The 9th European Congress on Computational Methods in Applied Sciences and Engineering
ECCOMAS Congress 2024
3–7 June 2024, Lisboa, Portugal

GENERALIZED PLASTICITY MODEL FOR NONLINEAR ANALYSIS OF STEEL FRAMES AT ELEVATED TEMPERATURES

MILAN BURSAĆ[1] and SVETLANA M. KOSTIĆ[1*]

[1] Faculty of Civil Engineering University of Belgrade
Bulevar kralja Aleksandra 73, 11000 Belgrade, Serbia
e-mail: bursacmilan4@gmail.com, svetlana@grf.bg.ac.rs

Key words: Inelastic analysis, Beam-column element, Plastic hinge, Resultant plasticity, Thermal analysis, Yield surface

Summary. The paper presents a nonlinear steel frame analysis at elevated temperatures, using a recently developed 3D beam/column element. The element belongs to the concentrated resultant plasticity type and is based on the relations of the Generalized Plasticity material model. It has two plastic hinges at the element ends and accounts for the interaction of the axial force and the bending moments about the principal axes of the cross-section and the gradual yielding of the cross-section. The nonlinear geometry under large displacements is taken into account with the corotational formulation. The element has shown high computational efficiency and numerical accuracy for modeling 3d steel frames under fire conditions.

1 INTRODUCTION

Steel frames, in general, offer numerous construction benefits, such as fast assembly, strong weight-to-strength ratio, and flexibility, and they are widely used. In order to avoid structural failure of frames, it is essential to evaluate and access their behaviour properly under elevated temperatures. In the past, numerous numerical models have been proposed with the aim of predicting the inelastic behaviour of steel frames under fire conditions. Most of these models use either 3D solid finite elements [4] that are highly accurate but computationally very expensive, or the frame finite element models that offer good compromise between numerical accuracy and computational efficiency [5, 6]. The element used in this study is recently developed frame element [2, 8] that belongs to the group of concentrated resultant plasticity elements. As depicted in Figure 1 the element is a serial model with two zero-length plastic hinges that may form at element ends, and a linear elastic component in between. This figure also shows the basic element forces (in the system without rigid body modes), q_1 to q_5.

The element adopts the relations of the Generalized Plasticity (GP) material model [7] for the relations between stress resultants (the axial force N, and two bending moments M_z and M_y about the principal axes) and the element end deformations. It accounts for the nonlinear geometry under large displacements with the corotational formulation and gradual yielding of the cross section. More details about the element formulation and parameter selection can be found in [2]. Here, just major steps of the analysis will be resumed.

The following two simplifying assumptions are adopted within the element formulation:

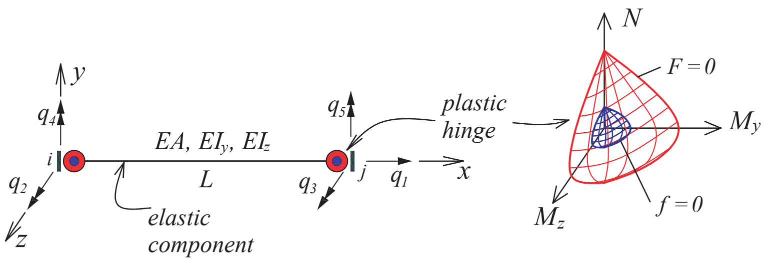

Figure 1: GP element

- The element properties are temperature dependent and are calculated from the average temperature of the cross section.
- The effects of shear forces and torsion on an element's capacity reduction are neglected.

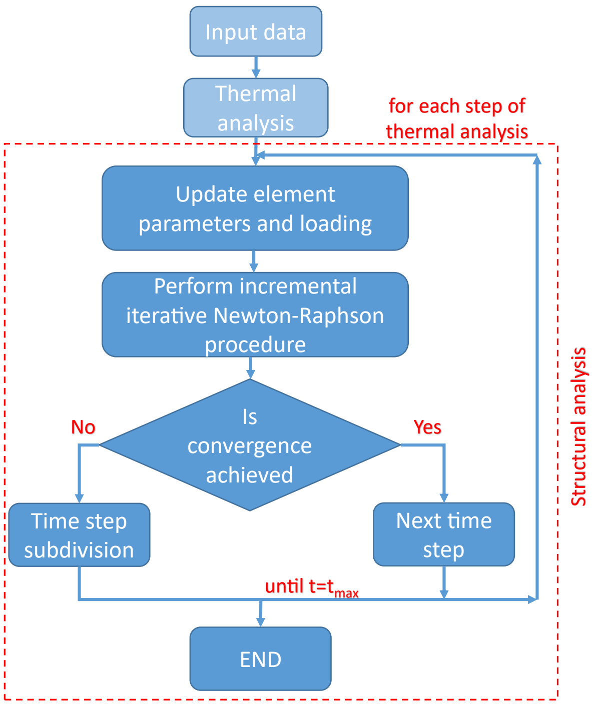

Figure 2: Flowchart of the thermo-structural analysis

The analysis is performed in Matlab in two steps. The first analysis step is thermal analysis which is performed using the Matlab solver for the heat analysis. The thermal properties of the material: thermal conductivity, specific heat, and mass density are defined as input parameters and, here, the values proposed by Eurocode 3 [9] are adopted. From this analysis, the temperature distributions over all beam/column cross-sections are found and the average section temperatures are calculated. In the second step, the structural analysis, is performed in FEDEASLab [10]. Firstly, the element properties and thermal loading are updated and, then, the structural (mechanical) analysis step is performed. The flowchart of the analysis procedure is depicted in Figure 2.

2 GP ELEMENT PARAMETERS

In order to use the GP element for the steel frame analysis under elevated temperatures, the following parameters need to be defined:

- the values for δ and β element parameters that control the approach rate of the force-deformation relation to the hardening asymptote, and the distance between the yield function and the asymptote, respectively
- the parameters of the elastic component,
- the temperature dependent yield surface equation for I shaped cross-section.

In this study, the selection and calibration of the element parameters from the first two groups specified above are the same as in [2]. At the same time, a slightly different expression is used for the yield surface equation. Without hardening, the yield surface for each hinge, f, becomes equal to the function Φ [2, 8]. This function is adopted in the following unitless polynomial form:

$$\Phi(p, m_z, m_y) = 1.2p^2 + m_z^2 + |m_y|^5 + 3.5p^2 m_z^2 + \frac{6A_f}{A_f + A_w}|p|^3|m_y|^3 + 3m_z^2 m_y^2 - c \qquad (1)$$

The symbol A_f in Eq. 1 represents the area of both flanges of the cross-section, A_w is the area of the web, while the variables p, m_z and m_y are given by:

$$p = \frac{q_1}{k_{y,\Theta} N_p}, \quad m_z = \frac{q_2}{k_{y,\Theta} M_{pz}} \text{ or } \frac{q_3}{k_{y,\Theta} M_{pz}}, \quad m_y = \frac{q_4}{k_{y,\Theta} M_{py}} \text{ or } \frac{q_5}{k_{y,\Theta} M_{py}} \qquad (2)$$

where N_p is the axial capacity, M_z plastic moment capacity around the strong axis, and M_y plastic moment capacity around the weak axis. The thermal coefficient $k_{y,\Theta}$ represents the ratio of the steel yield stress $f_{y,\Theta}$ at temperature Θ and the steel yield stress f_y at the reference temperature of 20°, i.e. $k_{y,\Theta} = \frac{f_{y,\Theta}}{f_y}$. This equation, for the case when the coefficient $c = 1$ reduces to the expression recently proposed by Singh [1] that outperforms the commonly used equation proposed by Orbison et al. [11]. The variation of coefficient c with temperature is adopted as in [2].

3 NUMERICAL EXAMPLE

To validate the accuracy and capabilities of the GP model, a numerical study was conducted on a two-story two-bay plane frame. The frame is exposed to fire on the second story in one bay, and all exposed members are considered to be heated from all four sides. The geometry of the frame, loads, material characteristics, and members exposed to fire are depicted in Fig. 3. The frame was previously analyzed by Luu et al. [3]. For the validation of their proposed calculation model, the analysis was also conducted in Abaqus and those results are considered as accurate. In this analysis, all members are discretized with a single GP element.

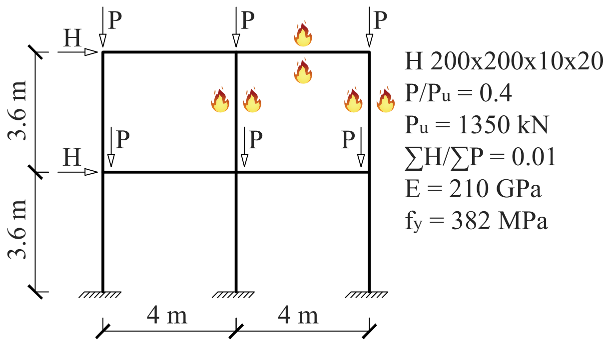

Figure 3: Two story frame

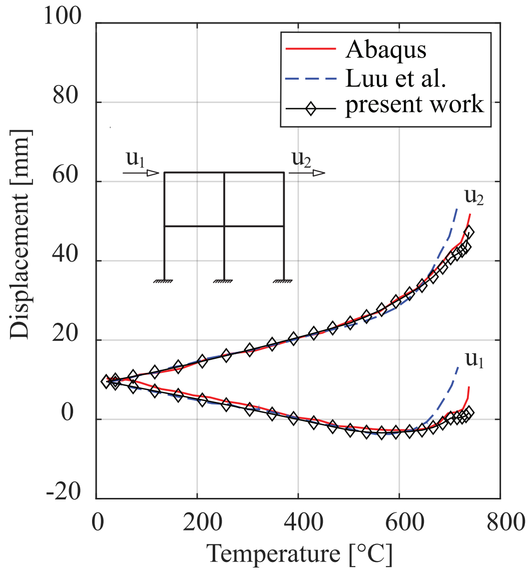

Figure 4: Comparison between numerical results for the two-story frame

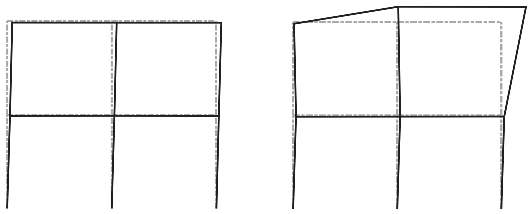

Figure 5: Frame deformations at different temperatures

Figure 4 shows the relation between displacements at two selected points at the top of the frame vs temperature. As can be seen, there is excellent agreement between the results obtained by the GP model and the values calculated by the more complex model from Abaqus.

Figure 5 shows twenty times scaled deformations of the frame for the room temperature and for the temperature at which failure occurs. The displacements of selected points (depicted in Figure 4) are equal at room temperature, as expected due to uniform geometry and loading, while these displacements, during heating, become different because of the expansion of elements subjected to the fire conditions.

Number of other validation examples for the proposed GP element can be found in [2].

4 CONCLUSIONS

The paper uses a concentrated plasticity beam-column element for the 3D analysis of steel frames subjected to fire exposure, whose formulation is based on the Generalised Plasticity material model. The element includes the effects of material and geometrical nonlinearities, cross-section gradual plastification, thermal expansion, and strength degradation due to fire exposure. The strength degradation is introduced in a simplified way, based on the average temperature of the cross-section, via the reduction coefficients proposed in Eurocode 3. Due to the simplicity and robustness of the formulation, the element is computationally efficient and offers an excellent compromise between the result accuracy and the required time for the analysis. The correlation studies have proved the element's capabilities to successfully predict the inelastic behaviour of steel frames exposed to fire.

5 ACKNOWLEDGEMENT

The first author thanks Ministry of Science, Technological Development and Innovation of the Republic of Serbia for financial support via a PhD student stipend. The second author thanks Ministry of Science, Technological Development and Innovation, Republic of Serbia, for financial support under the Project number 200092.

REFERENCES

[1] Singh, A. 2020. Performance Based Analysis of Steel Structures using Concentrated Plasticity Frame Element. Master thesis. University of California, Berkeley

[2] Bursac, M. and Kostic S. M. 2024. An Efficient Model for 3D Analysis of Steel Frames Exposed to Fire. Computers and Structures, Volume 297. https://doi.org/10.1016/j.compstruc.2024.107363

[3] Luu, V.-T. and Kim, S.-E. 2023. An advanced beam-column element for analysis of frames in fires. International Journal of Mechanical Sciences 260. https://doi.org/10.1016/j.ijmecsci.2023.108650

[4] Franssen, J.-M. and Gernay, T. 2017. Modeling structures in fire with SAFIR: theoretical background and capabilities. Journal of Structural Fire Engineering 8(3). https://doi.org/10.1108/JSFE-07-2016-0010

[5] Iu, C.K. and Chan, S.L. 2004. A simulation-based large deflection and inelastic analysis of steel frames under fire. J Constr Steel Res 60(10). https://doi.org/10.1016/J.JCSR.2004.03.002

[6] Barros, R.C., Pires, D., Silveira, R.A., Lemes, I.J. and Rocha, P.A. 2018. Advanced inelastic analysis of steel structures at elevated temperatures by scm/rphm coupling. J Constr Steel Res 145. https://doi.org/10.1016/J.JCSR.2018.03.001

[7] Lubliner, J., Taylor, R.L. and Auricchio, F. 1993. A new model of generalized plasticity and its numerical implementation. Int J Solids Struct 30(22)

[8] Kostic, S.M., Filippou, F.C. and Deretic-Stojanovic, B. 2016. Generalized plasticity model for inelastic rcft column response. Comput Struct 168. https://doi.org/10.1016/J.COMPSTRUC.2016.02.006

[9] E. C. for Standardization. 2005. Eurocode 3: design of steel structures - Part 1-2: General rules - Structural fire design

[10] Filippou, F. and Constantinides, M. 2004. Fedeaslab getting started guide and simulation examples. Technical report. Berkeley: University of California

[11] Orbison, J.G., McGuire, W. and Abel, J.F. 1982. Yield surface applications in nonlinear steel frame analysis. Comput Methods Appl Mech Eng 33(1–3). https://doi.org/10.1016/0045-7825(82)90122-0