Evaluation of the Cross-Section of Support Construction Elements in Wood with Gypsum Submitted to Fire on One Side

Domingos A. M. Pereira[1], Elza M. M. Fonseca[1]

[1]*ISEP, Instituto Politécnico do Porto*
R. Dr. António Bernardino de Almeida, 4249-015 Porto, Portugal
1180819@isep.ipp.pt, elz@isep.ipp.pt

Abstract. In this work, different numerical tests are presented, for the assessment of the thermal and transient analysis, of different constructive elements in wood, with gypsum protection, in fire conditions. This type of constructive element presents cavities, making the numerical analysis in the study complex in the approximation to real models. The analysis was carried out considering the nonlinearity of the materials. The objectives were to investigate which is the best numerical model, using the finite element method, which addresses experimental tests carried out by other authors and to be used in different parametric models that can be used by designers to prevent fire resistance in these components.

Keywords: fire, protection, char layer, wood.

1 Introduction

The constructive wood elements are protected with gypsum, such as inner partitions, inner boards of outer walls and ceilings, for example. This type of construction is fast and designed as light wood. These elements are submitted to certification, which is the fire resistance classification of materials according to their integrity and insulation [1-3]. Wood presents some limitations because it is combustible in the presence of fire. Then, the recourses to protected elements with insulating properties which do not allow direct exposure to fire and delay the increase in temperature, are very important in their use. The constructive components in the study are unloaded elements with one side exposed to fire, such as inner wood-stud walls, combined with solid wood beams and gypsum board used to protect them or other insulation materials. The internal air cavity, formed by the insulating boards, may or may not be filled with insulating material. The fire resistance of these elements depends on the imposed protection. The fire resistance rating is defined in accordance with the safety laws against fire in buildings RT SCIE (DL220 of 2008) [4]. Fire resistance is a measure for a specified time under fire conditions of a standard heat, in which the constructive elements have a performance to develop the purposes for which are designed. The calculation of the temperature distribution of this type of constructive and protected elements when subject to fire, can be carried out both through experimental and numerical tests, as well as through analytical and simplified models.

In this work, different numerical models are developed, compared with one experimental test (Test 4) referenced by Takeda and Mehaffey [5]. It is intended to validate our numerical model, using the finite element method with ANSYS® to enable the study of new parametric models applied to the project. The results of the temperature field were obtained, for any measurement points, the temperature profile for the end time instant of the exposure fire, and the residual wood cross-sections for different time instants.

2 Experimental Model

To analyze the experimental model presented by Takeda e Mehaffey [5], the properties of wood and gypsum materials were used, considering the density and thermal conductivity close to the study, and imposing the non-linear variation according to Eurocode 5, part 1-2 [6]. The authors [5] presented different experimental tests, but only Test 4 will be used to compare our numerical solution, with 2 gypsum board type X with 15.9 mm thickness and density equal to 648 kg/m^3. The wood material was considered in glulam GL24H and the gypsum board as type A, both according to the referenced Test 4 [5], as shown in Figure 1.

The obtained temperature fields at different measured points by the authors [5] were analysed, with special interest at the interface points between the wood and the gypsum board, and another point between the air cavity and the wood element. The end temperature fields at the last time of fire exposure were also observed. Simultaneously, the residual wood cross-sections will be compared for instants of time equal to 90, 100, and 110 min.

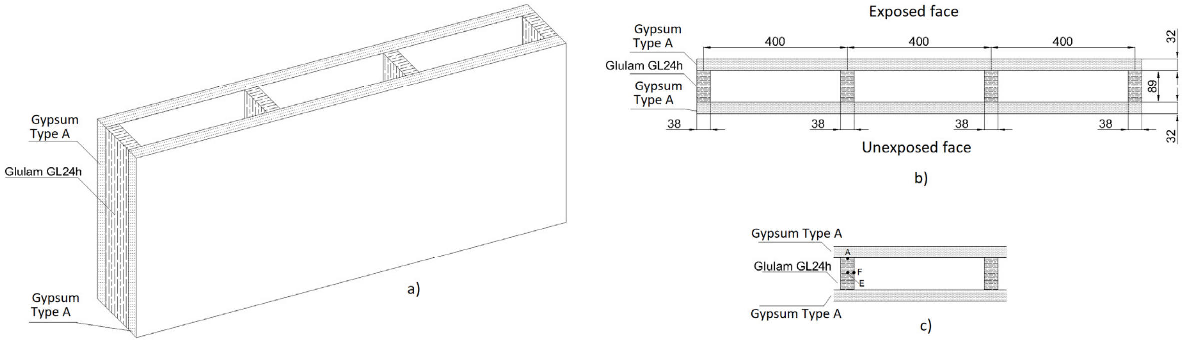


Figure 1. a) Constructive model, b) Measuring points; c) Dimensions.

3 Numerical Model

In the development of the numerical models, different hypotheses were considered regarding the cavities of the constructive model: M1 - model with air mesh (considered as a solid model allowing the conduction of heat inside the cavity); M2 - model with radiation effect by the introduction of a new finite element SURF151, inside the cavity; M3 - model with radiation and convection boundaries in the cavity, but introducing a test curve Tf-Test4, representative of the temperature evolution inside the cavity; and M4 – model with radiation effect with the finite element SURF151 and the application of convection with LINK34 element. In the other parts of the constructive model, the two-dimensional (2D) element PLANE55 was used. In the numerical modelling, the finite elements used were:

PLANE55 - 2D Thermal Solid Element, can be used as a plane element for thermal conduction capability. The element has four nodes with a single degree of freedom, temperature, at each node. The element is applicable to a 2D, steady-state, or transient thermal analysis [8-9].

SURF151 - Thermal Surface Effect, used for various load and surface effect applications. It may be overlaid onto the face of any 2D thermal solid element. It allows the radiation between the inner surfaces of the cavity and any point inside it. The SURF151 element consists of two nodes, associated with an extra node, which is located inside the cavity.

LINK34 - Convection Link Element, is a uniaxial element with the ability to conduct heat between its nodes. The element has a single degree of freedom, temperature, at each node point. The convection element is applicable to 2D or 3D, steady-state, or transient thermal analysis. The element is defined by two nodes, a node on the surface and another one inside the cavity, a convection surface area, two empirical terms, and a film coefficient [8].

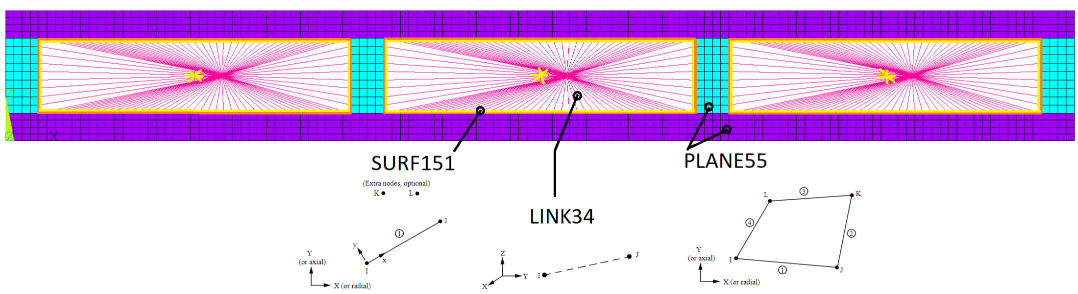


Figure 2. Mesh and finite elements, model M4.

Figure 2 represents model M4, with all finite elements chosen. The SURF151 and LINK34 elements allow the application of thermal boundaries, by overlapping the 2D PLANE55 element. By using surface effect elements with the radiation option for radiating between a surface and a point, the form factor between a surface and the point can be specified as a real constant or can be calculated from the basic element orientation and the extra node location.

Figure 3 represents the meshes developed of the different numerical models and their boundary conditions.

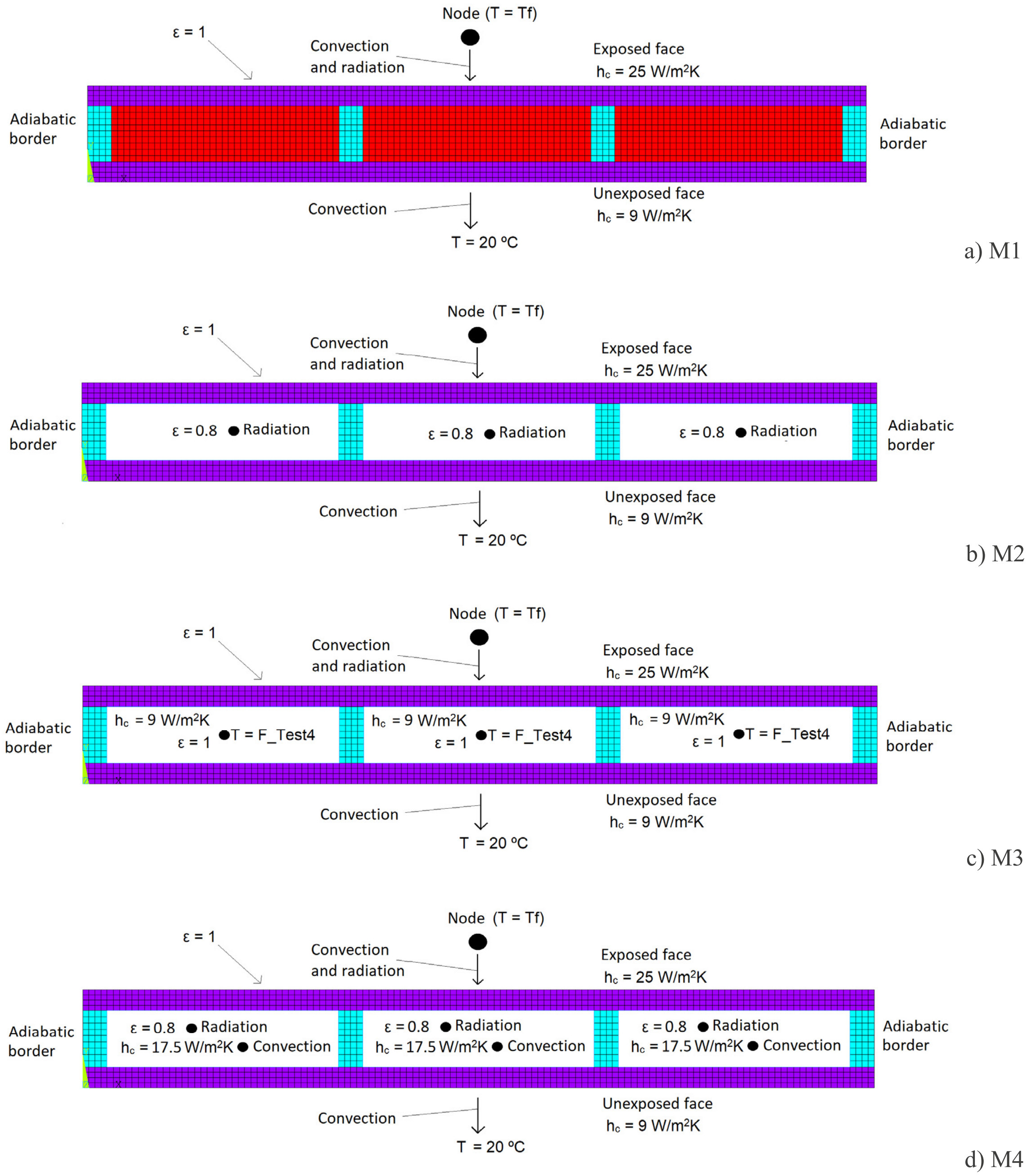
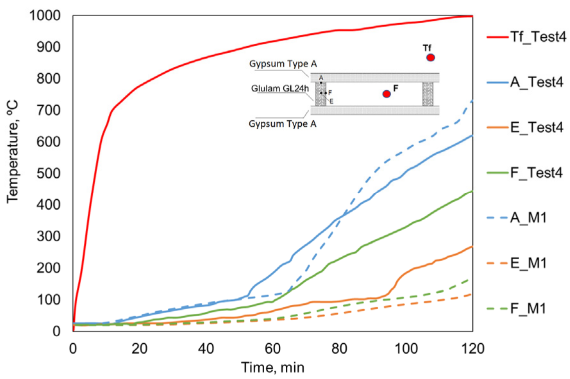


Figure 3. Meshes and boundary conditions. Cavity with: a) M1 - air mesh; b) M2 - radiation; c) M3 - heating curve; d) M4 – radiation and convection.

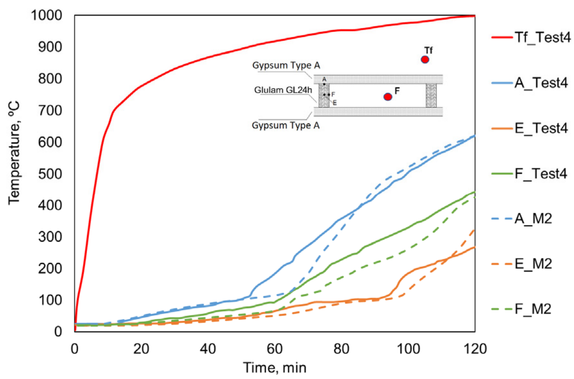
To elaborate the finite element mesh, the perfect contact was assumed between all the materials, to allow the thermal energy conduction between them, with the element edge dimension equal to 10 mm. The thermal characteristics associated with wood GL24H, the gypsum pasteboard type A, and the air, are according to references [6-7], [10], to approximate the materials involved in Test 4 [5]. On the exposed fire surface, the boundary conditions are radiation and convection, with the effect of increased temperature by introducing the experimental fire curve Tf. The initial conditions of the model correspond to an ambient temperature of 20˚C. The relative emissivity properties of the material for wood were 0.8 and 0.85 for gypsum plasterboard. On the unexposed face, only the convection was considered. In the lateral edges of the numerical model, an adiabatic condition was assumed, without heat exchanges between these zones. All these conditions comply with Eurocode 1 part 1-2 [1]. To satisfy the non-linear conditions of the problem, the program uses the Newton-Raphson iterative method, with a convergence criterion based on the heat flux with a tolerance equal to 0.9. The total time of each simulation corresponds to a fire exposure of 120 min.

4 Results and Discussion

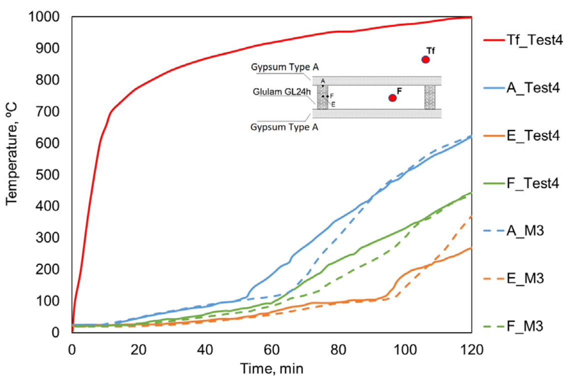
Based on the analysis carried out on each model, the results of the temperature evolution at the different nodal points are presented, according to those obtained experimentally (Test 4), Figure 4.



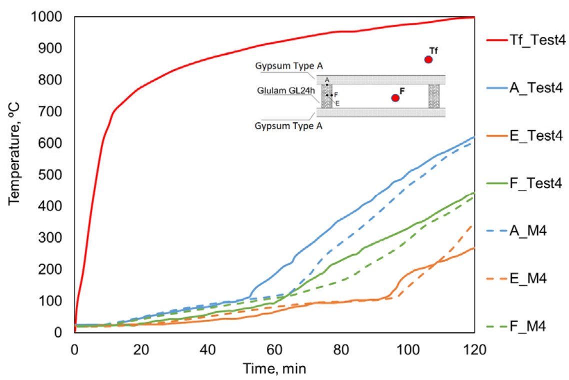
a) M1



b) M2



c) M3



d) M4

Figure 4. Temperature field results. Cavity with: a) M1 - air mesh; b) M2 - radiation; c) M3 - heating curve; d) M4 – radiation and convection.

Figure 4 represents the temperature evolution curves at the different nodal points of the numerical models, compared with Test 4 of the experimental test (A, E, F). All graphics represent the real fire curve used in the laboratory Tf_test4. In the M3 model, the heating curve considered in the cavity was the F_Test4 curve obtained experimentally in the middle of the internal wooden component. Curve A is the one with the highest temperature field because it is located between the gypsum plasterboard and the wood interface. Curve E is the one with the lowest temperature profile because it is located inside the wood component. Regarding the comparison of results, M1 and M2 model present the great variation of results. M4 is the model that has a behavior closer to the experimental one, approaching the results obtained with M3 model, and both can validate the results of the test used [5].

The temperature field obtained at the final instant, for each of the models under study, is represented in Figure 5.

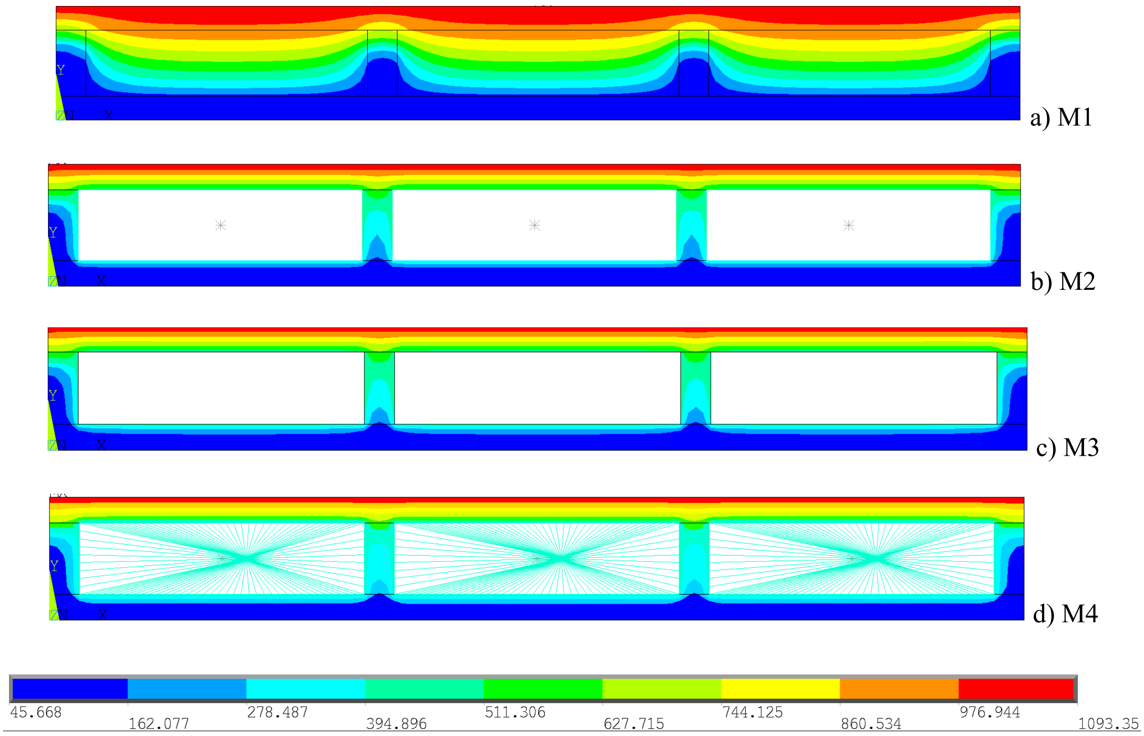


Figure 5. Temperature field for the final exposure instant (120 min). Cavity with: a) M1 - air mesh; b) M2 - radiation; c) M3 - heating curve; d) M4 – radiation and convection.

It is observed that the temperature profile between models M2 and M3 are very close, thus demonstrating the importance of the boundary conditions inside the cavity. M4 is the model which represents a temperature profile with more pronounced heating effect through the wood components. The M1 model, being for conduction, does not fully represent the characteristic test on this type of component.

The models were also analyzed according to the evolution of the char layer over time for the instants of 100 and 110 minutes. The wood profile used to analyze the char layer was in a cross-section of the central model, to obtain results for the most critical region. The limit temperature considered for the formation of the char layer, imposed by the authors, was 288 ˚C. The comparison of the char layer evolution of the experimental test was carried out with numerical models M2 - M3 - M4, as shown in Figure 6.

With the results, it is verified that the char layer in the numerical model presents an evolution close to that obtained in the experimental test by Takeda and Mehaffey [5]. The model M4 used to compare the results proves to be the one that best represents the evolution of temperatures for this type of constructive model.

To evaluate the char layer and compare their proximity with the results obtained experimentally by the authors [5] the residual wood cross-sections of the central wooden studs was determined. An image tool was used to calculate the charred area in all different models and time instants of fire exposure. Knowing the area of the wood stud, the residual cross-sections were calculated, Table 1.

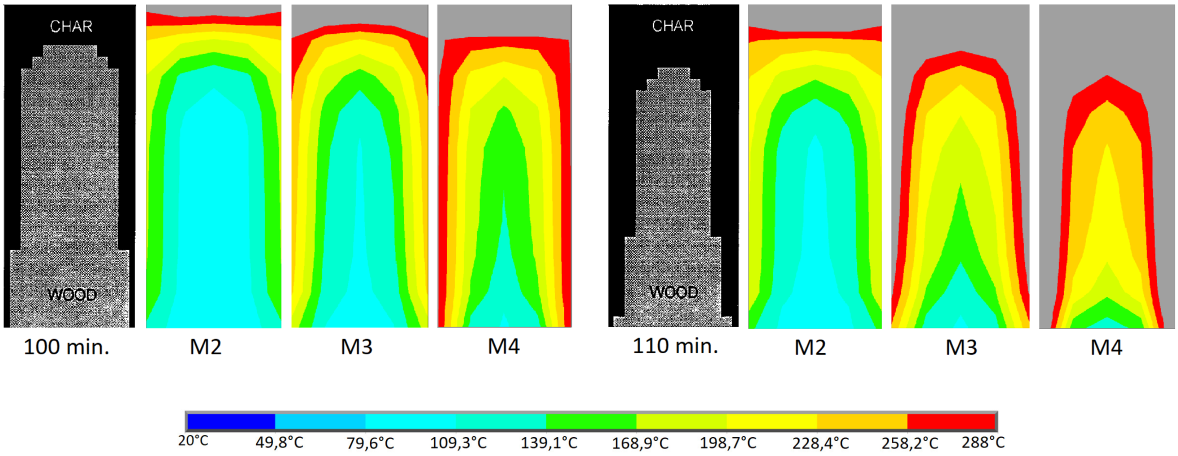


Figure 6. Comparison of char layer evolution with models M2 - M3 - M4.

Table 1. Calculation of the wood residual cross-sections.

Model	Area [mm^2]	Residual cross-sections [%]			
		90 min	100 min	110 min	120 min
Test 4 [5]	3382	81.8	65.7	51.7	25.6
M2	3382	100.0	96.6	92.1	88.2
M3	3382	100.0	92.6	72.6	36.4
M4	3382	96.8	88.3	51.3	13.3

Considering the results obtained in Table 1, it is concluded that the developed model with the effect of radiation and convection in the cavities (M4) presents the closest and most favourable results to Test 4 of the experimental test (5).

To obtain the fire resistance, it is necessary to apply the thermal insulation criterion, defined by the standard EN 1363-1:2020 [1]. The fire resistance attributed by this criterion is defined by the shortest time step, on the unexposed face, based on the maximum temperature T_{max}, that is, 180°C above the initial temperature T_0 of 20°C, or by the mean surface temperature T_{med}, 140°C above the initial temperature. To analyse the temperature evolution, different nodes were defined on the model surface not exposed to fire. For this study, only the model M4 was considered, because it presented the best approximation to the experimental test. Additionally, to reach the limit temperatures imposed by the thermal insulation criterion according to EN 1363-1:2020 [1], other numerical simulation was carried out considering the temperature evolution equivalent to 240 min of fire exposure. The average temperature on the surface not exposed to fire and on the node where the maximum temperature occurs was verified. It was concluded that the fire resistance imposed by the thermal insulation criterion is approximately equal to 216 min. Analyzing the results obtained by Takeda and Mehaffey [5] for Test 4, and based on the computational model, the authors obtained a fire resistance of approximately equal to 226 min. With this, it is concluded that the results obtained for the developed M4 model are close to the results obtained by the researchers, thus validating the numerical model.

5 Conclusions

The finite element method was used for the thermal and transient analysis of support construction elements in wood with gypsum, with internal air cavities, submitted to fire. The temperature field and the residual cross-sections of the wood elements were evaluated. The numerical model was validated and developed by the finite element method and can be used for another type of constructive model to assess fire resistance. In future work, it is intended to investigate the effect of different insulation thickness layers, under the support constructive element under fire, in the assessment of the residual cross-sections. Different parametric analyses will allow us to assess the fire resistance and function of different variables. To study the constructive wood elements not loaded, the fire resistance should be verified in relation to the insulation (I) and to the integrity (E), EN1363-1, ISO834, and EN1364-1.

References

[1] CEN, EN 1363-1: *Fire resistance tests Part 1: General Requirements*, Brussels: CEN- European Committee for Standardization, 2012.
[2] ISO, ISO834-1: Fire-resistance tests - Elements of building construction - Part 1: General requirements. International Organization for Standardization, 1999.
[3] CEN, EN 1364-1: Fire resistance tests for non-loadbearing elements. Part 1: Walls. Brussels: CEN, 2015.
[4] Decreto-Lei nº 220/2008 de 12 de novembro, *Regime Jurídico de Segurança Contra Incêndio em Edifícios* - RJSCIE, Portugal, 2008.
[5] Takeda, H.; Mehaffey, J. R. ''WALL2D: a Model for Predicting Heat Transfer through Wood-Stud Walls Exposed to Fire''. *Fire and Materials*, vol. 22, p. 133-140, 1998.
[6] CEN, EN1995-1-2: Eurocode 5: Design of timber structures. Part 1-2: General Structural fire design, Brussels, 2004.
[7] Carlos Gomes; Elza M. M. Fonseca; Hernâni M. Lopes. ''Thermomechanical Analysis of Steel-to-Timber Connections under Fire and the Material Density Effect''. *Applied Sciences*, vol. 12, p. 10516, 2022.
[8] Chao Zhang and Guo-qiang Li. ''Modified One Zone Model for Fire Resistance Design of Steel Structures''. *Advanced Steel Construction*, vol. 9, no. 4, p. 282-297, 2013.
[9] Ansys® Academic Research Mechanical, 2022 R 2, Help System, ANSYS, Inc.
[10] A. Ujma and N. Umnyakova. "Thermal efficiency of the building envelope with the air layer and reflective coatings," *E3S Web of Conferences*, vol. 100, p. 00082, 2019.