Numerical Modeling of Reinforced Concrete and EPS Core Sandwich Panels under Bending

Geovany F. Barrozo[1], William T.M. Silva[1], Luciano M. Bezerra[1], Jerfson M. Lima[2]

[1]*Dept. of Civil and Environmental Engineering, University of Brasilia*
Brasilia, Federal District, Brazil
geovany.sh75@gmail.com, taylor@unb.br, lmbz@unb.br
[2]*Dept. of Civil and Environmental Engineering, Federal University of Ceará*
Russas, Ceará, Brazil
jerfsonlima2009@hotmail.com

Abstract. The use of construction systems based on the use of Sandwich Panels has grown significantly in the Brazilian civil construction market in recent years, with greater application in residential buildings. However, due to the absence of normative regulation and reliable calculation standards, this system is often used without a fundamental understanding of its structural behavior. Thus, in order to evaluate the resistant capacity of these panels when subjected to bending forces, this work consisted of the elaboration of a three-dimensional numerical model of a Sandwich Panel of reinforced concrete with EPS core and metallic connectors, being used the ABAQUS for its conception, which is a software based on the Finite Element Method (FEM). In addition, the model considers the non-linear behavior of materials, through the use of the Concrete Damage Plasticity model (CDP) and the Plasticity model for steel, which is calibrated and validated by experimental studies present in the literature, presenting results in agreement with those obtained by the experiments.

Keywords: Sandwich Panel, Numerical Modeling, Finite Element Method.

1 Introduction

Sandwich Panels are presented as an alternative to traditional construction systems used in the Brazilian civil construction market, as they add several advantages such as the use of lighter structural elements, allowing the elimination of formworks, better thermal and acoustic insulation, greater agility in the execution, lower cost and greater rationalization of construction, between others (Bertini [1]). Due to these attractions, the application of this type of structure has been increasing in Brazil in recent years, especially in residential buildings, in which they can be used as masonry, floors, and roofs.

According to PCI [2], Sandwich Panels consist of structures composed of layers, which are distributed as two resistant layers separated by another layer of rigid foam plastic insulation, as illustrated in Fig. 1. In addition to this composition, elements responsible for promoting the connection between the layers, called connectors, which can be rigid or flexible and of varied geometry, are also added to the panel. Currently, the Sandwich Panels most used in construction in Brazil have resistant layers of reinforced concrete with welded steel meshes, an insulating layer of Expanded Polystyrene (EPS), and metallic connectors.

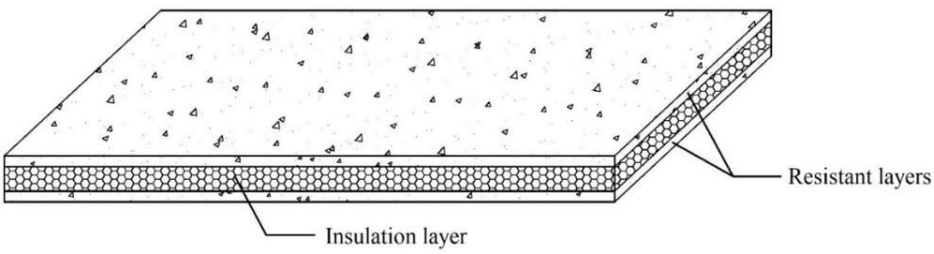

Figure 1. Sandwich Panel

Concerning its structural behavior, Sandwich Panels can be classified as fully composite, partially composite, or non-composite, according to the degree of composition that occurs between the interaction of their constituent layers. According to O'hegarty and Kinnane [3], in general, fully composite panels present complete shear transfer between their resistant layers, so that the concrete layers act together to resist bending stresses. On the other hand, in non-composite panels, the resistant layers act independently to resist the imposed loads.

However, for practical applications, all Sandwich Panels that do not contain a large number of connectors in the form of ribs or discrete portions of concrete, present structural behavior with some level of composite interaction between their resistant layers, thus being considered partial composites. In this way, each panel has a certain degree of composition, which depends on the ability of the connectors, and to a lesser extent, the insulation layer, to carry out the transfer of shear between the resistant layers. ([2], [4], [5], [6]).

The Sandwich Panels can also be designed to act as load bearing structures, having the ability to resist and transmit the loads to which they are subjected to the foundations. Due to the nature of their application in buildings, such as floors, roofs, and masonry, these panels will be susceptible to the action of eccentric horizontal and axial loads, imposing on these composite bending and flexo-compression stresses. For this reason, several researchers have developed test methodologies to predict and evaluate the behavior of Sandwich Panels subjected to bending stresses ([5], [7], [8], [9], [10], [11], [12], [13]).

Due to the absence of regulatory regulations and reliable calculation standards, the Sandwich Panel construction system is often used without a real understanding of its structural behavior. In this way, it is necessary to carry out experimental tests for a better characterization of the performance of these elements, however, the tests developed with panels in real scale are too expensive and complex assembly and operation, being, for this reason, very relevant to carry out numerical studies to justify its feasibility, in addition to promoting parametric analyzes to simulate a greater diversity of situations.

Therefore, justified by the importance of performing numerical analysis, this work consisted of the elaboration of a three-dimensional numerical model of a Sandwich Panel of reinforced concrete with EPS core and metallic connectors, using the ABAQUS software [14] for its design, which is based on the Finite Element Method (FEM). The model considers the non-linear behavior of materials, through the use of the Concrete Damage Plasticity model (CDP) and the Plasticity model for steel, which is calibrated and validated by the experimental results of Gara et al. [8].

2 Numerical model development

The numerical model of the Sandwich Panel developed at ABAQUS has a three-dimensional configuration and takes into account the physical and geometric non-linearities of steel and concrete. Its geometry is based on test specimen 7.1 from the experimental work by Gara et al. [8], in which the panel performed for the bending test contains a total length of 4600 mm, with a width of 1120 mm and a thickness of 120 mm, which is subdivided so that the EPS layer is 80 mm and the upper and bottom concrete layers are 50 mm and 30 mm thick, respectively. In addition, the prototype has a 300 mm wide reinforced concrete beam at each end, in which longitudinal reinforcement of 8 mm in diameter and transverse reinforcement of 6 mm in diameter with a spacing of 200 mm was used, as illustrated in Fig. 2.

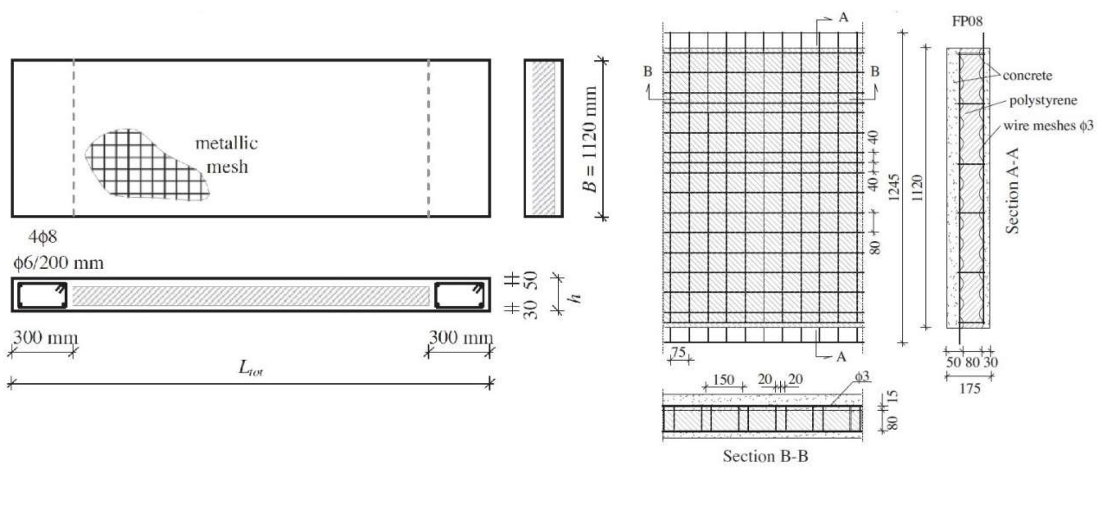

Figure 2. Sandwich Panel Geometry – Adapted from Gara et al. [8]

The panel also contains metallic meshes composed of welded steel wires with a diameter of 3 mm, which are surrounded by layers of concrete and arranged as shown in Fig. 2, in this same image, you can still see the connectors that cross the EPS layer and are welded to the metallic meshes, also consisting of steel wires with a diameter of 3 mm.

In addition to the bending test, Gara et al. [8] performed uniaxial compression tests of concrete, in cylindrical specimens, obtaining the average compression strength (f_{cm}), average tensile strength (f_{tm}), and Young's modulus of the order of 10500 MPa. The authors also performed tests of characterization of the metallic mesh and determined the average tensile strength (f_m) of the steel and the percentage elongation at failure (A_{gt}), according to the values detailed in Tab. 1

Table 1. Properties of the concrete and steel that make up the metallic meshes

Concrete	metallic meshes
$f_{cm} = 25.10\text{MPa}$	$f_m = 769.00\text{MPa}$
$f_{tm} = 2.40\text{MPa}$	$A_{gt} = 7.62\%$

The experimental setup of the bending test performed by Gara et al. [8] is shown in Fig. 3, and consists of a 4-point bending test, in which the supports are located at the centers of the width of the end beams, so that the load is applied to employ a hydraulic actuator, with the aid of a beam of distribution that concentrates the load in two strips along the entire width of the panel, positioned symmetrically concerning its center and separated by a distance$d = 1.0\ m$.

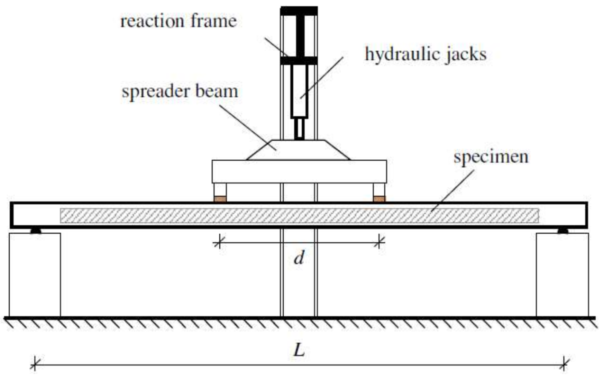

Figure 3. Experimental setup – Adapted from Gara et al. [8]

The Sandwich Panel model was developed at ABAQUS and subdivided into parts, according to the type of material and geometry. Initially, the EPS layer with a wavy cross-section was executed, then the concrete part that comprises the lower and upper layers of the panel and the end beams was modeled, finally, the metallic meshes interconnected by the steel wire connectors were made, and also the rebars and stirrups that make up the reinforcement of the beams. All the parts that make up the model are illustrated in Fig. 4.

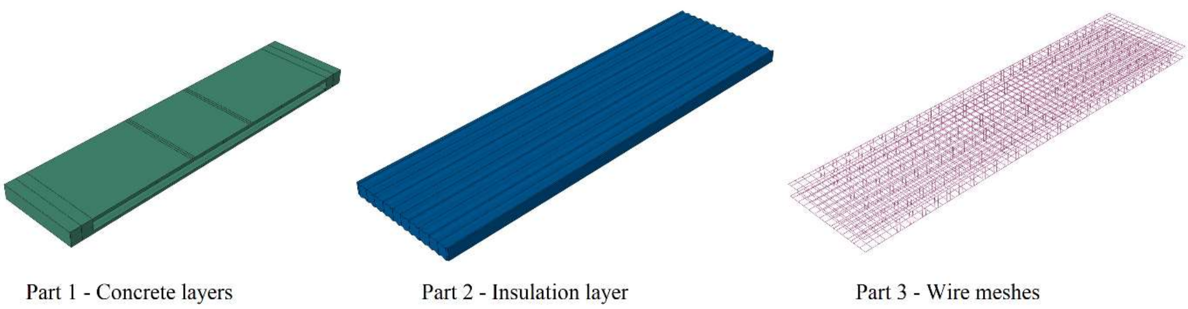

Figure 4. Model parts

The association between the parts of the model was carried out as follows, for the connection between the concrete elements and the EPS layer, the *surface-to-surface* contact interaction was used, whose mechanical constraints are based on the *kinematic contact method*, so that its properties consist of the tangential behavior with the friction *penalty* formulation and friction coefficient 0.1, in addition to the normal *hard* behavior, which prevents the penetration of one surface over the other. For the interaction between the metallic meshes, the reinforcements, the connectors, and the concrete, the *embedded* constraint was used, which promotes the joint movement of the bars and the concrete, disregarding the slip between them.

Fig. 5 shows the finite element mesh defined for the solid parts of concrete and EPS, in which the element C3D8R (linear quadrilateral solid element with 8 nodes and reduced integration) was used. Regarding the metallic parts, the truss element with two nodes and 3 degrees of freedom at each node (T3D2) was used. As for the finite element size, this was determined from a mesh convergence study, through which three different sizes were implemented for the elements that make up the model, namely: 30 mm, 40 mm, and 50 mm. The size of 30 mm was applied to the definitive numerical model, which presented the best results.

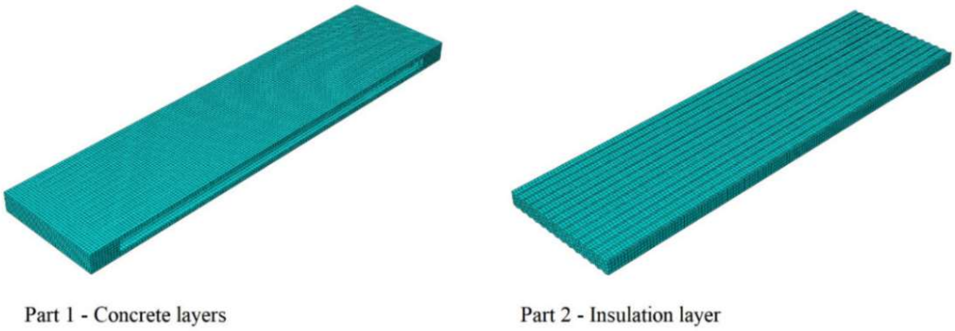

Figure 5. Finite element mesh

To simulate the test, the loads were evenly distributed over the surfaces positioned at the load application points indicated by the experimental setup, as illustrated in Fig. 6. The assignment of the boundary conditions of the numerical model took place through the modeling of supports, which consist of rollers cut in half so that translations and rotations in the X, Y, and Z directions were restricted on the flat surfaces of the supports that receive the support reactions. To perform the analysis of the model, the *Dynamic Explicit solver* was used, which makes use of a dynamic analysis method, but which can be applied to static analyzes of monotonic loads, provided that there is a reduction in the inertia effects arising from a rate of sufficiently small loading application [14].

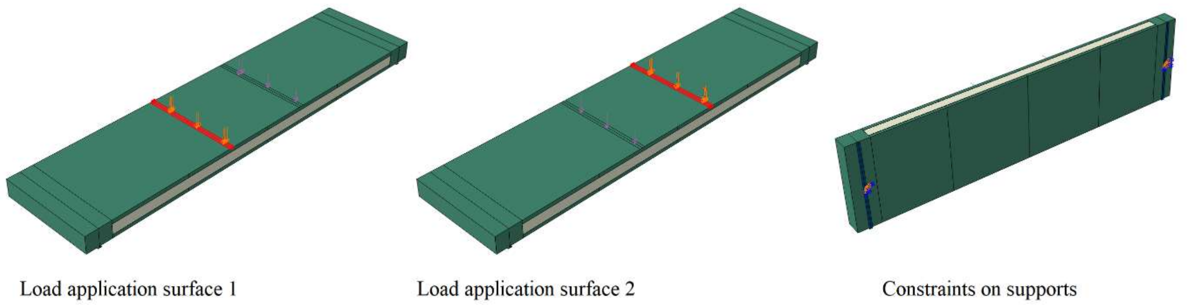

Figure 6. Load application surfaces and support restrictions

In modeling the materials, the constitutive model available in the ABAQUS library was used for the concrete, entitled the Concrete Damage Plasticity model (CDP), this tool takes into account the theory of plasticity and the mechanics of damage, and is suitable for describing the behavior of the concrete, as it is capable of simulating the degradation of the stiffness and the rupture of this material, to be based on the failure mechanisms that include cracking by traction and crushing by compression [15]. In addition, the CDP has a well-established application in the scientific environment, being used by several researchers as a constitutive model of concrete for numerical simulations ([16], [17], [18], [19]).

In this work, the CDP was implemented by adopting the plastic parameters and the calculation methodology for the evolution of the damage variables proposed by Alfarah et al. [20], being assigned as inputs parameters the concrete strengths detailed in Tab. 1, in addition to a Poisson ratio of 0.2 and a mass density of 2400 kg/m³, modulus of Young of 10500 MPa and average concrete compressive strain of 0.0042, obtaining for the ultimate strain of concrete the value of 0.051.

For steel modeling, the constitutive model present in the ABAQUS library under the name PLASTIC was implemented, which consists of an elastoplastic formulation that takes into account the Von Mises flow criterion, with an associative flow rule, being suitable for application in ductile materials, such as steel [15]. As for the uniaxial behavior of steel, the tri-linear relationship proposed by Nguyen & Kim [21] was admitted, in which the yield stress (σ_y) is determined for a strain of 0.2% (ε_y) and the ultimate stress (σ_u) is reached for a strain of 0.6% (ε_u). In addition, the average tensile strength obtained in the tests by Gara et al. [8], for the reinforcement of the beams, the specifications of BS EM 4449:2005 referring to class B500B were used, for the characteristic tensile properties, the other parameters adopted are detailed in Tab. 2.

Table 2. Steel properties used in the model

Component	Density (Kg/m³)	Young's Modulus (MPa)	Poisson's ratio	σ_y(MPa)	σ_u(MPa)
Meshes and Connectors	7850	210000	0.3	600	769
Reinforcement	7850	210000	0.3	500	540

Finally, for the EPS constitutive model, a linear behavior was assumed, adopting as parameters Young's modulus of 2.34 MPa, mass density of 22 Kg/m³, and Poisson's ratio of 0.35. These values were obtained from the work of Hopkins et al. [19] who also used this same approach in their numerical model, achieving good results.

3 Numerical model validation

The numerical model was validated from the experimental results of model 7.1 by Gara et al. [8], for that, it was verified the load versus deflection curve at the mid-span of the panel, ultimate load, and ultimate moment, and failure modes. For comparison purposes, it is presented in Fig. 7 the superposition of the load x deflection curves in the middle of the span, for the experimental and numerical model proposed, and in general, a good agreement between the curves is observed. Regarding the ultimate load and ultimate moment, the values obtained from the numerical model were 14.74 kN and 12.16 kNm, while the values reached in experimental tests were 14.40 kN and 11.88 kNm, thus verifying a difference of 2.36 % between numerical and experimental values for the ultimate load and ultimate moment, evidencing proximity between the results.

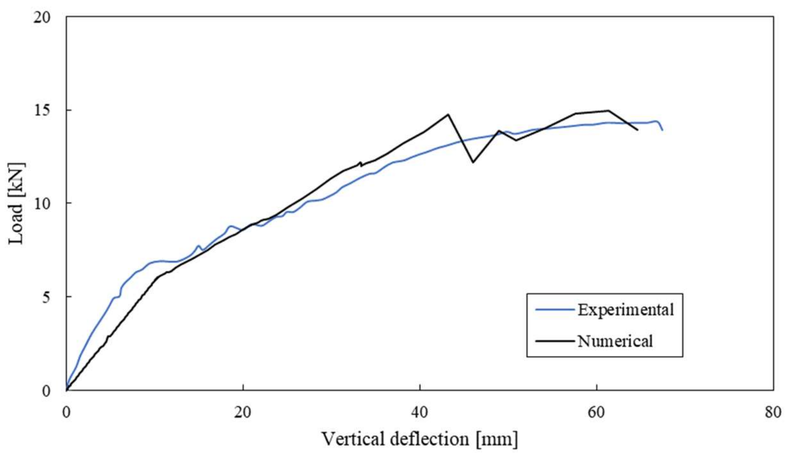

Figure 7. Mid-span load versus deflection curve

When analyzing the curves shown in Fig. 7, a reasonable agreement was identified between the numerical and experimental results, obtaining very close values for the ultimate load, demonstrating that the numerical model is capable of adequately estimating the resistant capacity of Sandwich Panels when subjected to bending stresses, this can also be proved by comparing the failure mode of the numerical model (Fig. 8) with the failure mode of the experimental model (Fig. 9), in which a similar behavior is observed for both with a rupture due to excessive deformation and cracking in the lower concrete layer and crushing of the upper layer in the mid-span section. In this way, it can be affirmed that the numerical model adequately simulated the evolution of the damage in the concrete and the contact interactions between the EPS and the resistant layers, demonstrating that the implementation of the CDP and the *kinematic contact method* in ABAQUS, applies well to the modeling of Sandwich Panels subjected to bending stresses.

However, it can also be seen in Fig. 7, that a considerable difference between the deflection of the numerical model and the experimental one, is due to the greater rigidity of the numerical model because it does not have the same accommodations that occur during an experimental test, an alternative to make the numerical model more flexible is to reduce the application of load application speed, however, this action would bring a high computational cost that is beyond the capacity of the devices available to perform this work.

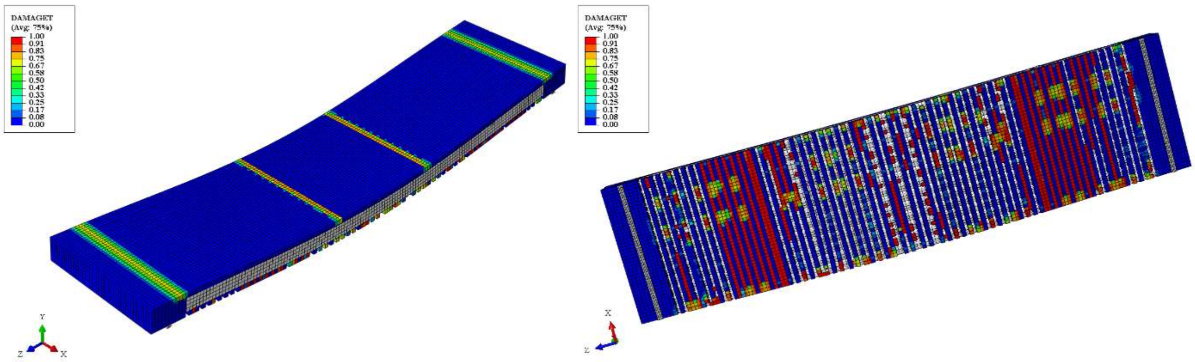

Figure 8. Failure mode of the numerical model

Figure 9. Failure mode of the experimental model – Adapted from Gara et al. [8]

4 Conclusions

In the present work, a non-linear three-dimensional numerical model was developed using the finite element method, capable of efficiently estimating the resistant capacity of reinforced concrete Sandwich Panels with EPS core and metallic connectors, when subjected to bending stresses. Furthermore, the model was validated by experimental tests by Gara et al. [8], and presented results in agreement with the experiments. In this way, the model obtained can be used as a tool to assess the feasibility of investments destined to carry out experimental tests, and also to support the progress of research without the need to perform new experiments. Therefore, the model resulting from this work allows the development of analyzes to better understand the structural behavior of Sandwich Panels, which are presented as a constructive system of increasing application in brazilian civil construction.

Acknowledgments. The authors are grateful for the support provided by the University of Brasília (UnB), which made this work possible.

Authorship statement. The authors hereby confirm that they are the solely liable persons responsible for the authorship of this work and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors or has the permission of the owners to be included here.

References

[1] Bertini, AA Sandwich structures with projected mortar plates. Doctoral Thesis – School of Engineering of São Carlos, University of São Paulo. San Carlos, 209p. 2002.
[2] PCI COMMITTEE REPORT. State of the Art of Precast/Prestressed Concrete Sandwich Wall Panels. PCI Journal, v. 56, no. 2, p. 131–176, 2011.
[3] O'HEGARTY, R., KINNANE, O. Review of precast concrete sandwich panels and their innovations. Construction and Building Materials, v. 233, no. 117145, 2020.
[4] Tomlinson, DG Behavior of Partially Composite Precast Concrete Sandwich Panels Under Flexural and Axial Loads. Doctoral Thesis – Queen's University, Kingston, Ontario, Canada, 353p. 2015.
[5] Bush, TD, Stine, GL Flexural Behavior of Composite Precast Concrete Sandwich Panels with Continuous Truss Connectors. PCI Journal, v. 39, no. 2, p. 112-121, 1994.
[6] Salmon, DC, and Einea, A. (1995). “Partially composite sandwich panel deflections.” Journal of Structural Engineering, 121(4), 778–783.
[7] Kim, J., You, YC Composite Behavior of a Novel Insulated Concrete Sandwich Wall Panel Reinforced with GFRP Shear Grids: Effects of Insulation Types. materials, v. 8, no. 3, p. 899-913, 2015.
[8] Fabrizio Gara, Laura Ragni, Davide Roia, Luigino Dezi, Experimental behavior and numerical analysis of floor sandwich panels, Engineering Structures, Volume 36, 2012, Pages 258-269.
[9] An Chen, Thomas G. Norris, Paul M. Hopkins, Mostafa Yossef, Experimental investigation and finite element analysis of flexural behavior of insulated concrete sandwich panels with FRP plate shear connectors, Engineering Structures, Volume 98, 2015, Pages 95-108.
[10] A. Benayoune, AAA Samad, DN Trikha, AAA Ali, SHM Ellinna, Flexural behavior of pre-cast concrete sandwich composite panel – experimental and theoretical investigations, Construct. Build. mother 22 (2008) 580–592.
[11] S. Al-Rubaye, T. Sorensen, M. Maguire, Investigating composite action at ultimate for commercial sandwich panel, Compos. Connect. (2017).
[12] S. Pessiki, A. Mlynarczyk, Experimental evaluation of the composite behavior of precast concrete sandwich wall panels, PCI J. 48 (2003) 54–71.
[13] Douglas Tomlinson, Amir Fam, Analytical approach to flexural response of partially composite insulated concrete sandwich walls used for cladding, Engineering Structures, Volume 122, 2016, Pages 251-266.
[14] ABAQUS, v 2020. User's Manual. Dessault Systemes Simulia Corp. Providence, RI, USA, 2020.
[15] JM Lima. Study of the load bearing capacity of the truss shear connector via the finite element method. MSC thesis, University of Brasilia, 2018.
[16] Jerfson M. Lima, Luciano M. Bezerra, Jorge Bonilla, Wallison CS Barbosa, Study of the behavior and resistance of right-angle truss shear connector for composite steel-concrete beams, Engineering Structures, Volume 253, 2022, 113778.
[17] BEZERRA, LM; BARBOSA, WCS; BONILLA, J.; CAVALCANTE, ORO Truss-type shear connector for composite steel-concrete beams. Construction and Building Materials, v. 167, p. 757–767, 2018.
[18] BONILLA, J.; BEZERRA, LM; LARRÚA, R.; RECAREY, C.; MIRAMBELL, E. Numerical modeling with experimental validation applied to the study of the behavior of stud-type connectors in concrete and steel composite structures. Revista Ingenieria de Construccion, v. 30, no. 1, p. 53–68, 2015.
[19] Paul M. Hopkins, An Chen, Mostafa Yossef, Static and dynamic analyzes of insulated concrete sandwich panels using a unified non-linear finite element model, Engineering Structures, Volume 132, 2017, Pages 249-259.
[20] ALFARAH, B.; LÓPEZ-ALMANSA, F.; OLLER, S. New methodology for calculating damage variables evolution in Plastic Damage Model for RC structures. Engineering Structures, v. 132, no. January, p. 70–86, 2017.
[21] NGUYEN, HT; KIM, SE Finite element modeling of push-out tests for large stud shear connectors. Journal of Constructional Steel Research, vol. 65, no. 10–11, p. 1909–1920, 2009.