Centrum
Rzeczoznawstwa
Budowlanego

Inżynieria Bezpieczeństwa
Obiektów Antropogenicznych 4 (2022) 42-53
https://inzynieriabezpieczenstwa.com.pl/

Inżynieria Bezpieczeństwa
Obiektów Antropogenicznych

Numerical analysis of a steel frame in a fire situation

Michał SZCZECINA*[1], Katarzyna NOWAK[1]

[1] Kielce University of Technology, Kielce, Poland

Abstract

In the paper, the stages of modelling of a steel frame under fire conditions were discussed. Numerical computations were performed using SIMULIA Abaqus software. The way of defining and developing the computational model was described. In the numerical procedure adopted in the paper, much attention was paid to formulas related to various types of the model parameters that depend on the fire temperature. The results were shown in a graphic form. They were compared with the results obtained from the analysis, in which the impact of elevated temperatures was disregarded. Based on numerical simulation results, an attempt was made to assess the effect of fire on the steel frame components. The paper is exploratory in character and provides an introduction to the issues related to the modelling of steel structure fires. The numerical modelling of the structure fire described in the paper can have practical potential for structural engineers.

Keywords: fire, steel frame, modelling, numerical analysis.

1 Introduction

The impact of elevated temperatures mainly depends on the type of material the structure is made from. As regards steel components, elevated temperature results in considerable changes in material properties. Steel shows high thermal conductivity, which directly translates into low fire resistance. Under elevated temperatures, steel yield strength, tensile strength and also Young's modulus of steel decrease (Gewain and Iwankiw, 2003; Kodur et. al., 2010).

As a result, the action of high temperatures can pose a real hazard to the safety of the whole system, because it leads to reduced load-bearing capacity, or even failure of steel elements. Consequently, it is crucial to understand the complex behavior of the steel structure under fire conditions already at the beginning of structure design, namely at the computational analysis stage (Lausova et. al., 2015; Pantous et. al., 2022, Maślak et. al., 2021; Snela, 2017).

Structure computations with respect to fire conditions must account not only for the type material of the structure individual components, but also for appropriate identification of other fire-affecting determinants. The complexity of the fire pattern, including the rate of temperature increment, is also related, among others, to the structure geometry, ventilation conditions, insulation used, and also the furniture, fixtures and equipment of the building. (Chudyba, 2019; Tkaczyk, 2017). In a fire, the following development stages of can be distinguished:

- fire outbreak - fire usually starts locally and gases rise and spread in the near-ceiling zone as they combust,
- fire development – is characterized by a gradual increase in temperature,

---

* **Corresponding author:** E-mail address: (m.szczecina@tu.kielce.pl) Michał SZCZECINA

https://dx.doi.org/10.37105/iboa.157
Received 15 December 2022
Available online 31 December 2022
ISSN 2450-1859, eISSN 2450-8721
Published by Centrum Rzeczoznawstwa Budowlanego

- fully developed fire - a sudden outbreak of fire throughout the whole room at temperature ranging 500 - 550°C, which continues to develop until the combustible material is consumed
- post-combustion - temperature decreases due to consumption of combustible material,
- extinction and cooling.

It should be noted that the effect produced on the structure by elevated temperatures also depends on the structural design, the actual pattern of temperature changes and the fire duration. In fire engineering, the assessment of real usability and structural reliability after a fire is a complex task. For example, in steel structures, when elements are cooled down following a fire, mechanical properties of the material are partially restored. In concrete, however, additional damages can occur (Chudyba, 2019). Additionally, precise calculations must account for the impact of elevated temperatures on the connections and joints between structural elements. That is crucial because joints of individual steel components undergo considerable strain and deformation in the heating and cooling stages. Under fire conditions, that can lead to a change in the overall joint stiffness, resulting in the failure of the elements being connected and, ultimately of the whole structure (Tkaczyk, 2017).

General guidance as how to determine actions under fire conditions are provided in the standard PN-EN 1991-1-2. Such actions are classified as specific actions. They involve a combination of actions where loads found in ordinary actions and those possible to occur under fire conditions are taken into account. In the calculations, however, cumulative occurrence of accidental actions that are different in character is not considered. Additionally, the standards recommend that the so-called 'fire scenarios' should be taken into consideration when defining an accidental design situation. Under those scenarios, a fire occurs in only one fire cell of the building at a time.

When carrying out the thermal analysis of the structure, it is also necessary to determine gas temperature in combustion. To this end, a calculation procedure using a nominal temperature-time curve, or a fire model can be applied. As regards nominal curves, the temperature-time standard curve, the external fire curve and the hydrocarbon fire curve can be implemented. In design practice, the standard curve is the one most commonly used. It is expressed by a strictly monotonic increasing function of the fire duration, and is applicable to a fully developed fire (Standard PN-EN 1991-1-2; Chudyba, 2019).

One of the basic tasks of a structural engineer is to perform a numerical analysis that accounts for the structure performance under relatively real conditions. The thermal analysis is intended to ensure the safety of people and property. That is achieved by reducing the risk of fire, and by appropriately estimating the structure fire resistance. Numerical solutions that are applied need to predict the development of a potential fire. The prediction must take into account fire protection features (Biegus, 2013; Tkaczyk, 2017). In accordance with the Standard EN 1991-1-2), structural reliability strategy of built features with respect to the design and execution for fire conditions assumes the following:

- maintaining the structure load-bearing capacity for a specific period of time,
- limiting the occurrence and spread of fire and smoke,
- restricting the spread of fire to neighboring structures,
- making it possible for the users to leave the building,
- ensuring the safety of emergency services.

Summing up, structural engineers are required to ensure fire resistance sufficient to prevent premature failure of the structure due to elevated temperatures.

The Standard PN-EN 1993-1-2 allows the application of computational methods that are based on simple computational models, advanced computational models and experimental investigations. In engineering practice, however, the first option is chosen by structural engineers as it is the simplest and the cheapest one. This method involves the analysis of individual structural elements, and the adoption of conservative simplifying assumptions. Thus, the question arises whether the structural engineer can develop, in a relatively easy manner, an advanced computational model that is compliant with fire engineering principles and the standard guidelines (Standard PN-EN 1993-1-2). This paper makes an attempt at providing an answer to the question posed above.

2 Problem definition

In this paper, numerical simulation in SIMULIA Abaqus for a steel structure under fire conditions is discussed. A three-bay orthogonal plane frame was considered. The frame diagram, the means of support, and dimensions are shown in Fig.1. The columns and beams were made of the same steel, beams had different cross-sections (Fig.1).

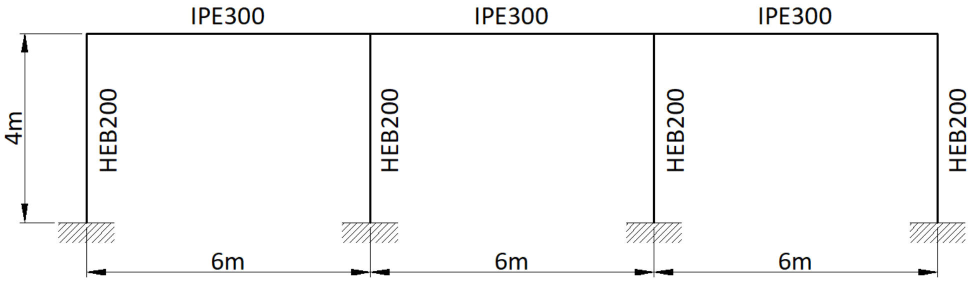


Figure 1. Geometry of the steel frame.

The frame was subjected to elevated temperatures in accordance with the standard curve available from Standard ISO 834-11:2014, and the Standard PN-EN 1991-1-2:

$$\theta_g = 20 + 345 log_{10} e^{(8t+1)} \tag{1}$$

where:

θ_g – temperature of combustion gases [^{0}C],

t – time [min].

The temperature distribution above (1) is widely used in thermal analysis. Based on it, the temperature-time relationship (Fig. 2) was determined. It should be noted that the standard curve (Fig. 2) does not account for extinction and component cooling stages after the fire. In the considerations below, it was assumed that the beams are elements directly exposed to fire, i.e. heated in fire (Fig. 2). Two fire occurrence scenarios were considered in the analysis. (Fig.3):

- variant 1 – fire is located in the left bay,
- variant 2 – fire is located in the central bay.

It was assumed that the structure does not have fire insulation.

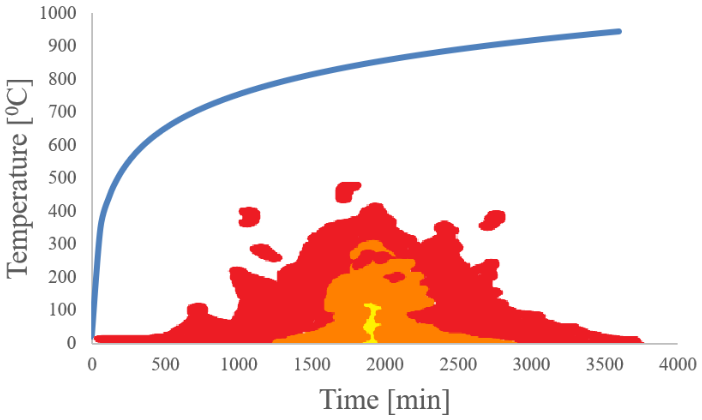


Figure 2. Time-dependent temperatures according to the ISO 834-1995 standard fire.

The configuration of the external loads on the frame and the fire location variant, are shown in the diagram in Fig.3.

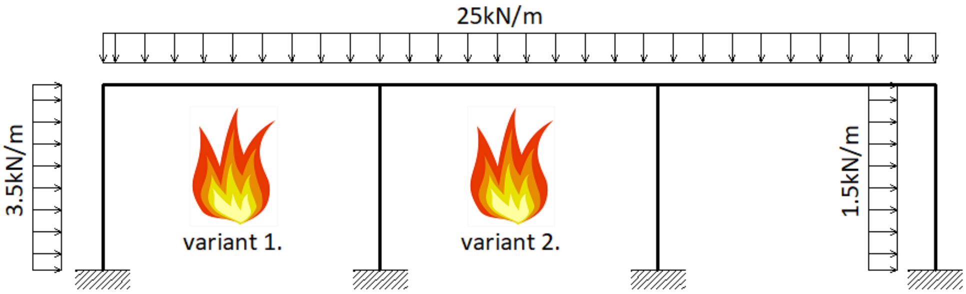


Figure 3. Loading and two variants of fire location.

The frame numerical model and its behavior under fire conditions were implemented into SIMULIA Abaqus software. Different uncoupled and coupled finite elements can be included in the thermal behavior simulation. In Abaqus, it is possible to adopt fully coupled thermal-stress analysis. The problems can be transient or steady-state, and linear or nonlinear. In coupled thermal-stress analysis, heat transfer equations are integrated using a backward-difference scheme, and the coupled system is solved using the Newton's method (Abaqus material library, 2017):

$$\begin{bmatrix} K_{uu} & K_{u\theta} \\ K_{\theta u} & K_{\theta\theta} \end{bmatrix} \begin{bmatrix} \Delta u \\ \Delta\theta \end{bmatrix} = \begin{bmatrix} R_u \\ R_\theta \end{bmatrix} \tag{2}$$

where:

K_{ij} – submatrices of the fully coupled Jacobian matrix,

$\Delta u, \Delta\theta$ – corrections to the incremental displacement and temperature,

R_u, R_θ – mechanical and thermal residual vectors.

A coupled temperature-displacement four-node shell finite element was used in the analysis. The heat conduction equation in 3D solid element can be written in the form:

- the Fourier law:

$$\boldsymbol{q} = -\boldsymbol{D}\,\boldsymbol{\nabla T} \tag{3}$$

where:

$\mathbf{q} = \{\mathbf{q_x}\ \ \mathbf{q_y}\ \ \mathbf{q_z}\}$ – heat flux vector,

D – heat conductivity matrix,

$\mathbf{\nabla T} = \left\{\frac{\partial \mathbf{T}}{\partial \mathbf{x}}\ \ \frac{\partial \mathbf{T}}{\partial \mathbf{y}}\ \ \frac{\partial \mathbf{T}}{\partial \mathbf{z}}\right\}$ – vector of temperature gradient.

- heat conduction equation (Poisson’s equation) – isotropic material, therefore **D** = ***k*****I**:

$$\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} = -\frac{f}{k} \tag{4}$$

where:

f – body heat flux,

k – thermal conductivity of material,

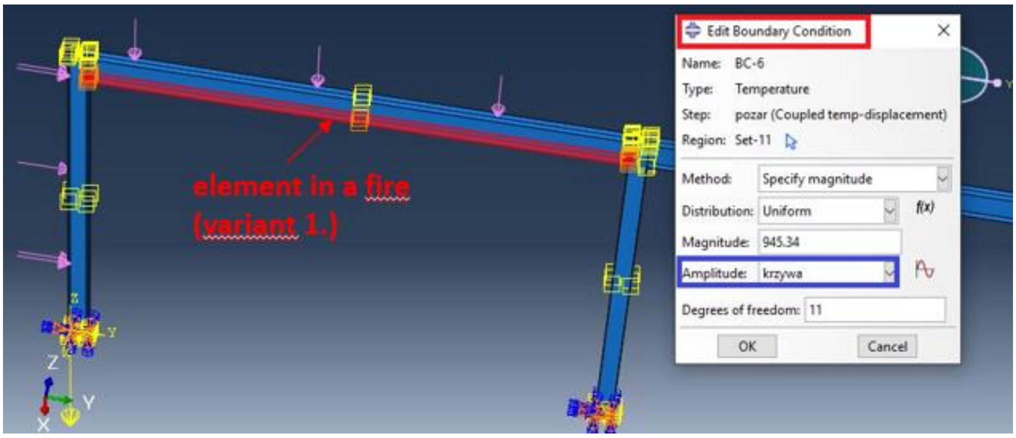
I – identity matrix.

3 Numerical analysis in simulia abaqus

Thermal analysis performed using FEM-based software makes it possible to take into account many geometric and materials parameters of the structure, heat transfer parameters, different temperature distributions and fire scenarios. The accuracy of numerical computations largely depends on input data provided by the structural engineer.

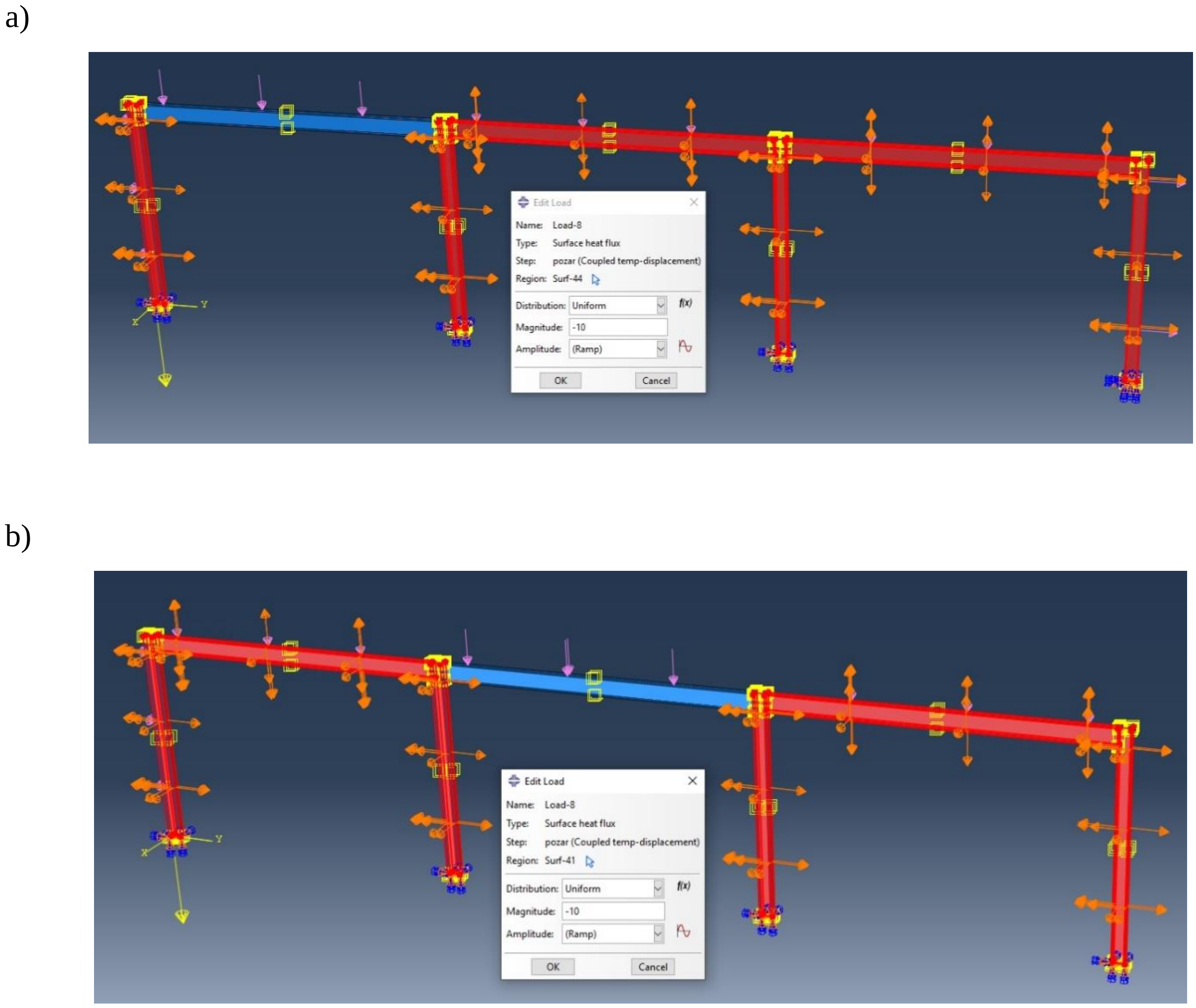
This paper discusses the application of SIMULIA Abaqus software for numerical simulation of structure performance under fire conditions. As the modelled phenomenon is complex in character, it was necessary to explain how the computational model was defined and developed. At the input stage, special attention was given to the way elevated temperatures were taken into account.

The frame was modelled using 4-node shell elements (S4RT) with the mesh size of 5 [cm]. The structure was subjected to external loads of the following type: Pressure (Fig. 3). The heat flux density was set as Body heat flux type measuring 45 [W/m^3] (Mika and Słoński, 2006). Then, the boundary conditions were defined: Displacement/Rotation – the column base was considered to be stiff, and Temperature – the impact of temperature on the beam was specified, temperature variation in time was given (Amplitude) in accordance with curve (1). In considerations, it was assumed that the beam heating starts from the beam lower flange where the temperature grows with the fire development. Next, while the boundary conditions were specified, two variants of fire location were taken into consideration. Figure 4 shows the situation covered by variant 1. In the fire scenarios proposed, only beams are exposed to fire. Consequently, for the remaining adjacent elements, the heat transfer coefficient was assumed to be -10 [W/m^{2}K] (Fig. 5) (Wiśniewski S. and Wiśniewski T.S., 2000), where a minus sign denotes heat being carried away.



Name: BC-6
Type: Temperature
Step: pozar (Coupled temp-displacement)
Region: Set-11
Method: Specify magnitude
Distribution: Uniform
Magnitude: 945.34
Amplitude: krzywa
Degrees of freedom: 11

Figure 4. Boundary conditions for variant 1.



Edit Load
Name: Load-8
Type: Surface heat flux
Step: pozar (Coupled temp-displacement)
Region: Surf-44
Distribution: Uniform
Magnitude: -10
Amplitude: (Ramp)

Edit Load
Name: Load-8
Type: Surface heat flux
Step: pozar (Coupled temp-displacement)
Region: Surf-41
Distribution: Uniform
Magnitude: -10
Amplitude: (Ramp)

Figure 5. Heat outflow to the surface: variant 1 (a), variant 2 (b).

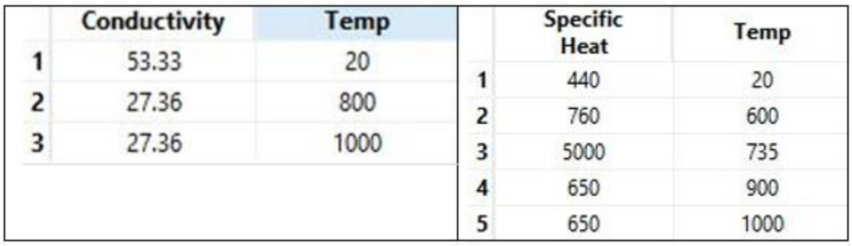
As regards materials data, steel initial elastic properties at room temperature were as follows: density 7850 [kg/m^3], Young’s modulus: 210 [GPa], Poisson’s ratio 0.3, expansion coefficient $1.2 \cdot 10^{-5}$, yield stress 235 [MPa]. Additionally, for Young’s modulus and yield strength, their reduction was taken into account due to temperature change over time. Change in time -dependent values of those two quantities was given in accordance with Standard 1993-1-2 (Table 1) (Standard PN-EN 1993-1-2)).

Table 1. Reduction of Young's modulus and yield strength.

Young's modulus [Pa]	Poisson's Ratio	Temp [^{o}C]	Yield strength [Pa]	Plastic strain	Temp [^{o}C]
210000000000	0.3	20	235000000	0	20
210000000000	0.3	100	235000000	0	100
189000000000	0.3	200	235000000	0	200
168000000000	0.3	300	235000000	0	300
147000000000	0.3	400	235000000	0	400
126000000000	0.3	500	183300000	0	500
65100000000	0.3	600	110450000	0	600
27300000000	0.3	700	54050000	0	700
18900000000	0.3	800	25850000	0	800
14175000000	0.3	900	14100000	0	900
9450000000	0.3	1000	9400000	0	1000

Additionally, thermal properties of the material were defined in accordance with Standard PN-EN 1993-1-2. For computations, thermal conductivity [W/mK] (Table 2a) and specific heat [J/kgK] (Table 2b) were assumed depending on temperature. The stress-strain relation, which is shown in Fig. 6, was also established. Compared with the curve available from Standard PN-EN 1993-1-2, the curve implemented in the software was simplified with the resulting three-segment curve (Fig. 6a).

Table 2. Thermal properties of steel: conductivity (a), specific heat (b).



a)

	Conductivity	Temp
1	53.33	20
2	27.36	800
3	27.36	1000

b)

	Specific Heat	Temp
1	440	20
2	760	600
3	5000	735
4	650	900
5	650	1000

The last stage of modelling involved defining the computational steps. The analysis covered the basic step Static/general, and also Coupled temp-displacement. In the step that accounted for temperature impact, the geometric non-linearity is marked and the time period - 3600 [s], in which the maximum temperature is reached is given T_{max}=945.34 [^{0}C] (1).

Name: stal

Description:

Material Behaviors

Conductivity
Density
Elastic
Expansion
Plastic

Plastic

Hardening: Isotropic

☐ Use strain-rate-dependent data

☑ Use temperature-dependent data

Number of field variables: 0

Data

	Yield Stress	Plastic Strain	Temp
1	235000000	0	20
2	235000000	0.0189	20
3	235000000	0.1489	20
4	0.1	0.1989	20
5	235000000	0	100
6	235000000	0.0189	100
7	235000000	0.1489	100
8	0.1	0.1989	100
9	235000000	0	200
10	235000000	0.0188	200

Figure 6. Stress-strain relationship depending on temperature: according to Standard PN-EN 1993-1-2 (a), data in Abaqus (b).

4 Numerical simulation results and conclusions

The paper discussed the way of modelling steel frame under fire conditions with the use of SIMULIA Abaqus software. Numerical simulation results made it possible to check the structure performance under elevated temperature. The analysis is comparative in character as different fire location variants were considered. The frame numerical model was not validated against experimental data.

The frame numerical computations were stopped before the fire maximum temperature was reached (1) because the heated element showed excessive strain increments. For variant 1, the analysis stopped when the temperature reached 625 [^{0}C], whereas for variant 2 – 629 [^{0}C] (Fig.7). At this stage of the fire, plasticization initiated in the heated elements (Fig. 8) and significant structure deformation were observed.

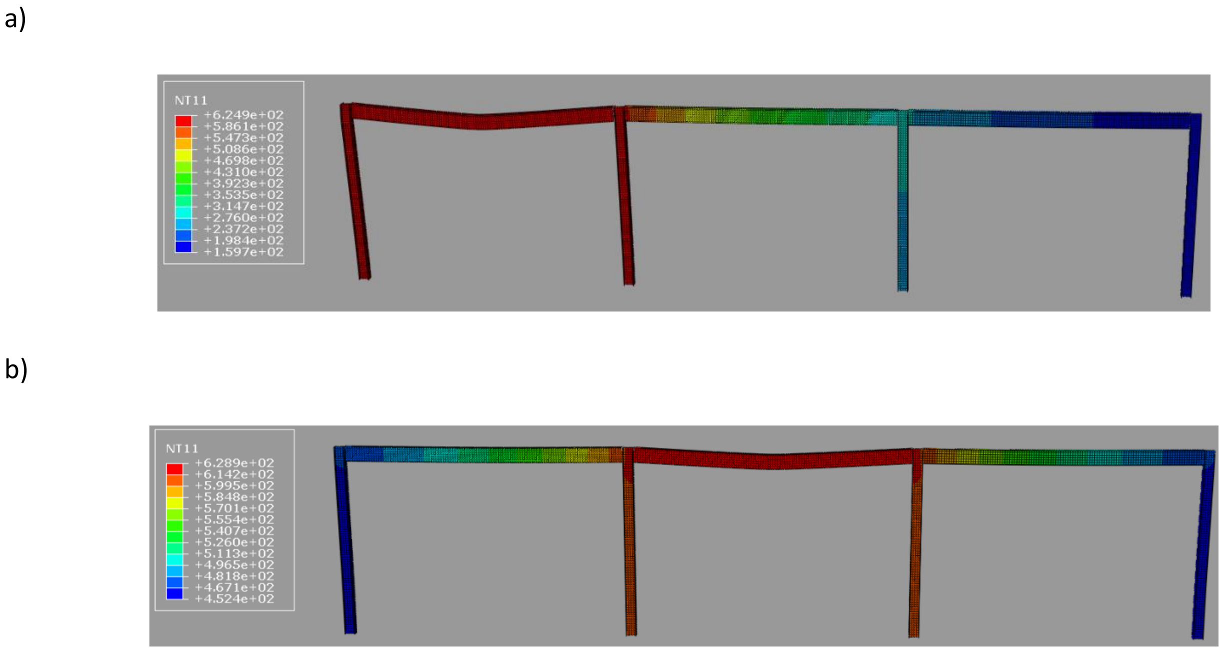


Figure 7. Temperature [0C] distribution in the steel frame: variant 1 (a), variant 2 (b).

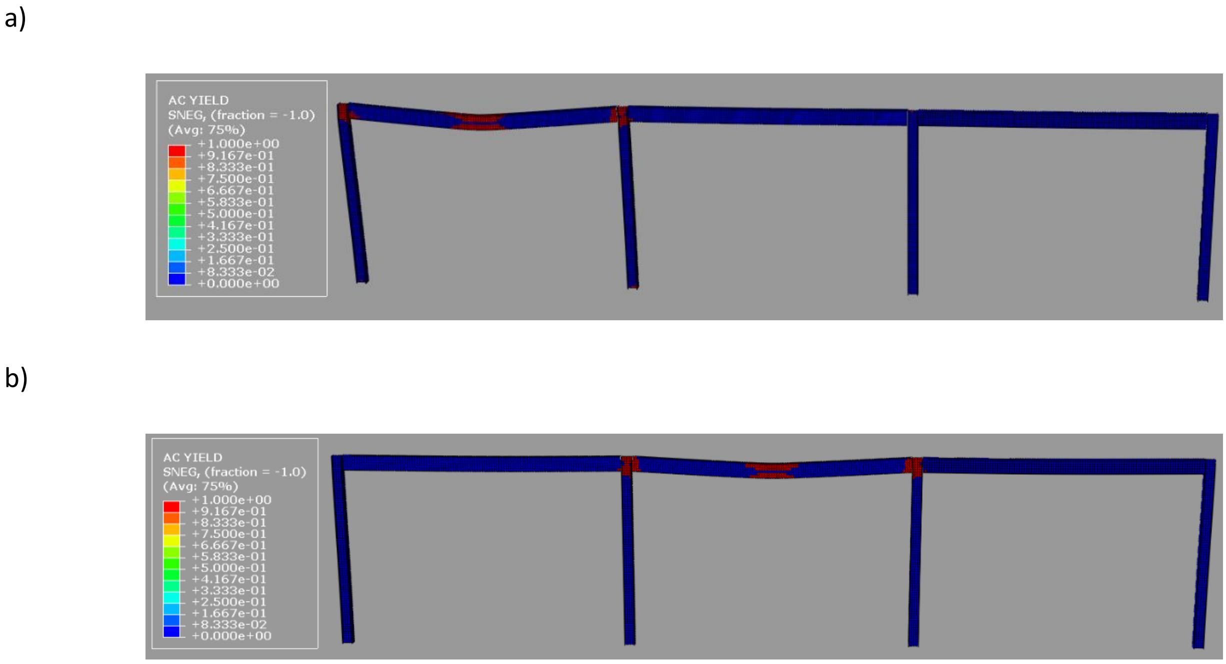


Figure 8. Yield flag: variant 1 (a), variant 2 (b).

Figure 9 shows vertical displacements obtained in the frame under fire conditions when the effect of high temperatures is disregarded. Based on the results received, it can be seen that the values change under the temperature effect. For the frame subjected only to external load, uniformly distributed over the beams and columns (Fig. 3), a vertical displacement was found not to exceed 0.98 [cm] (Fig. 9a). In the thermal frame analysis, however, a significant difference in displacement was observed depending on the fire scenario considered. For fire location variant 1, that is, when beam is heated in the left bay, the maximum vertical displacement amounts to 20.1 [cm]. However, in variant 2, in which the central beam was exposed to fire, the displacements obtained do not exceed 11.2 [cm]. Difference in displacements between two scenarios may result from the fact that in variant 2 end frame sections resist deflection build-up in the central beam. Consequently, displacements generated in the frame under fire conditions, when the fire is located in the central bay, are lower.

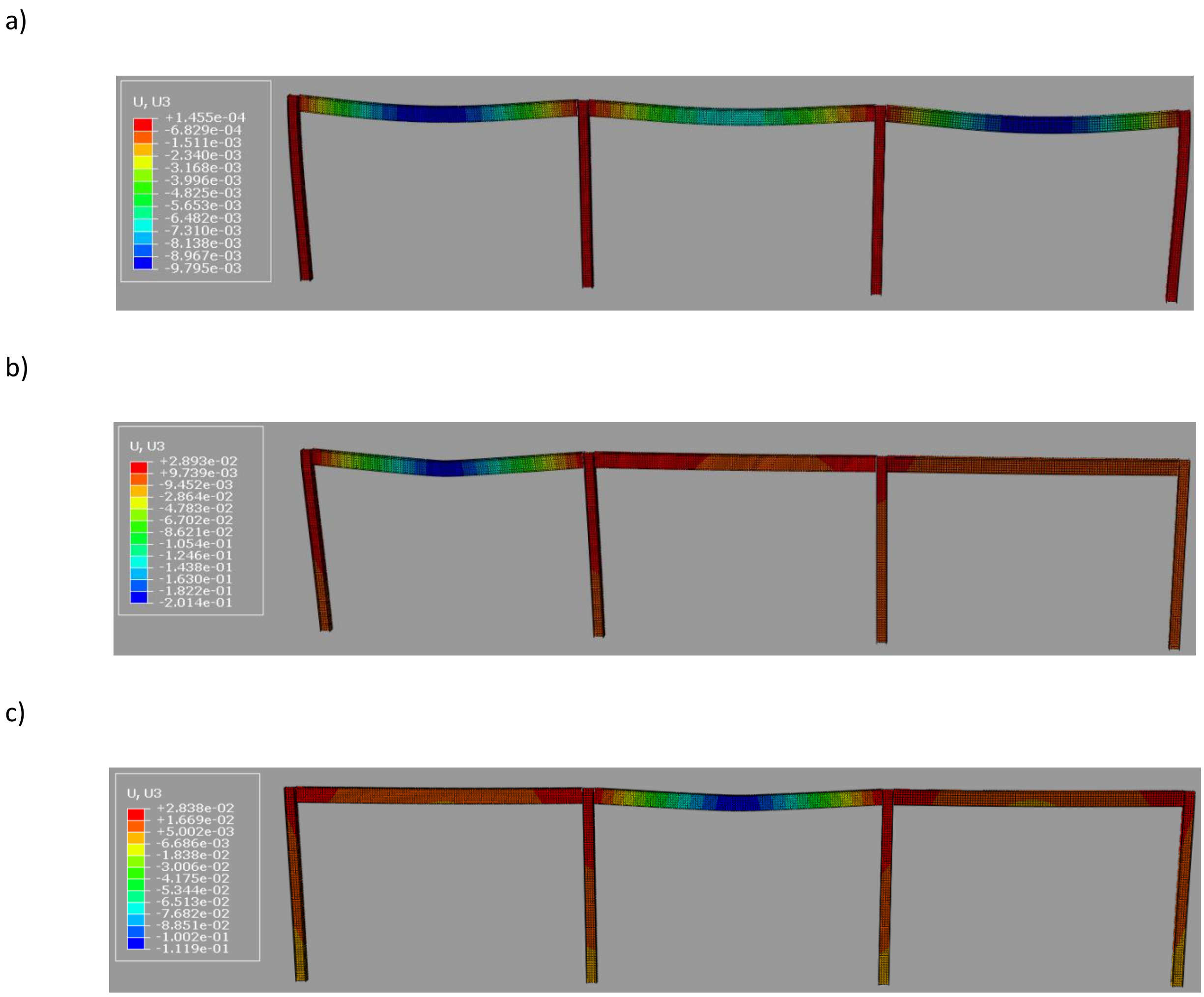


Figure 9. Deformed shape [m] of the steel frame: before fire situation (a), fire situation – variant 1 (b), fire situation – variant 2 (c).

Numerical investigations reported in this paper are exploratory. They are intended as introduction to a broader understanding of the impact fire has on steel structures. To investigate the issue thoroughly, it is necessary to employ an advanced numerical model. The paper demonstrates that fire simulation involves not only the determination of the gas temperature at combustion (1). It is also necessary to account for of a range of data relevant to the problem, namely materials parameters and their dependence on temperature, boundary conditions and loads. The results obtained from the numerical analysis confirm the relevance of investigations into the issues related to structures under fire conditions.

Advanced computational models can become an alternative, or be complementary to simplified computational models used in structural design. Undoubtedly, further experimental and numerical investigations are needed to better understand the performance of steel structures under fire conditions, so that good and safe operating practices could be ensured. It is also important to define guidelines with respect to structure reuse after a fire. The authors of the paper intend to extend the numerical model, test different numerical approaches, and validate the solutions against experimental data. A number of numerical procedures developed in this way could provide guidance for engineers dealing with the fire resistance rating of structures.

References

1. Abaqus CAE 2017, Dassault Systems Simulia.
2. Biegus A. (2013). *Fundamentals of design and actions on civil engineering structures* (in Polish), a collection of lectures, Politechnika Wrocławska, Wydział Budownictwa Lądowego i Wodnego.
3. Chudyba K. (2019). *Analysis of a concrete structure under fire conditions* (in Polish), Wydawnictwo Politechniki Krakowskiej.
4. Gewain R. G., Iwankiw N. R., Farid A. (2003). *Facts for Steel Building – Fire*, United States: American Institute of Steel Construction. https://www.cce.ok.edu.pl/images/skrypty/ANSYS_przeplyw_ciepla.
5. ISO 834-11:2014, Fire resistance tests — Elements of building construction — Part 11: Specific requirements for the assessment of fire protection to structural steel elements.
6. Kodur, V., Dwaikat, M., Fike, R. (2010). *High temperature properties of steel for fire resistance modeling of structures*, Journal of Materials in Civil Engineering 22(5) pp. 423–434. https://doi.org/10.1061/(ASCE)MT.1943-5533.0000041.
7. Lausova, L., Skotnicova, I., Michalcova, V. (2015). *Thermal transient analysis of steel hollow sections exposed to fire*, In Proceedings of the 1st Czech-China Scientific Conference, pp. 247–252. Elsevier GmbH, https://doi.org/10.1016/j.pisc.2015.11.040.
8. Maślak M., Pazdanowski M., Suchodoła M., Woźniczka P. (2021). *Fire resistance evaluation of simple steel hall carried out by using various computational models,* Proceedings of the XIV International Conference on Metal Structures (ICMS2021), Poznań, Poland, 16-18 June. Doi:10.1201/9781003132134-66.
9. Mika P, Słoński M. (2006), Analysis of steady-state heat transfer in the disc with Ansys software (in Polish), Reports, Politechnika Krakowska.
10. Pantousa D., Karavasilis T., Maraveas Ch. (2022). *Numerical Investigation of the Post-Fire Performance of Steel Columns*, Buildings 12(3), 288.
11. PN-EN 1991-1-2:2006, Eurocode 1: Actions on structures. Part 1-2: General actions – Actions on structure under fire conditions.
12. PN-EN 1993-1-2:2007, Eurocode 3: Design of steel structures. Part 1-2: General rules– Structural Fire Design.
13. Snela M. (2017). *Estimation of the critical temperature of steel tilt frames in a developed fire, taking into account the changing structural properties of the nodes* (in Polish), Politechnika Lubelska, Lublin.
14. Tkaczyk A. (2017). *Strength limit of steel frame structures in developed fire - estimation with a kinematic method* (in Polish), Doctoral thesis, Politechnika Krakowska, Wydział Inżynierii Lądowej, Kraków.
15. Wiśniewski S., Wiśniewski T.S. (2000): Heat Transfer (in Polish), Wyd. 5, WNT, Warszawa.