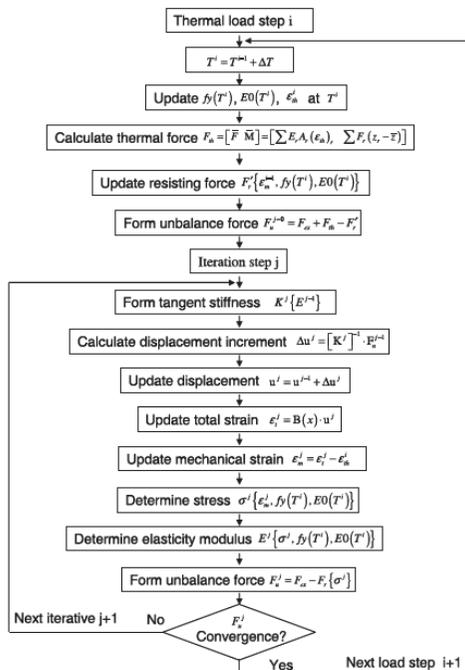


Fig. 2. Flow chart of thermal-mechanical analysis in OpenSees [5]

Temperature dependent seel material classes Steel01Thermal and Steel02Thermal were created based on existing steel classes Steel01 and Steel02. The yield stress and modulus of elasticity at elevated temperature were defined according to EN 1993-1-2 for carbon steel. Thermal elongation of steel fibre was calculated according to EN 1993-1-2, too.

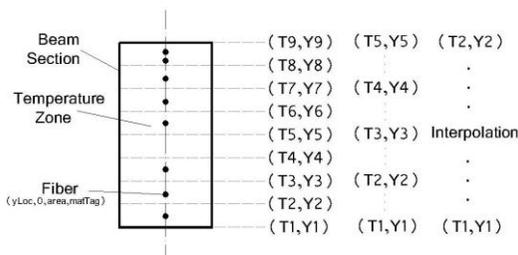


Figure 3. Thermal action defined in 9, 5 or 2 points and with two parameters [1]

The existing section class FiberSection2d was modified to FiberSection2dThermal by adding a new function to calculate the thermally induced section force. The existing stiffness-based beam element class DispBeamColumn2d was modified to DispBeamColumn2dThermal to include thermal force and material softening.

### 3. ANSYS WORKBENCH (WB)

ANSYS Workbench 16.0 is a commercial software package capable of performing various multi-physics analyses, based on a numerical finite element method. Fire analysis begins with a transient thermal analysis, resulting in the thermal response of the structure for the whole duration of fire. The calculated thermal profiles are then imported to the structural analysis model, as the thermal load at specified time points. In addition to the mechanical load, the thermal load is also influencing the mechanical behavior of the structure, through generated thermal strains and degradation of mechanical properties of steel at elevated temperatures. Coupling of the two systems is presented in Figure 4.

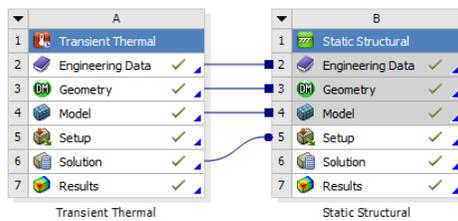


Fig. 4. Coupling of thermal and mechanical model in ANSYS Workbench

### 4. MATERIAL PROPERTIES

As described in the section above, OpenSEES has material classes defined according to EN 1993-1-2. ANSYS WB, as a multi-purpose software, offers the possibility for a material to be defined by the user, depending on the type of analysis to be performed, and the material properties necessary to conduct the desired analysis.

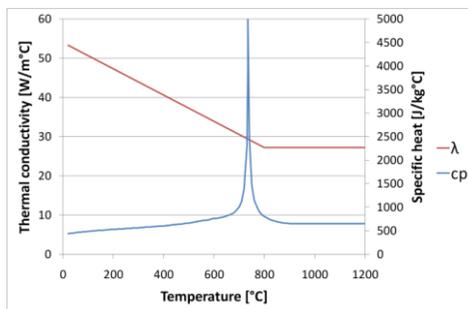


Fig. 5. Thermal conductivity and specific heat of steel

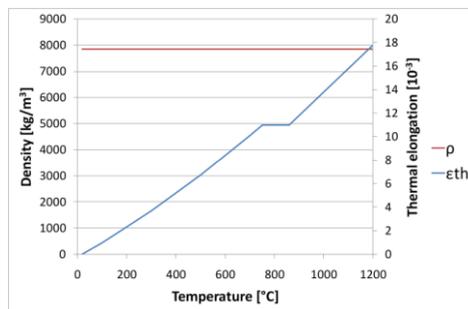


Fig. 6. Density and thermal elongation of steel

Heat transfer is governed by the material density, specific heat and thermal conductivity. Density of steel is considered constant for the entire range of temperatures that could occur during fire, while specific heat and thermal conductivity are temperature-dependent (Figure 5). Coefficient of thermal expansion varies with temperature, resulting in nonlinear thermal elongation (Figure 6). Stress-strain curves for steel at elevated temperatures are defined as multilinear with isotropic hardening plasticity, often used for large strain simulations (Figure 7).

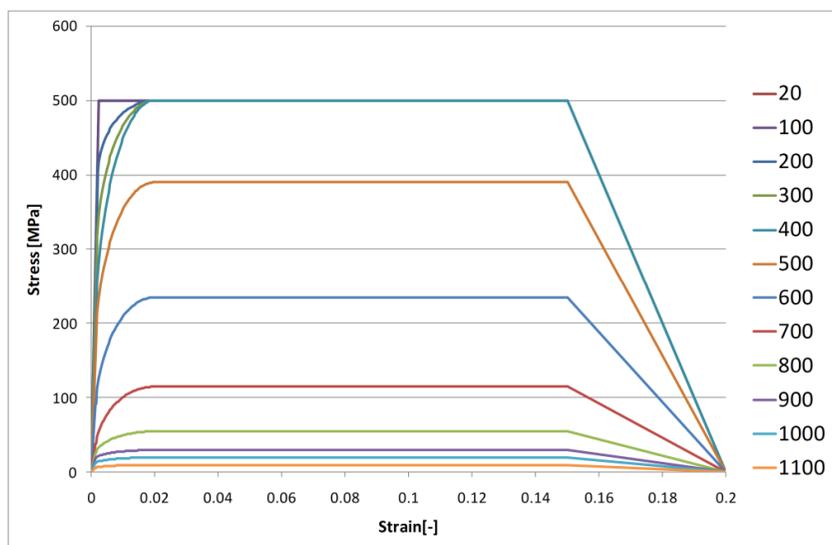


Fig. 7. Stress-strain relations of steel at elevated temperatures

## 5. ANALYSED EXAMPLE

A theoretical example of a simply supported steel beam (Figure 8) with a total length of 3 m and a full rectangular cross-section ( $b/h = 0.3/0.3$  m) subjected to a standard ISO 834 fire curve at the bottom surface and at the bottom and side surfaces, is analyzed (Figure 9). Before applying the thermal load, the beam is subjected to a constant concentrated load force of different magnitude, to produce the initial stress state. [6]

Thermal response is calculated using ANSYS WB and presented in Figure 10 and Figure 11 (based on the type of fire exposure). In the subsequent mechanical analysis, the temperatures are imported in each time step according to the obtained thermal response and the mechanical behavior is assessed. For the mechanical analysis performed using OpenSEES, temperatures are extracted in 30 time points (every 8 minutes) and applied to the model in 9 points across the height of the cross-section. Since the time step for the mechanical analysis is defined as  $\Delta t = 10$  s, linear interpolation of temperatures is used.

Displacement-time curves of the beam midpoint for the exposure to fire on one side and on the bottom and side surfaces are presented in Figure 12 and Figure 13, respectively.

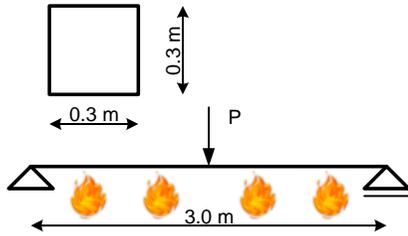


Fig. 8. Analyzed beam

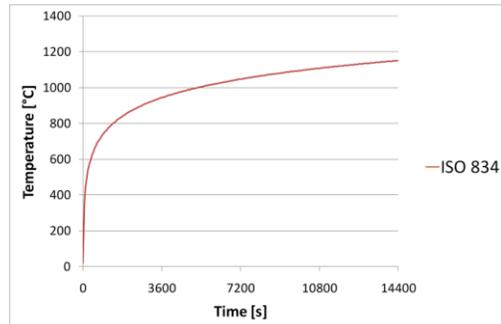


Fig. 9. Thermal load

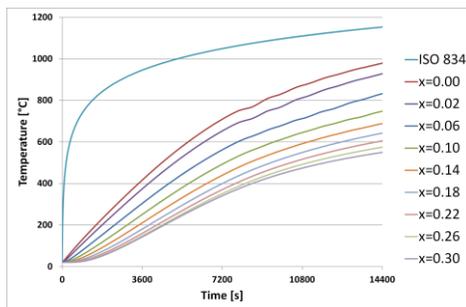


Fig. 10. Temperature profiles – exposure from 1 side

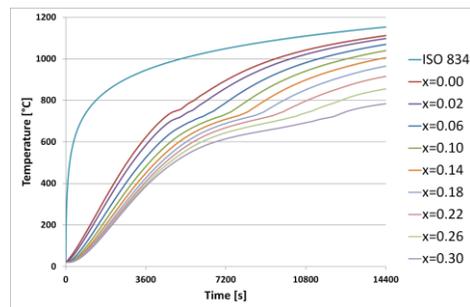


Fig. 11. Temperature profiles – exposure from 3 sides (centerline)

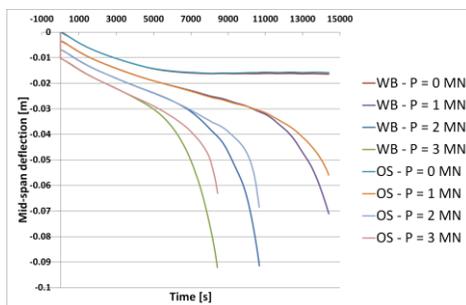


Fig. 12. Displacement-time curves – exposure from 1 side

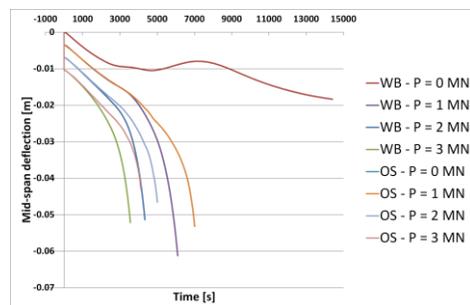


Fig. 13. Displacement-time curves – exposure from 3 sides

Failure is established when both of the following criteria are met [Ryan]: (1) deflection equal to  $L^2/800d$ , and (2) hourly deflection rate  $L^2/150d$ , where  $L$  is the span between supports and  $d$  is the distance between upper and lower extreme fibers of the element. Fire resistance for different types of fire exposure and initial mechanical load are presented in Figure 14.

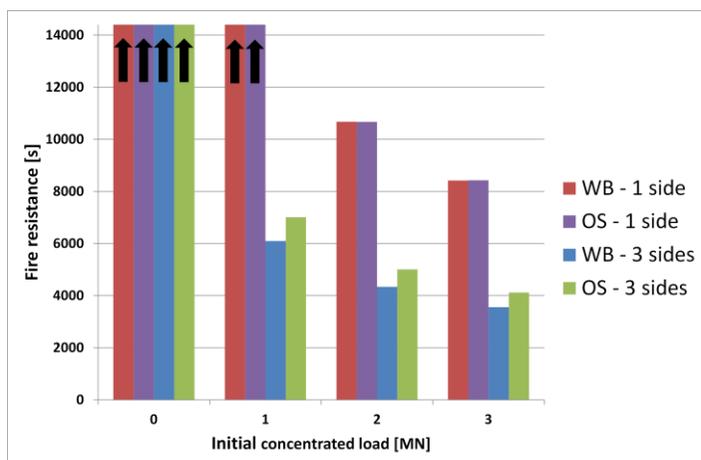


Fig. 14. Fire resistance of analyzed beams

## 6. DISCUSSION

Temperature profiles are calculated on a 3D model using ANSYS WB. If the beam is exposed to fire only on the bottom side, heat transfer occurs through the height of the section, while the temperatures through the width of the section remain constant. Since OpenSEES assumes constant temperature values through the width of the section, the imported temperature profiles used for the mechanical analysis are only valid for the fire exposure on the bottom side. Comparing the results of the beam midpoint deflection in time, it can be seen that smaller displacements are calculated using OpenSEES beam element than in the 3D analysis using ANSYS WB. In all cases analyzed, the hourly deflection rate was the one to determine the time at which failure occurs, which practically happens simultaneously in OpenSEES and ANSYS WB (Figure 14), for the fire exposure on one side. For the exposure on three sides, heat transfer also occurs through the width of the section, resulting in a faster heat penetration. This can also be observed from Figures 10 and 11 (temperature variations in time at the vertical axis of the element). In this case, the subsequent mechanical analysis in OpenSEES implements the degradation of mechanical properties only through the height of the section, resulting in a higher overall stiffness of the beam, which produce displacement results that are not on the safe side. The time at which failure occurs, calculated in OpenSEES are approximately 15% higher than fire resistance calculated in ANSYS WB.

## 7. CONCLUSION

Numerical analysis (if experimental tests are excluded) is generally used to determine the behavior of structures subjected to fire, since analytical solution to the nonlinear problem is unknown. Two models of a simply supported steel beam are developed using OpenSEES and ANSYS WB, and subjected to a constant mechanical load and standard

fire curve. Results, in terms of fire resistance, calculated based on the proposed failure criteria, match very well using both software, when the beam is subjected to fire from one side (as a 2D analysis). If the exposure is 3-dimensional, when using rectangular cross-sections, OpenSEES is unable to account for the temperature variation in two directions, but only through the height of the cross-section, which leads to a higher fire resistance than expected.

## ACKNOWLEDGEMENTS

The research has been conducted within the scientific research project TR 36043, funded by the Ministry of Science of Serbia. The authors would like to thank Faculty of Civil Engineering in Subotica, for the ceded license for ANSYS Academic product.

## REFERENCES

- [1] <https://www.wiki.ed.ac.uk/display/opensees/UoE+OpenSees> download, 01/25/2016
- [2] ANSYS® Academic Teaching Mechanical: *ANSYS Help Documentation, Release 16.0*, Canonsburg (PA), ANSYS, Inc., **2015**
- [3] EN 1991-1-2: *Actions on Structures, General Actions, Actions on Structures Exposed to Fire*, Brussels, European Committee for Standardization, **2002**
- [4] EN 1993-1-2: *Design of Steel Structures, General Rules, Structural Fire Design*, European Committee for Standardization, **2002**
- [5] Jiang, J.: *Nonlinear thermomechanical analysis of structures using OpenSees*, The University of Edinburgh, Edinburgh, **2012**
- [6] Džolev I., Cvetkovska M., Lađinović Đ., Radonjanin V.: *Load influence and fire exposure of a simply supported beam*, 16th International Symposium of MASE, Ohrid, Macedonia, **2015**, pp. 475-482

## POŽARNA ANALIZA ČELIČNE PROSTE GREDE KORIŠĆENJEM OPENSEES I ANSYS WORKBENCH

**Rezime:** *Ukoliko je teško sprovesti eksperimentalna istraživanja, verifikacija rezultata dobijenih korišćenjem numeričkih procedura može se ostvariti komparativnim analizama sprovedenim različitim kôdovima numeričkih modela. U ovom radu analizirana je čelična prosta greda izložena požarnom dejstvu, korišćenjem "open source" programa OpenSEES i komercijalnog programa ANSYS Workbench. Materijalna i geometrijska nelinearnost su uzete u obzir. Metodologija proračuna je usvojena prema Evrokod standardima EN 1991-1-2 i EN 1993-1-2. Rezultati, u smislu pomeranja središnje tačke u toku vremena, su upoređeni i diskutovani.*

**Ključne reči:** *OpenSees, Ansys Workbench, prenos toplote, požarna analiza*