XLIII CILAMCE
21-25, November, 2022
FOZ DO IGUAÇU - PR - BRAZIL

Study of the high temperature effects on steel structural elements via Finite Element Method

Julia A. Freitas [1]

[1] *Dept. of Civil Engineering, Federal University of Uberlândia*
Av. João Naves de Ávila, 2121, 38400-90, Uberlândia - MG, Brazil
juliaravequia@ufu.br

Abstract. This article addresses the analysis of the steel behavior, as a structural material, subjected to high temperature and then cooled. This study began with the influence of the cooling method on steel elements at high temperatures questioning, considering the impact this research can bring to day-to-day fire situations. Thus, to have a better understanding of the consequences of thermal actions on steel, tests were carried out on the structural parts, under conditions like those of material characterization, at room temperature and after cooling, using immersion in water and cooling without interference. To validate the proposed constitutive model, the results obtained are compared by the numerical analysis of finite elements. Such simulation was performed using the Abaqus software and considering the refinement of the mesh of the elements for better data accuracy. Thus, the results achieved in the modeling were in accordance with the results obtained experimentally, so that the numerical model was validated.

Keywords: high temperatures, steel elements, numerical analysis, finite elements.

1 Introduction

In steel-structured buildings, the occurrence of fire situations represents one of the most harmful actions that can lead to structure instability, since the mechanical properties of steel can be significantly altered under high temperatures. Linked to this, steel is a material that has high thermal conductivity, which can lead to a sudden increase in temperature and a consequent shock in the strength of the structure. The rise in temperature contributes to the depreciation of the strength and rigidity of the structural elements, contributing to the partial or global collapse of the building, justified by the decrease in the modulus of elasticity and the yield strength of the steel. Thus, it is essential that there is a study on the behavior of steel under the influence of high temperatures, to improve the structural design processes under these conditions.

The early reduction of the tensile strength and rigidity of a structural element, even for isolated parts, may not guarantee the total evasion of the building for the minimum time necessary to perform this task safely. Considering it, the use of advanced computational techniques, such as those based on the Finite Element Method, is justified to analyze the thermomechanical behavior of isolated elements in a fire situation, contributing to the academic understanding of the behavior of this material under the conditions described, and still collaborating for a possible revision of the standard fire curve, used in the regulations.

In the case of this work, the ABAQUS/Standard software was chosen due to its modeling capacity and versatility, and also, experimental tests for calibration and validation of the numerical models were performed. However, it is understood the difficulty in carrying out experiments and tests on a large scale, which hinders the global investigation of the materials behavior in the building.

2 Bibliographic Review

A synthesis of works on thermal analysis of steel structures, studied in the literature review for this dissertation, is presented below.

ZHAO and SHEIN [1] carried out a theoretical-experimental study of the behavior of steel frames without fire protection, subjected to different levels of loading, temperature, and boundary conditions. Among the main conclusions, they observed that the temperature distribution in the steel profiles is quite non-uniform under real fire conditions and that the temperature heating rate influences the fire resistance of steel structures, and faster the heating higher the limit temperature.

MARTINS [2] includes in his work the foundations of NBR-14323 (1999) for the design of structural steel elements in a fire situation, which includes the characteristics of fires, the concepts related to actions and safety, obtaining the required fire resistance time (TRRF), the types of fire protection that may involve the structural elements, the procedures for obtaining the temperature rise in the elements and the methods for obtaining the design resistances. A program was also developed for dimensioning tensioned, compressed, flexed bars, and subjected to combined stresses in a fire situation and at room temperature.

SILVA [3] deals with steel structures in fire situation, determination of temperature in structural elements, safety, and behavior of structures in fire situation and the simplified method of design. Silva also discusses thermal protection materials, sizing procedures for structures in a fire situation and provides some basic concepts about reinforced concrete structures in a fire situation.

FERNANDES [4] presented a computational tool based on Finite Element Methods, for non-linear geometric analysis of flat steel frames under fire. Analyzes of isolated structural elements and plane frames were performed, discretizing the structures into bar elements. The numerical results of structural analysis of isolated bars and of flat framed structures are presented and compared with the results obtained by the computer program SAFIR.

So far, all these scientific investigations have been based on theoretical concepts and computational tools of structural analysis. This is due to the high cost of equipment, materials and labor needed to carry out tests.

3 Research Method

The project was developed in two fronts, the first being the realization of experimental tests carried out in the laboratory, with the objective of providing data for the validation of the numerical model, second front. To simulate the high temperatures, an electric oven (Figure 1) was used for this task.

Figure 1: Electric oven

3.1 EXPERIMENTAL TESTS

The specimens tested in the laboratory were divided into three groups, containing two samples each. The first group, called test A, was directly subjected to the tensile test in the material characterization machine.

The other two groups were identified and differentiated between tests B and C, being submitted to the temperature increase inside the oven, before the material characterization. Both differ from each other by the cooling method. The specimens from group B were cooled to room temperature, while group C was immersed in water for 5 minutes after thermal exposure.

Three K-type thermocouples (GG-K-24-25) were positioned inside the equipment to follow the heating curve provided by the four resistors arranged inside the electric oven. *Catman* software was used to monitor the measurement of thermocouples and enable the preparation of the heating curve *easy*.

Despite the arrangement of three thermocouples, only two provided valid data, so we will disregard the thermocouple called sensor 2 (Figure 2).

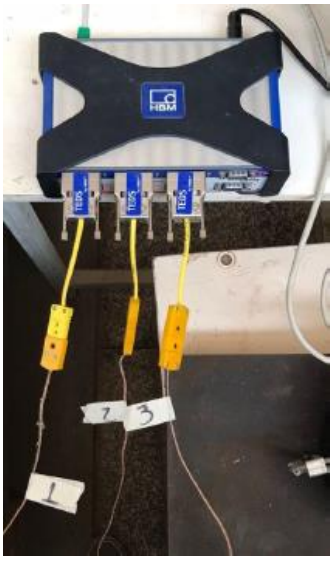

Figure 2: Temperature measurement equipment

Figure 3 shows the graph of the temperature, in °C, by the time, in seconds, inside the oven recorded by the thermocouples. As can be seen, the maximum temperature reached by the oven, recorded by sensor 1, was 355 °C. Sensor 3, due to its position inside the oven, recorded a slightly lower temperature of 348 °C.

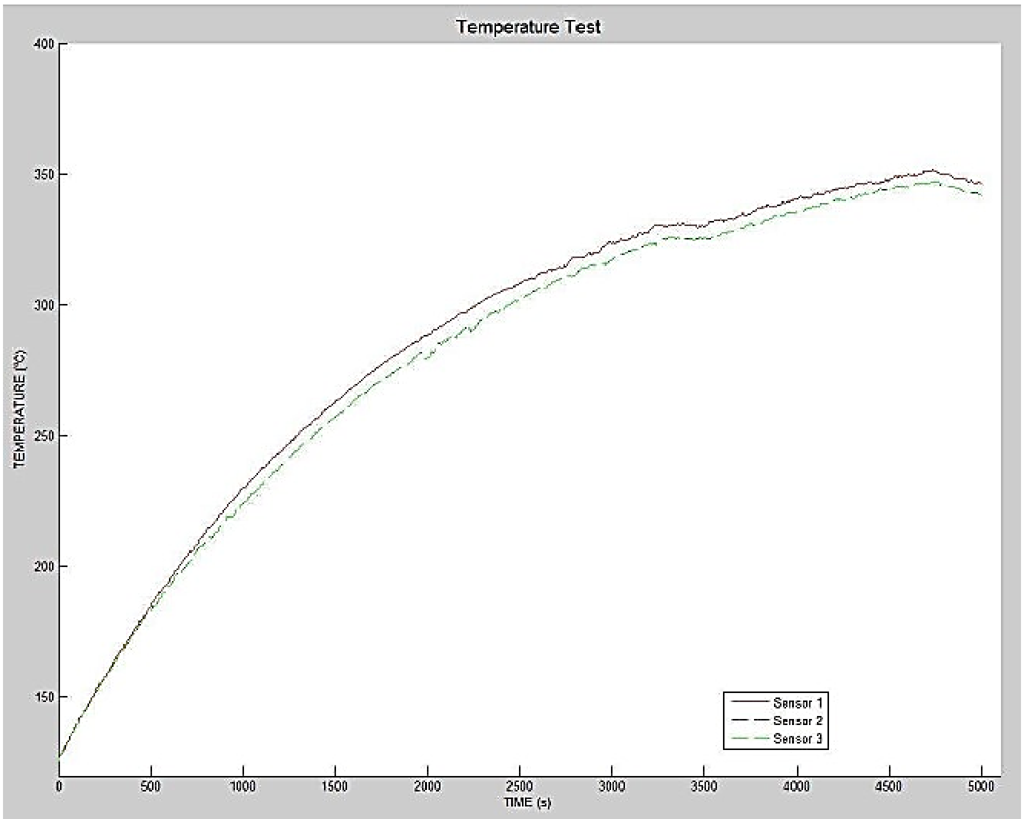

Figure 3: Temperature (°C) x time (s) graph inside the oven

Initially, there were 20 specimens, numbered and identified from 1 to 20. Four were used in tests, fourteen were duly tested and with results considered, and two were left in reserve, as shown in Table 1.

To compare the results, the groups were placed inside the oven as soon as it reached 250 °C and the timing started when it reached 300 °C. Exposure times varied by 10 minutes, from 5 minutes to 55 minutes (Table 1).

Table 1. Identification of specimens and test description

Specimen	Test	Exposure Time (min)
1	A	-
2	A	-
3	A	-
20	A	-
16	A	-
17	A	-
4	B	5
5	C	5
8	B	15
11	C	15
12	B	25
7	C	25
10	B	35
13	C	35
6	B	45
15	C	45
14	B	55
9	C	55
18	BACKUP	-
19	BACKUP	-

For comparison between the groups, we have chosen, due to good data collection, specimens 17, 4 and 5 (Figures 4).

Figure 4: Specimen belonging to group A, B and C

Thus, the results presented in Table 2 were experimentally and analytically obtained, which main characteristics used for comparison were the modulus of elasticity, yield stress, and ultimate strain.

Table 2. Properties of bodies exposed to high temperatures

Specimen	4	5
Young modulus (Mpa)	13179	12579
Poisson's ratio	0,25	0,25
Yield Stress (Mpa)	164	162
Ultimate Tensile Strength (Mpa)	174	172
Ultimate Strain (mm/mm)	0,23	0,22
Maximum displacement (mm)	0,22	0,22

After heating and cooling, the specimens were submitted to the material characterization test. Following, there are the three specimens graphs (Figure 5, 6 and 7).

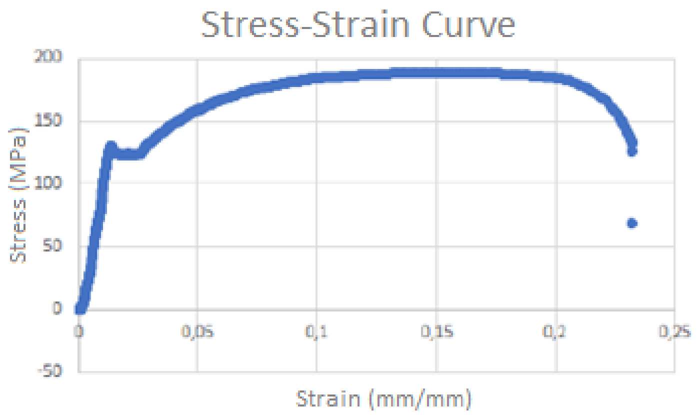

Figure 5: Stress x strain curve - CP 17

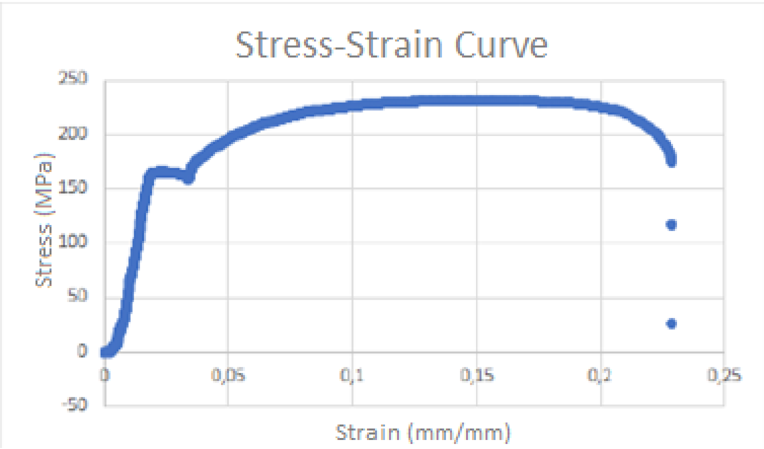

Figure 6: Stress x strain curve - CP 4

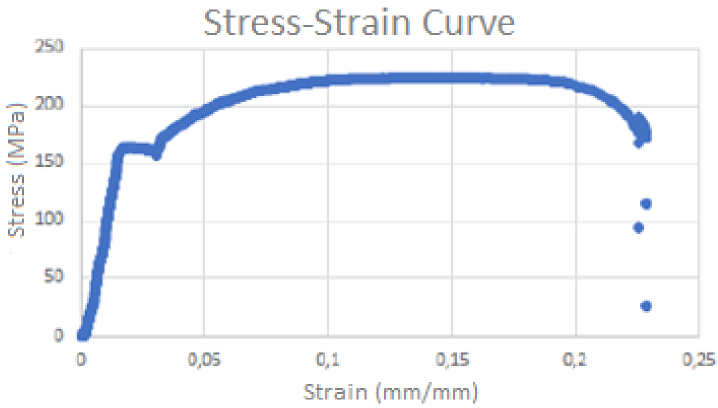

Figure 7: Stress x strain curve - CP 5

The three specimens presented a stress-strain curve very characteristic of the steel material, and even with stresses very close to each other, regardless of the temperature faced or cooling mode.

Furthermore, comparing the graph of specimen 17 (without heating) with the graphs of specimens 4 and 5 (heated and cooled to room temperature and by submersion, respectively), it is possible to conclude that the steel when heated to the temperature of 350°C, free of loading during the test, presents a negligible variation in the material behavior.

For data validation and model calibration, specimen 17 was chosen, submitted to test A, that is, sample tested in the characterization machine without prior heating. Below are the properties of specimen 17.

Table 3. Properties of specimen 17

Young modulus (Mpa)	11643
Poisson's ratio	0,22
Yield Stress (Mpa)	150
Ultimate Tensile Strength (Mpa)	216
Ultimate Strain (mm/mm)	0,18
Maximum displacement (mm)	0,23

3.2 NUMERICAL SIMULATION OF TRACTION VIA FINITE ELEMENT METHOD

As previously mentioned, the modeling process was carried out using the ABAQUS software. This program is a powerful tool, based on the finite element method, which can be used in the simulation of different natures problems, among them: dynamic, electromagnetic, thermal, and electrical events, for example. However, due to the challenge of performing the simulation under thermal effect, the numerical modeling in this article was performed only including the stress-strain relationship obtained experimentally.

The ABAQUS simulation process is carried out in three stages: pre-processing, simulation, and post-processing. Pre-processing consists of defining the physical model input data. In this step, some initial conditions are defined, such as the model geometry, materials, section properties, assembly of the parts (if the model does not consist of only one part), the boundary and loading conditions, the mesh, among other conditions. Special care must be taken with the units used and their compatibility, since the software works with specific units. In the case of this work, the units Newton, millimeter and second were used.

Simulation is the problem solving step. In this step, the resolution algorithm to be used is decided, which may belong to the ABAQUS/ *Explicit* or ABAQUS/ *Standard* . The choice of solver to be used will depend on the phenomenon to be simulated. In the case of this work, we have chosen to work with the implicit model of the Standard method, as it is a less complex simulation. Finally, post-processing consists of analyzing the modeling results. ABAQUS allows a detailed and diversified view where it is possible to analyze deformed configurations, contours, animations, graphics, among others. The entire process of collecting the results is carried out in the *Visualization tab* .

After defining the part geometry, the boundary and loading conditions were applied as pressure on the 3mm face of the table region. However, in order to obtain more accurate results, we opted for the displacement control method, which consists of applying the load at only one point. For this, the *Interaction* option was used, the rigid body constraint was created, and a reference point was used to apply all the loading on one face of the table region of the specimen, while the opposite end remained fixed. This method helped to reduce the errors in the model (Figure 8).

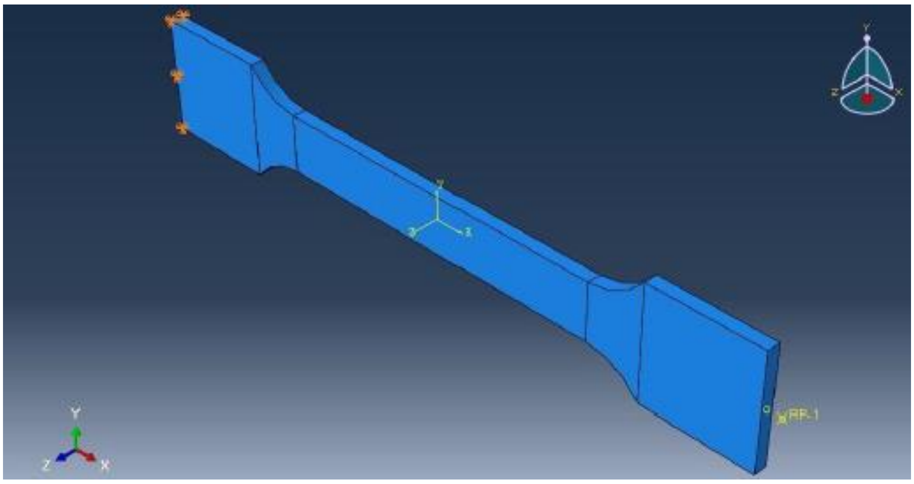

Figure 8: Model with boundary conditions and loading

Thus, a displacement of 23 mm was attributed to the reference point, according to the maximum displacement obtained in the stress-strain curve of specimen 17 (Table 3). In this way, the force obtained was not coincident with the applied load steps, but with the reaction force that prevents this applied displacement. When the displacement control method is adopted, the model acquires a lower rigidity, which makes it closer to reality. When validating a model, it is important to check whether it is better to apply force control or displacement, given that there are cases in which force control brings the model closer to reality, which was not the case in this work.

Then, the mesh was applied to the model, always being careful about the mesh refinement to obtain accurate results that were as close as possible to the experimental results (Figure 9). The mesh was made with 357 elements and the type applied was the C3D8R, which is one of the best elements of ABAQUS, both in terms of computational cost and in terms of results, since it usually generates satisfactory results. The refinement used in the mesh was 3mm, which was considered a good size based on the comparison of the numerical result with the experimental one.

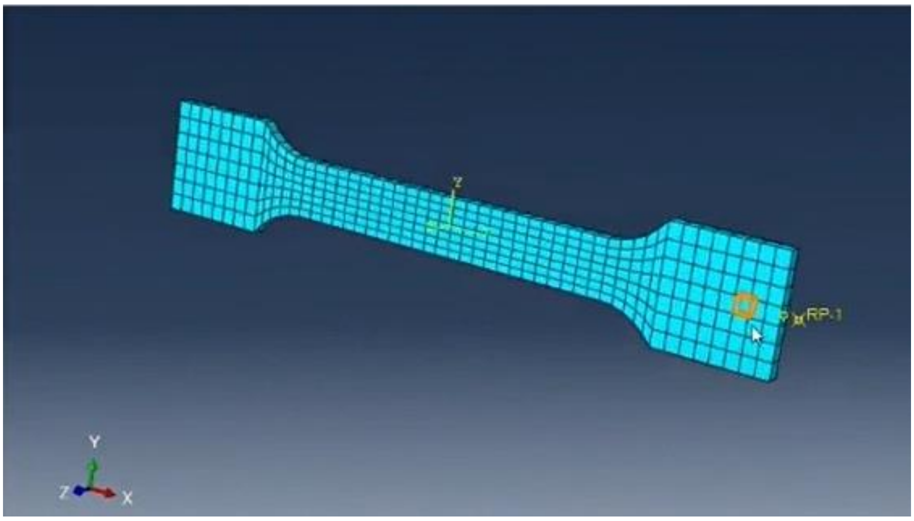

Figure 9: Mesh applied to the model

After refining the mesh and the boundary conditions, the post-processing stage was carried out. In this step, it was possible to analyze whether the results obtained were accurate and in accordance with the results obtained in the experimental tests. In the analysis of the stress obtained in the modeling, it was verified that the ultimate stress of 216 MPa was reached, in the region of the smallest section, as seen in Figure 10.

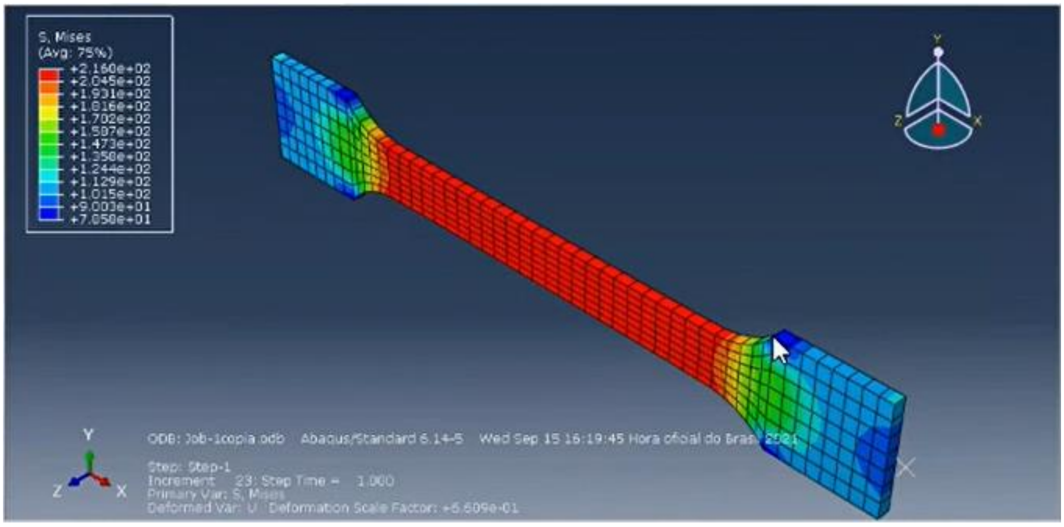

Figure 10: Results obtained

Comparing with the result achieved in the laboratory, it was verified that the model arrived at the same ultimate stress. With this result, it was possible to conclude that the model was validated.

4 Conclusion

It is known that steel structures under the effect of temperature increase have reduced elastic modulus and yield stress. However, how affected are these properties and at what temperature? This question outlined the objective of this work, which consisted of contributing to the academic understanding of changes in the steel material properties under the influence of high temperatures. The biggest limitation faced was the structural inability of the laboratory to carry out the tests of heating and traction simultaneously. After analyzing the experimental results, can be concluded that the variation in the behavior of the material, at the temperature reached and free of loads during the test, is negligible.

Simulating the thermomechanical behavior of steel structures using software such as ABAQUS, which requires knowledge of the Finite Element Method, the performance of materials under high temperatures, in addition to advanced modeling techniques, is challenging and complex.

In this context, this first part of the research was limited to performing a numerical simulation of the tensile test and finding a valid numerical model, which would obtain results consistent with reality. Thus, with the model calibrated from the experimental data, the research sequence will consist of a deeper study of the ABAQUS software and the simulation sequence applying the thermomechanical behavior.

5 References

[1] ZHAO, J.C. and SHEN, Z.Y. Experimental Studies of the Behavior of Unprotected Steel Frames in Fire. Journal of Construction Steel Research 50. p137-150. Elsevier Science Publishers Ltd. London - UK. 1999

[2] MARTINS, M. M. Design of steel structures in fire situation. Masters dissertation. UFMG. Belo Horizonte - MG. 2000

[3] SILVA, V. P. Steel structures in fire situation. Ziggurat Publisher. Sao Paulo-SP. 2001.

[4] FERNANDES, Danilo de Hollanda. Nonlinear Elastic Analysis of Flat Steel Frames Under Fire. Rio de Janeiro, 2004 IPQ. NP EN 1993-1-2: Eurocode 3 - Design of steel structures. Part 1-2: General Rules. Fire resistance check. Caparica: Portuguese Institute of Quality, 2010. 2, 3