Civil Engineering
Engenharia Civil

Finite element computational development for thermo-mechanical analysis of plane steel structures exposed to fire

http://dx.doi.org/10.1590/0370-44672021750006

Natan Sian das Neves[1,3]
https://orcid.org/0000-0002-4679-0461
Rodrigo Silveira Camargo[2,4]
https://orcid.org/0000-0001-9850-7018
Macksuel Soares de Azevedo[2,5]
https://orcid.org/0000-0003-3011-3420

[1]Universidade Federal do Rio de Janeiro - UFRJ, Instituto Alberto Luiz Coimbra de Pós-Graduação e Pesquisa em Engenharia - COPPE, Programa de Engenharia Civil - PEC,
Ilha do Fundão - Rio de Janeiro - Brasil.

[2]Universidade Federal do Espírito Santo - UFES, Programa de Pós-Graduação em Engenharia Civil - PPGEC, Vitória - Espírito Santo – Brasil.

E-mails : [3]natan.sian@gmail.com,
[4]rodrigo_camargo2000@yahoo.com,
[5]macksuel.azevedo@ufes.br

Abstract

This article presents a numerical formulation based on finite element procedures for application in nonlinear thermo-mechanical analyses of steel planar structures under fire condition. The mechanical properties of structural elements degrade when subjected to high temperatures, resulting in significant reductions in strength and stiffness. Under these conditions, the structures present complex behaviors associated with nonlinear models, requiring an advanced mathematical analysis. As such, a computer program called NASEN has been developed to investigate the behavior of steel structures subjected to fire, considering the effects of geometric and material nonlinearity, as well as the thermal gradients acting on the cross-section. The solution strategy is based on sequential coupling of numerical processes. Initially, the two-dimensional thermal field is determined, followed by an assessment of structural behavior. In each solution step, corrective processes are implemented to ensure convergence of the temperature and displacement nodal vectors. Numerical experiments are performed in order to evaluate the accuracy and capacity of the computer program. Results are compared with experimental tests and computer simulations found in pertinent literature. The program shows good agreement with reference solutions, indicating its accuracy and applicability for the cases studied.

Keywords: finite elements, fire analysis, thermo-structural, steel, advanced computational method.

1. Introduction

The various occurrences of fire throughout history, associated with technological advancement, increased the demand for scientific research concerning the analysis of structures subjected to high temperatures. As such, standardized design guidelines were developed to improve structural safety and minimize risks associated with the eventual incidence of a fire. The Brazilian Association of Technical Standards (ABNT) published a number of guidelines on the subject. These Brazilian standards are a result of extensive research in the field of fire safety engineering. In addition, well-known international standards are cited, such as the EN 1991:2002, EN 1992:2004 and EN 1993:2005.

Generally, the standards not only present simplified design methods, but also prescribe the application of advanced methods for the analysis of structural elements exposed to fire, as well as the use of experimental tests. Certain scenarios are beyond the scope of the simplified methods, requiring the use of advanced analysis procedures. As such, the ever-increasing development of computational resources makes this approach an excellent alternative for verifying the behavior of structures exposed to high temperatures, allowing improvements in cost-benefit, overall design, simulation of load types, resistance and safety criteria (Purkiss and Li, 2013). The analysis of structures under fire aims to verify the behavior of structural elements subjected to external effects of thermal origin. The response of the structural system is obtained by determining stresses, strains and displacements resulting from thermal effects, in addition to quantifying the degradation of thermo-mechanical properties of materials as a consequence of temperature increase within the structural elements (Caldas, 2008).

In this context, numerous researches were carried out in the area of computational development for fire safety engineering, providing important recommendations for analyzing the performance of structures subjected to fire. Initially, thermal analyses of structural elements exposed to fire consider a transient regime and nonlinear characteristics. These analyzes are performed with advanced computational models, mostly based on the finite element method (FEM). Amid numerical codes developed to predict thermal fields, Wickström (1979) stands out among pioneering researches for the development of the computer program TASEF-2. The program uses a rectangular finite element mesh, accounting for material and boundary condition nonlinearity.

Additional algorithms for numerical treatment of heat transfer by convection and radiation in cavities are also presented. Ribeiro (2004) developed THERSYS, a computer program based on FEM for transient thermal evaluation of two-dimensional and three-dimensional steel, concrete and steel-concrete composite structures. The author validated the program with numerical data from programs that are well-established in the scientific community, comparing results from numerous cases with procedures prescribed in the Brazilian standards. Pierin *et al.* (2015) presented a numerical program called ATERM, which performs two-dimensional thermal analysis using linear triangular and rectangular finite elements, including specific treatments for air enclosed in cavities. The program features a didactic interface, allowing the definition of various geometries, boundary conditions, materials and other parameters. Following the same line, Pires *et al.* (2018) presented an application of the computational module CS-ASA/FA, capable of performing analyzes with different solution strategies, such as simple incremental processes, Picard and Newton-Raphson iteration methods. Regarding researches on computational development aimed at thermo-structural analysis, Franssen (1987) presented a computational model based on the finite element method for analysis of steel-concrete composite plane frames subjected to high temperatures. Said program, called CEFICOOS, is the first computer program for analysis of structures under fire developed at the University of Liège, in Belgium. Subsequently, the calculation procedures were improved and expanded, resulting in the SAFIR program, which included different element types, three-dimensional analyses and constitutive models (Franssen, 2005). This program is widely known and used in the scientific community to assess and analyze the behavior of structures subject to fire.

Wang and Moore (1995) developed a computer program based on the finite element method that studies the behavior of steel and reinforced concrete structures under fire considering uniform and non-uniform temperature distribution. The program features semi-rigid connections, second-order effects, effects of residual stresses and the consideration of initial imperfections. Iu (2004) developed a formulation for the analysis of steel structures under fire using the finite element method, based on principles of the plastic hinge method. The nonlinear equations were solved with the Newton-Raphson method. The program considers effects of geometric and material nonlinearity, as well as the strain hardening of steel. Reduction in material strength as a result of increased temperature is approximated by a set of temperature-stress-strain curves. Additionally, the program also allows analyses during the cooling phase of the structure.

In Brazil, Landesmann (2003) developed the PNL-F computer program for non-linear elastoplastic analysis of plane steel frames under fire conditions. The thermal analysis is conducted by a one-dimensional transient thermal model based on the finite element method. The structural analysis was based on concepts of the concentrated plasticity method, obtained from the improvement of plastic hinge models, tangent modules, stability functions and inelastic strength reduction surfaces, allowing estimation of the critical time of fire resistance associated with structural collapse mechanisms. The author later revised the developed program, calling it SAAFE. Several additional studies in this area were conducted to investigate the behavior of structures, such as Mouço (2008), Rigobello (2011), Ribeiro (2009) and Caldas (2008). Barros (2016) recently implemented two new modules to the CS-ASA program, aimed at obtaining the temperature distribution at the cross section and performing inelastic second-order numerical analyses of steel structures exposed to fire. The principles adopted by the program, to model the inelastic structural behavior, are based on the refined plastic hinge method coupled with the strain compatibility method (Barros *et al.*, 2018). Later, Maximiano (2018) expanded the program's functionalities to study the behavior of steel, reinforced concrete beams, columns and frames subjected to fire. Prakash and Srivastava (2019) developed a fully coupled hydro-thermo-mechanical formulation, based on the direct stiffness method for analysis of reinforced concrete and steel spatial structures. The stiffness matrix is constructed by directly integrating stability and curvature functions with the effects of material properties as a function of temperature, fire damage, pore pressure, nonlinear thermal gradients and large deformations of structural elements.

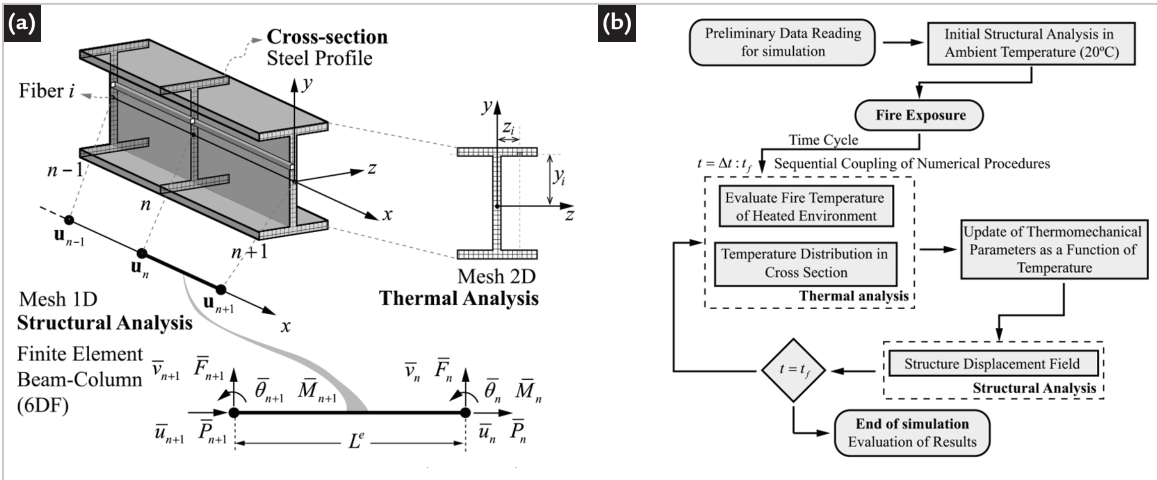

Figure 1 - (a) Levels of discretization of the structural element and (b) general solution procedure.

The investigations and numerical analysis carried out in this study are performed with the computer program NASEN (*Numerical Analysis System for Engineering*). The computational implementations in the program are developed in a MATLAB R2015a environment, based on structured programming. The program features modules capable of performing advanced numerical analyses of numerous classic engineering problems. So far, the program is applicable to the initial solution of structural, thermal, thermo-mechanical and acoustic problems. Specific sub-routines for treatment of structures subjected to fire are also included (Neves, 2019; Neves *et al.*, 2020). In this study, the performance of the computational modules developed for the analysis of planar steel structures under high temperatures is evaluated. Two computational modules were used from the NASEN program. The first code, NASEN/TA-FIRE (*Thermal Analysis - Fire*), determines the temperature distribution throughout the cross-section of structural elements with FEA. The second code performs thermo-mechanical analyses of structures under fire. This module, designated NASEN/TSA-FIRE (*Thermal-Structural Analysis-Fire*), considers the degradation of physical and mechanical properties as a result of increased temperatures, as well as material and geometric nonlinearity. The interface between thermal and structural analyses is achieved by defining equivalent properties and the thermal fixed-end forces present in the elements exposed to fire.

2. Basic formulation of fire analysis

The computational model implemented to solve the thermomechanical problem of plane steel structures exposed to high temperatures focuses on three essential aspects of numerical simulations of this nature. Firstly, a heat source equivalent to fire is modelled based on normative curves prescribed in EN 1991:2002. Subsequently, thermal analysis of the cross-section is performed, followed by the definition of structural displacements. Moreover, the following basic assumptions of the physical-mathematical formulation adopted in the fire analysis:

(i) The cross-section area of the member is assumed undeformed due to applied load and high temperature;

(ii) The temperature distribution in the steel cross-section is not considered uniform, as is the case with simplified calculation methods. The thermal strain which consists of both thermal axial strain and curvature;

(iii) The plane sections remain plane. It assumes that any section of a beam that was a flat plane before the beam deforms will remain a flat plane after the beam deforms (Bernoulli–Euler hypothesis);

(iv) Shear deformation and warping deformation are neglected;

(v) Local buckling effects that occur on the flanges and web of steel profiles are not considered in the formulation. Thai *et al.* (2017) and Maraveas (2019) show more information on the consideration of local buckling in structures under fire conditions;

(vi) The strain is small, but arbitrarily large deformation and rotation are allowed;

(vii) Effects from higher than second-order terms are neglected in the mathematical formulation.

(viii) The applied loading does not change during the fire and only independent loading is considered.

2.1 Thermal analysis

The prediction of the thermal field at the cross-sectional level of the structures is simulated separately from the one-dimensional beam-column model, as shown in Figure 1a. Thus, based on the principle of energy conservation, the governing equation for nonlinear transient heat conduction (Purkiss and Li, 2013) is given by Equation (1).

$$\nabla^T \mathbf{D}(T)\nabla T + Q = \rho(T)c(T)\frac{\partial T}{\partial t} \qquad (1)$$

where T is the temperature, Q is the heat source, ρ is the specific mass, c is the specific heat and $\mathbf{D} = \mathbf{I}k$ is the thermal conductivity matrix for an isotropic material. The behavior of physical properties for steel and concrete as a function of temperature increase, follows the mathematical formulas described in EN 1992:2004 and EN 1993:2005. In a fire situation, the structural elements are subjected to a combination of convection and radiation heat transfer. These effects are considered in the boundary conditions of the problem in linearized form according to Equation 2.

$$-\left(\nabla^T T\right)\mathbf{D}\mathbf{n} = q_n = \alpha(T - T_g) \tag{2}$$

where α is the equivalent convection-radiation heat transfer coefficient, **n** is the normal unit vector at the boundary and T_g is the temperature of the gases around the structure under fire, usually modeled by standardized temperature-time curves, as can be seen in EN 1991:2002. For thermally insulated surfaces, the heat flow is zero, that is, $q_n = 0$. Applying differential-integral calculus and using finite element approximations (Reddy, 2015), the discrete variational formulation for heat diffusion problems in transient regime is given by Equation (3).

$$\int_\Omega \mathbf{N}^T \rho c \mathbf{N}\left\{\frac{\partial \mathbf{T}}{\partial t}\right\} d\Omega + \int_\Omega \mathbf{B}^T \mathbf{D}\mathbf{B}\left\{\mathbf{T}\right\} d\Omega = \int_\Omega \mathbf{N}^T Q d\Omega - \int_\Gamma \mathbf{N}^T q_n d\Gamma \tag{3}$$

where $\mathbf{N} = [N_1\ N_2\ \dots\ N_n]$ is the vector that contains the shape functions and $\mathbf{T} = [T_1\ T_2\ \dots\ T_n]$ is the approximated temperature nodal vector. The gradient vector can be written in conjunction with the finite element interpolation function, $\nabla\mathbf{T} = \nabla\mathbf{N}\mathbf{T} = \mathbf{B}\mathbf{T}$. Thus, after spatial discretization, it is possible to rewrite Equation (3) in the compact matrix form given by Equation (4).

$$\mathbf{C}_\theta\left\{\frac{\partial \mathbf{T}}{\partial t}\right\} + \mathbf{K}_\theta\left\{\mathbf{T}\right\} = \mathbf{F}_\theta \tag{4}$$

where $\mathbf{C}_\theta$ is called the thermal capacity matrix, $\mathbf{K}_\theta$ is the total capacitance matrix and $\mathbf{F}_\theta$ is the vector of thermal loads. Temporal discretization is based on finite difference approximations applied to the time-dependent ordinary differential equation (Neves, 2019; Reddy, 2015). Finally, these procedures result in an effective algebraic system that provides the temperature at each node of the two-dimensional mesh at instant $n + 1$, given by Equation (5).

$$\left[\mathbf{C}_\theta + \beta\Delta t\mathbf{K}_\theta\right]\left\{\mathbf{T}_{n+1}\right\} = \left[\mathbf{C}_\theta - (1-\beta)\Delta t\mathbf{K}_\theta\right]\left\{\mathbf{T}_n\right\} + \Delta t\left[(1-\beta)\mathbf{F}_{\theta,n} + \beta\mathbf{F}_{\theta,n+1}\right] \tag{5}$$

where Δt is the time step and β is the parameter that controls the time integration scheme. The present work adopts the Galerkin method ($\beta = 2/3$), characterized as an unconditionally stable method. In addition, Equation (5) is of nonlinear nature due to the temperature-dependent properties and the presence of convection-radiation boundary conditions. Thus, an incremental-iterative strategy called the Newton-Raphson method is implemented in the code, in which the convergence tolerance is predefined by the user.

2.2 Thermal-structural analysis

The structural analysis considers a one-dimensional plane beam-column model with three degrees of freedom (DOF) in each node. More specifically, two translations and one rotation, as shown in Figure 1a. Based on the principle of virtual displacements, using the updated Lagrangian method and the hypotheses of the Euler-Bernoulli theory (Yang and Kuo, 1994), the incremental equation is given by Equation (6).

$$\left\{\Delta\mathbf{f}\right\} = \left[\mathbf{K}_e\right]\left\{\Delta\mathbf{u}\right\} \tag{6}$$

where $\Delta\mathbf{u} = \{u_{n+1}\ v_{n+1}\ \theta_{n+1}\ u_n\ v_n\ \theta_n\}$ is the displacement incremental vectors and the force incremental vectors is $\Delta\mathbf{f} = \{P_{n+1}\ F_{n+1}\ M_{n+1}\ P_n\ F_n\ M_n\}$. The elementary stiffness matrix ($\mathbf{K}_e$), obtained by summation of matrices associated with the linear and nonlinear behaviors, according to Equation (7).

$$\left[\mathbf{K}_e\right] = \left[R\right]^T\left[K_l\right]\left[R\right] + \left[K_g\right] \tag{7}$$

The geometric matrix (K_g) accounts for the nonlinear portion of the structural problem, and is obtained with a simplified theory, disregarding the combined incidence of axial and flexural behavior. In addition, R is a transformation matrix, of order 3x6. The linear elastic stiffness matrix (K_l) is represented by:

$$k_l = \begin{bmatrix} 4\overline{EI}/L & 2\overline{EI}/L & \\ 2\overline{EI}/L & 4\overline{EI}/L & \\ & & \overline{EA}/L \end{bmatrix} \tag{8}$$

The matrix of elastic properties is written as a function of equivalent axial ($\overline{EA}$) and flexural ($\overline{EI}$) stiffnesses, both calculated based on the modulus of elasticity, area and distance between the center of gravity and each fiber of the structural cross section. Reductions in the value of the longitudinal elastic modulus as a result of high temperature exposure may be represented by different mathematical models.

For elements exposed to fire, specific treatments related to thermal effects on the structure must be carried out. The equivalent nodal forces are obtained by assuming that both ends of the beam-column element are totally restricted (Landesmann, 2003; Mouço, 2008). The perfect fixed-end vector $\mathbf{f}_{th} = \{P_\theta \; 0 \; M_\theta\}^T$ is composed of the contributions resulting from the effects of thermal expansion and curvature due to the temperature gradient of the cross section. Vector components are detailed in Equation (9).

$$P_\theta = \int_A \varepsilon_{th} E_\theta dA \approx \sum_{k=1}^{n} \varepsilon_{th,k} E_{\theta,k} A_k \qquad M_\theta = \int_A \varepsilon_{th} E_\theta y dA \approx \sum_{k=1}^{n} \varepsilon_{th,k} E_{\theta,k} y_k A_k \quad (9)$$

The variable ε_{th} represents the thermal elongation of the material as a function of temperature, determined in accordance with recommendations from the EN 1991: 2002. The equivalence is completed by summation of the thermal fixed-end force vector with the vector of forces of the structural system.

Considering a non-uniform temperature distribution across the cross-section, the effective plastic strength limits, respectively associated with the axial and flexural stiffnesses, are determined, as suggested in EN 1993:2005, by the following expressions:

$$P_{y\theta} = \int_A f_{y,\theta} dA \approx \sum_{k=1}^{n} f_{y,\theta,k} A_k \qquad M_{p\theta} = \int_A f_{y,\theta} |y| dA \approx \sum_{k=1}^{n} f_{y,\theta,k} |y_k| A_k \quad (10)$$

where $f_{y,\theta}$ represents the steel strength limit as a function of temperature, obtained based on the EN 1993:2005 recommendations. The plastic strength limits of the cross-section, under ambient temperature conditions (20°C), do not change at all, and may be obtained directly by $f_{y,\theta} A$ and $f_{y,\theta} Z$, where Z is the plastic module of the steel profile cross-section.

2.3 Computer program information

To perform the simulation and modeling of the behavior of structural elements under fire situation, the computer program developed is based on an uncoupled mathematical strategy. The temperature distribution is previously calculated, disregarding the mathematical coupling between thermal and mechanical effects. Subsequently, a mechanical analysis is performed in which the incidence of thermal effects is included in the structural model. This methodology was applied by several researchers, as seen in the Caldas (2008), Landesmann (2003), Ribeiro (2009), Neves *et al.* (2021) and Maximiano (2018). To illustrate this idea, Figure 1b shows the global scheme of the solution process.

Regarding the general solution strategy, the program starts with the preliminary nonlinear analysis of the structure in ambient temperature subjected exclusively to external loads. During the fire exposition phase, for each time interval, the temperature of the gases in the environment is updated and, based on advanced calculation methods, the temperature field in the cross-section, equivalent properties associated with stiffness, as well as strength and thermal capacities are determined. Finally, the mechanical analysis is performed. Iterative processes are implemented in each step to ensure that results converge within a predefined tolerance.

To perform the numerical simulations, the computational code for thermomechanical analysis of steel structures under fire condition requires reading from three input files: (i) two-dimensional finite element mesh (**.msh*) – file containing node coordinates and element connectivity at the cross-section level and it is obtained with the Gmsh program (Geuzaine; Remacle, 2009); (ii) file containing structural characteristics of the model (**.pos*) – structural coordinates, materials, loads and support conditions are entered into this file; (iii) thermo-structural model file (**.txt*) – contains information about thermal conditions (faces exposed to fire, emissivity, convection coefficient), tolerance values, load factors and other parameters.

In the process of discretization of the structural model, one-dimensional finite elements of 6DOF are used (see Figura 1a). In addition, linear and cubic polynomials are used to describe the axial and bending components of the displacement vector. In the two-dimensional finite element library available in the NASEN program, there are different types of plane finite elements: T3 and T6 – Triangular element with 3 and 6 nodes, respectively, and Q4, Q8 and Q9 – Quadrilateral element with 4, 8 and 9 nodes. These elements are used in discretization across the cross-section of the members. In the present study, only the T3 element is applied, it does not require the use of numerical integration.

3. Numerical experimentation

The examples used for the NASEN program tests and validations consist of isolated columns and plane steel frames in fire situations. In all cases, to evaluate the performance of the program, the results are compared with data obtained in computer simulations and experimental tests found in literature.

3.1 Isolated steel column under fire situation

Consider the column models with a length of 4 m subjected to the thermal and contact loading shown in Figure 2a and Figure 2b. The cross-section is that of an IPE 360 steel profile, shown in Figure 2c. Concentrated moments of magnitude 20%M_{p20} are applied at both ends, where M_{p20} is the plastic moment of the cross-section at 20ºC. Additionally, a compression load of 30%P_{y20} acts on the upper end, where P_{y20} is the resistant plastic axial force at 20ºC. In Figure 2a, the combined convection-radiation heat flux acts only on three sides of the steel profile. This situation simulates a column partially protected from fire. In the second configuration, presented in Figure 2b, all four sides are exposed to high temperatures. This situation is a common representation of an interior column in a building subjected to fire. As such, this example is strategically chosen to carry out the program verification process, since it has been tested in numerous works with different approaches, such as in Landesmann *et al.* (2005), Rigobello (2011), Maximiano (2018) and Caldas (2008).

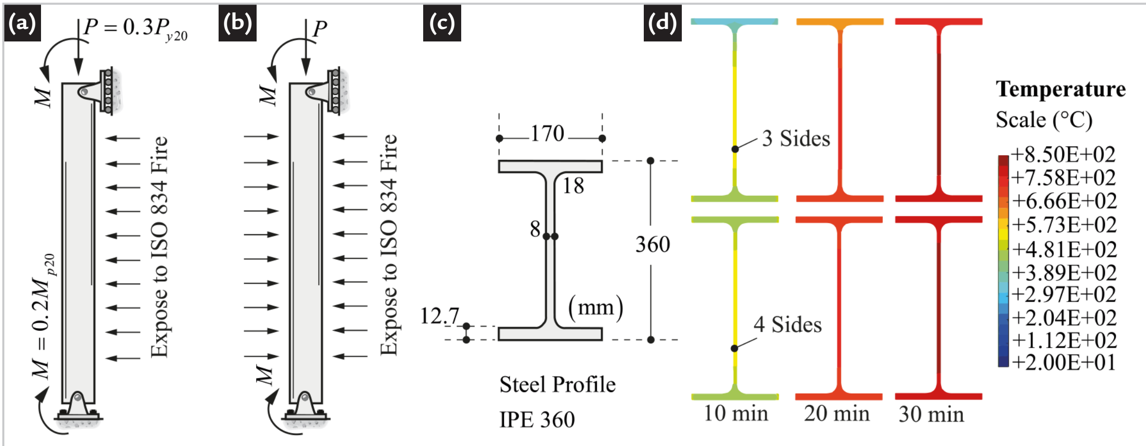

Figure 2 - Structural scheme and loads applied to the steel beam-columns model with (a) 3 and (b) 4 sides exposed to fire, (c) cross-section dimensions and (d) 2D thermal field for 10, 20 and 30 min.

Initially, a thermal analysis is performed considering the two boundary conditions applied to the steel profile, using a linear triangular mesh with 230 finite elements and a time interval Δt of 10 s. The temperature-time curves for 3 and 4 sides exposed to the fire are shown in Figure 3a. The curves are obtained at points located on the flange and web of the IPE 360 profile. The results of the developed program are satisfactory when compared with the SAFIR program (Franssen, 2005), where the numerical data are obtained in Maximiano (2018). As seen, when all sides are exposed to fire, due to the symmetry, the temperatures of both flanges remain practically the same, and with lower temperatures than those of the web. On the other hand, there is a significant difference between the temperature levels when considering only three sides subjected to fire. In this situation, there is a reduction in temperature on the top flange and the highest temperatures occurs on the web as result of the slenderness of this element. In addition, to check the temperature distribution in the cross-section of the structure, Figure 2d shows the thermal field for 10, 20 and 30 min of exposure to fire. Note that the temperatures in each fiber of the section remain close, indicating an almost constant temperature in the section.

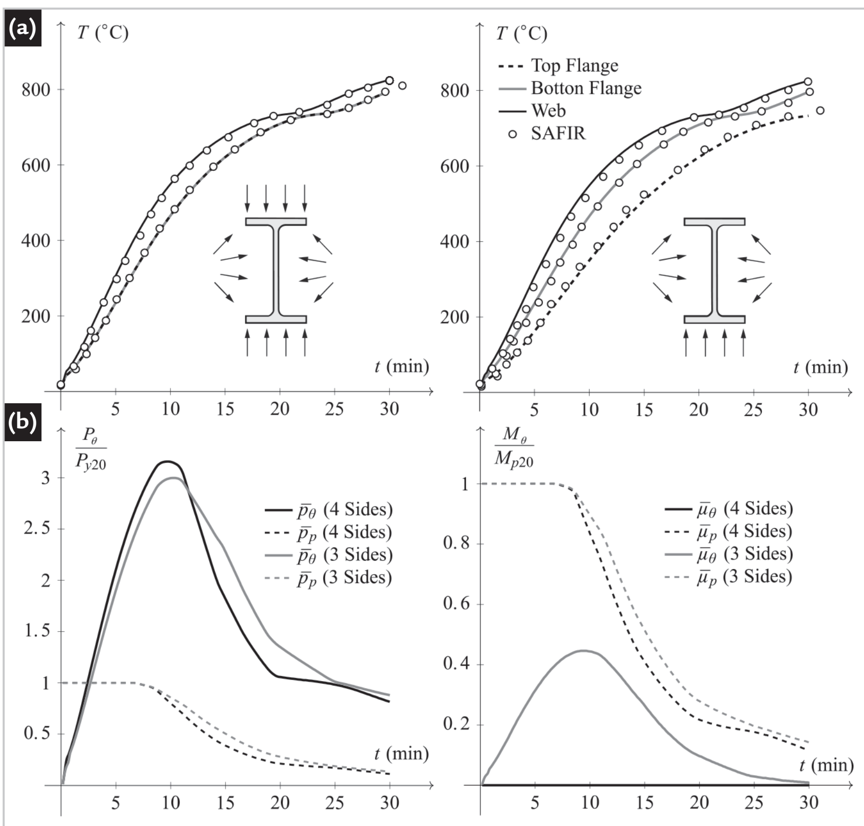

Figure 3 - (a) Temperature-time curves and (b) normalized range of thermal fixed-end forces and plastic strength.

The normalization of the thermal fixed-end forces and the plastic resistance associated with axial stiffness are represented by $\bar{p}_\theta = P_\theta / P_{y20}$ and $\bar{p}_p = P_{y\theta} / P_{y20}$. Alternatively, these variables are represented by $\bar{\mu}_\theta = M_\theta / M_{y20}$ and $\bar{\mu}_p = M_{p\theta} / M_{p20}$ for flexural stiffness, as shown in Figure 3b. These variables are computed through the application of Equation (9) and Equation (10), which depend on the two-dimensional thermal field of the cross-section of the steel element exposed to fire (emphasizing that the properties of steel (thermal and mechanical) undergo severe degradations at high temperatures). During the fire, these values are important to verify the action levels of thermal effects on the steel elements of the computational model.

Figure 3b shows that symmetrical heating of the section results in a greater loss of strength capacity if compared with its asymmetrical counterpart. Moreover, maximum values of axial and bending moment thermal fixed-end forces occur at approximately 10 minutes during asymmetrical heating. Both boundary conditions yield similar values of axial thermal reactions. In contrast, bending moment thermal reactions are not developed during symmetrical heating.

The mechanical analysis performed with NASEN is compared with results obtained by Landesmann *et al.* (2005) and results from SAFIR (Franssen, 2005) and VULCAN (Huang *et al.*, 2003) programs, extracted from the simulations performed by Caldas (2008). The structural mesh is discretized into 8 one-dimensional elements and the maximum horizontal deflections in the beam-column model are measured at midspan. Figure 4a show the horizontal displacement over time for both boundary conditions. Comparison with reference solutions show satisfactory results. Additionally, it is noted that the asymmetric thermal condition indicates a longer critical time, if compared with symmetrical heating.

In the context of structural design, the interaction curves have important characteristics related to the plasticity limits of the section. Therefore, Figure 4b shows the normalized interaction curves of the IPE 360 steel profile. As heat exposure time increases, a reduction in the plastic resistant capacity of the cross section occurs. The symmetrical thermal condition shows slightly higher levels of reduction, if compared to its asymmetric counterpart. This is a result of accelerated degradation of the properties of steel caused by the exposure of all four sides to the heat source, inducing a decrease in resistant capacity.

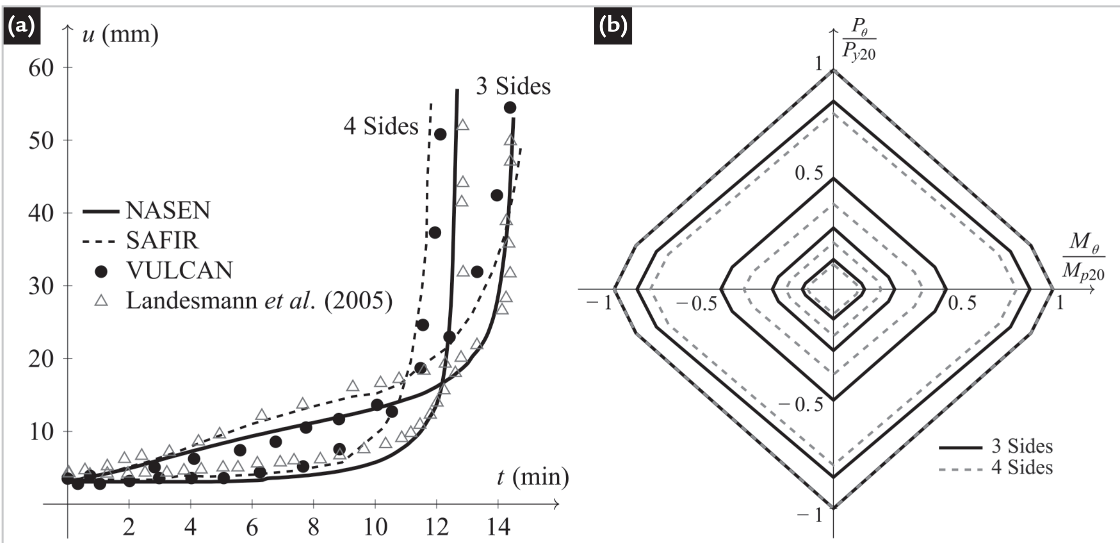

Figure 4 - (a) Deflection of the column simply supported and (b) interaction curves in fire conditions.

3.2 Steel frames in fire

This section numerically analyzes the thermomechanical behavior of a series of experimental tests on steel frames in a fire situation. The first case analyzed in this section is composed of the frames studied by Rubert and Schaumann (1986). In the present study, only the configurations called EGR (simple frame) and ZSR (double frame) are analyzed, as shown in Figure 5a. The frames were previously loaded and then subjected to heating at a constant rate until collapse. All structural elements of the EGR configuration are evenly heated. In contrast, only the left compartment of the ZSR configuration is heated, while the other is kept at ambient temperature. The performance of the numerical analyzes is assessed by comparison with experimental data, as well as with additional numerical results from the computer programs SYSAF (Rigobello, 2011) and CS-ASA/FSA (Maximiano, 2018).

All profiles have an IPE 80 cross-section with a modulus of elasticity of 210 GPa. For the numerical analyses, the cross-sections are discretized in 242 three node linear triangular elements, while the global structural system is subdivided into 5 one-dimensional elements. Figure 6a shows the horizontal displacements u_1, u_2 and u_3 measured on the simple frame and Figure 6b shows the horizontal displacements u_4 and u_5 of the double frame under fire condition.

The numerical values obtained with NASEN program show good agreement when compared with reference experimental results and numerical simulations. It is observed that the critical temperatures of the analyzed frames are close to 500°C. Quantitatively, Table 1 shows the values extracted from the computer program proposed here and the numerical solutions found in literature, which are also compared with the experimental data.

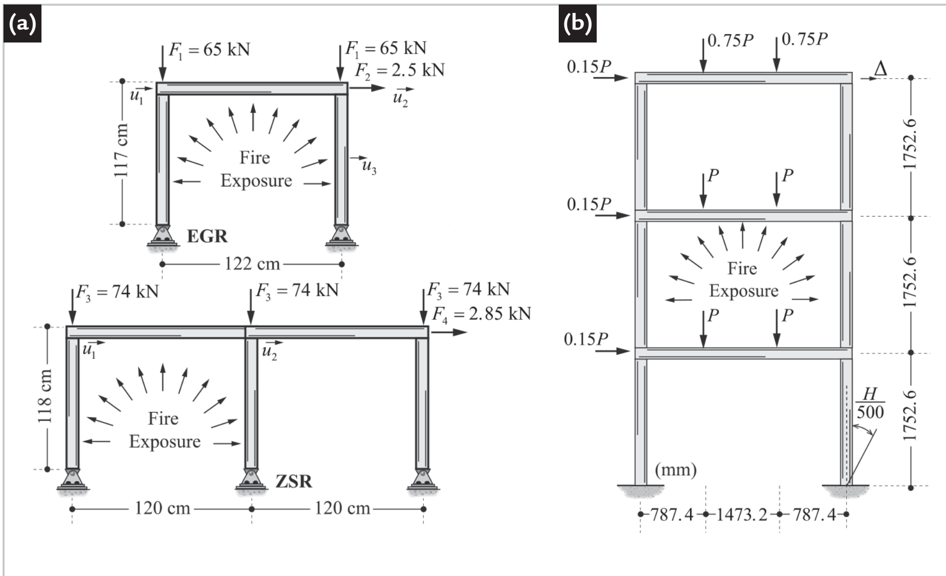

Figure 5 - (a) Simple and double frames, and (b) three-story steel frame exposed to fire.

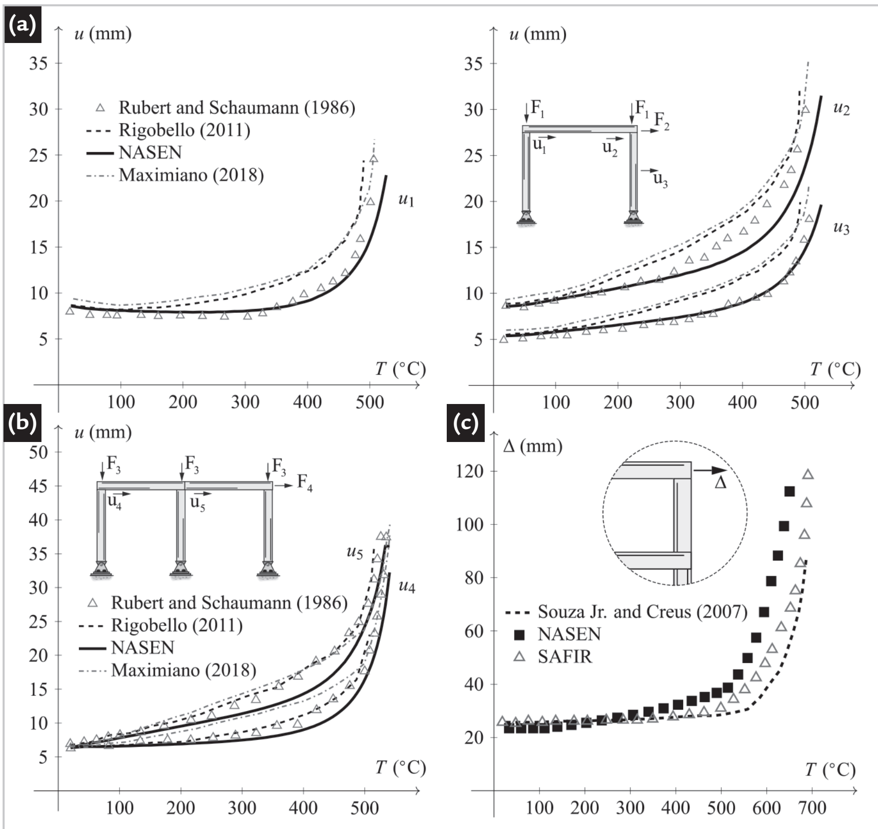

Figure 6 - Displacement-temperature curves: (a) simple, (b) double and (c) three-story steel frame.

Table 1 - Critical temperature of frames.

Model	Frame Configuration Type			
	EGR	Δ (⁰C)	ZSR	Δ (⁰C)
Rubert and Shaumann (1896)	515	--	547	--
Rigobelo (2011)	491	24	515	32
Maximiano (2018)	507	8	549	-2
NASEN	526	-11	540	7

*Differences (Δ) are measured in relation to experimental tests

The second case studied is defined by a three-story steel frame with an initial imperfection subject to horizontal and vertical loads, as shown in Figure 5b. This example was first studied by McNamee and Lu (1972). In the article by Souza Junior and Creus (2007), the second floor of the structure is subjected to the action of fire. The force *P* is equal to 30 kN and the modulus of elasticity of the steel is 200 GPa. The beams consist of a profile W150x100x24 mm, while the columns are formed by the profile W100x100x19,3 mm. For simulation, the structural system is considered discretized into 15 finite elements. The horizontal displacement at the right end of the frame in relation to the temperature increase is shown in Figure 6c. The values obtained show reasonably satisfactory results, showing a more conservative behavior after heating exceeds 300ºC. As can be seen in Figure 6c, the critical temperature measured with the NASEN computer program is lower than the values found in the literature solutions. In general, the developed program was able to adequately simulate the behavior of the studied frame.

4. Conclusions

This article described aspects of interest related to numerical implementation in engineering, focused on the development of a computer program, called NASEN, capable of simulating the behavior of structures under fire situations. Investigations carried out herein are based on the analysis of plane reticulated steel structures exposed to high temperatures. Due to the computational implementation being carried out in a Matlab environment, it is possible to use different graphic resources in the post-processing, making it possible to carry out investigations and evaluations of the results of interest to engineering. Results obtained with the NASEN program showed acceptable agreement when compared to data found in literature. The developed program presents a low computational cost and fast convergence, since the structural mesh is described by one-dimensional elements and the steel cross-section is slender and requires few plane finite elements. Generic cross sections can be considered, providing a valuable tool for the general analysis of structures subjected to elevated temperatures. Thus, it is concluded that the computational and simulation strategies adopted are able to accurately assess the behavior of structures subjected to fire. However, there is still considerable room for improvement of the program by generalizing the underlying mathematical principles of the algorithm in order to expand the applicability of the computational modules.

References

BARROS, R. C. ***Avaliação numérica avançada do desempenho de estruturas de aço sob temperaturas elevadas***. 2016. 134 f. Dissertação (Mestrado em Engenharia Civil) – Escola de Minas, Universidade Federal de Ouro Preto, Ouro Preto, 2016.

BARROS, R. C.; PIRES, D.; SILVEIRA, R. A.; LEMES, I. J.; ROCHA, P. A. Advanced inelastic analysis of steel structures at elevated temperatures by SCM/RPHM coupling. ***Journal of Constructional Steel Research***, v. 145, p. 368–385, 2018.

CALDAS, R. B. ***Análise numérica de estruturas de aço, concreto e mistas em situação de incêndio***. 2008. 226 f. Tese (Doutorado em Engenharia de Estruturas) – Escola de Engenharia, Universidade Federal de Minas Gerais, Belo Horizonte, 2008.

EUROPEAN COMMITTEE FOR STANDARDIZATION. ***EN 1991-1-2***: Actions on structures - part 1-2: general actions - actions on structures exposed to fire. London: CEN, 2002. 59 p.

EUROPEAN COMMITTEE FOR STANDARDIZATION. ***EN 1992-1-2***: Design of concrete structures - part 1-2: general rules-structural fire design. London: CEN, 2004. 225 p.

EUROPEAN COMMITTEE FOR STANDARDIZATION. ***EN 1993-1-2***: Design of steel structures - part 1-2: general rules - structural fire design. London: CEN, 2005. 91p.

FRANSSEN, J. M. ***Etude du comportement au feu des structures mixtes acier-béton***. 1987. 276 f. PhD Thesis (Docteur en Sciences Appliquées) – Faculté des Sciences Appliquées, Université de Liège, Liège, Belgium, 1987.

FRANSSEN, J. M. SAFIR: A thermal/structural program for modeling structures under fire. ***Engineering Journal American Institute of Steel Construction Inc***, v. 42, p.143–158, 2005.

GEUZAINE, C.; REMACLE, J. F. Gmsh: a 3-d finite element mesh generator with built-in pre-and post-processing facilities. ***International Journal for Numerical Methods in Engineering***, v. 79, n. 11, p. 1309–1331, 2009.

HUANG, Z.; BURGESS, I. W.; PLANK, R. J. A non-linear beam-column element for 3D modelling of general cross--sections in fire. ***Research Rep. Nº. DCSE/03/F/1***. Sheffield, UK: Department of Civil and Structural Engineering, University of Sheffield, 2003.

IU, C. K. J. ***Numerical simulation for structural steel member or framed structure at elevated temperature***. 2004, 275 f. Thesis (Doctor of Philosophy) - Department of Civil and Structural Engineering, Hong Kong Polytechnic University, Hong Kong, 2004.

LANDESMANN, A. ***Modelo não-linear inelástico para análise de estruturas metálicas aporticadas em condições de incêndio***. 2003. 295 f. Tese (Doutorado em Engenharia Civil) - COPPE, Universidade Federal do Rio de Janeiro, Rio de Janeiro, 2003.

LANDESMANN, A.; BATISTA, E. D. M.; ALVES, J. L. D. Implementation of advanced analysis method for steel--framed structures under fire conditions. ***Fire safety journal***, v. 40, n. 4, p. 339–366, 2005.

LEE, S.-L.; MANUEL, F. S.; ROSSOW, E. C. Large deflections and stability of elastic frame. ***Journal of the Engineering Mechanics Division***, v.94, n. 2, p.521–548, 1968.

MARAVEAS, C. Local buckling of steel members under fire conditions: a review. ***Fire Technology***, v. 55, n. 1, p. 51-80, 2019.

MAXIMIANO, D. P. ***Análise numérica avançada de estruturas de aço e de concreto armado em situação de incêndio.*** 2018. 197 f. Tese (Doutorado em Engenharia Civil) – Escola de Minas, Universidade Federal de Ouro Preto, Ouro Preto, 2018.

MCNAMEE, B. M.; LU, L.-W. Inelastic multistory frame buckling. ***Journal of the Structural Division***, v. 98, n. 7, p. 1613–1631, 1972.

MOUÇO, D. L. ***Modelo inelástico para análise avançada de estruturas mistas aço-concreto em situação de incêndio.*** 2008, 129 f. Dissertação (Mestrado em Engenharia Civil) - COPPE, Universidade Federal do Rio de Janeiro, Rio de Janeiro, 2008.

NEVES, N. S. ***Modelo computacional avançado para análise de estruturas sob ação de gradientes térmicos.*** 2019. 277 f. Dissertação (Mestrado em Engenharia Civil) – Centro Tecnológico, Universidade Federal do Espírito Santo, Vitória, 2019.

NEVES, N. S.; AZEVEDO, M. S.; BARCELOS, C. B.; SILVA, V. P.; PIERIN, I. Estudo térmico de pilares mistos de aço e concreto de seção circular em situação de incêndio. ***REA- Revista da Estrutura de Aço***, v. 9, p. 122-140, 2020.

NEVES, N. S.; CAMARGO, R. S.; AZEVEDO, M. S. Advanced computer model for analysis of reinforced concrete and composite structures at elevated temperatures. ***Revista IBRACON de Estruturas e Materiais***, v. 14, n. 4, 2021.

PIERIN, I.; SILVA, V.; LA ROVERE, H. Thermal analysis of two-dimensional structures in fire. ***Revista IBRACON de Estruturas e Materiais***, v. 8, n. 1, p. 25–36, 2015.

PIRES, D.; BARROS, R. C.; ROCHA, P. A. S.; SILVEIRA, R. A. D. M. Thermal analysis of steel-concrete composite cross sections via CS-ASA/FA. ***REM-International Engineering Journal***, v. 71, n. 2, p. 149–157, 2018.

PRAKASH, P. R.; SRIVASTAVA, G. Nonlinear analysis of reinforced concrete plane frames exposed to fire using direct stiffness method. ***Advances in Structural Engineering***, v. 21, n. 7, p. 1036–1050, 2018.

PURKISS, J. A.; LI, L. Y. ***Fire safety engineering design of structures.*** 3rd. ed. New York: CRC Press, 2013. 452 p.

REDDY, J. N. ***An introduction to nonlinear finite element analysis***: with applications to heat transfer, fluid mechanics, and solid mechanics. 2 nd. ed. Oxford: Oxford University Press, 2015. 768 p.

RIBEIRO, J. C. L. ***Simulação via método dos elementos finitos da distribuição tridimensinal de temperatura em estruturas em situação de incêndio.*** 2004. 178 f. Dissertação (Mestrado em Engenharia de Estruturas) – Escola de Engenharia, Universidade Federal de Minas Gerais, Belo Horizonte, 2004.

RIBEIRO, J. C. L. ***Desenvolvimento e aplicação de um sistema computacional para simulação via método dos elementos finitos do comportamento de estruturas de aço e mistas em situação de incêndio.*** 2009. 260 f. Tese (Doutorado em Engenharia de Estruturas) – Escola de Engenharia, Universidade Federal de Minas Gerais, Belo Horizonte, 2009.

RIGOBELLO, R. ***Desenvolvimento e aplicação de código computacional para análise de estruturas de aço aporticadas em situação de incêndio.*** 2011. 272 f. Tese (Doutorado em Engenharia de Estruturas) – Escola de Engenharia de São Carlos, Universidade de São Paulo, São Carlos, 2011.

RUBERT, A.; SCHAUMANN, P. Structural steel and plane frame assemblies under fire action. ***Fire Safety Journal***, v. 10, n. 3, p. 173–184, 1986.

SOUZA JUNIOR, V.; CREUS, G. Simplified elastoplastic analysis of general frames on fire. ***Engineering structures***, v. 29, n. 4, p. 511–518, 2007.

THAI, H.T; KIM, S. E.; KIM, J. Improved refined plastic hinge analysis accounting for local buckling and lateral-torsional buckling. ***Steel and Composite Structures***, v. 24, n. 3, p. 339-349, 2017.

WANG, Y.; MOORE, D. Steel frames in fire: analysis. ***Engineering Structures***, v. 17, n. 6, p. 462-472, 1995.

WICKSTRÖM, U. ***TASEF-2: a computer program for temperature analysis of structures exposed to fire.*** Lund, Sweden: Department of Structural Mechanics, Lund Institute of Technology, 1979. (Report / Lund Institute of Technology, Department of Structural Mechanics; Vol. 79-2).

YANG, Y. B.; KUO, S. R. ***Theory & analysis of nonlinear framed structures.*** Singapore: Prentice Hall, 1994. 450 p.

Received: 22 February 2021 - Accepted: 6 August 2021.