XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ – BRAZIL

Mechanical Behavior comparison of aircraft joints modeling

Marco Túlio dos Santos[1], Rodrigo de Sá Martins[2], Marcelo Greco[1]

[1] *Graduate Program in Structural Engineering, Federal University of Minas Gerais*
Av. Pres. Antônio Carlos, 6627 - Pampulha, Belo Horizonte - MG, 31270-901, Brazil
mtuliods@hotmail.com, mgreco@dees.ufmg.br
[2] *Graduate Program in Mechanical and Materials Engineering, Federal University of Technology - Parana - UTFPR*
Rua Deputado Heitor Alencar Furtado, 5000, Curitiba, PR, 81280-340, Brazil
rodrigo.desa.martins@gmail.com

Abstract. The procedure of structural design for aircraft parts is widely known and discussed in the academy and in the industry, although it has been improved along the time. It is based on a detailed process of aerodynamics loads study coupled or not with specifications required by regulatory agencies. Further, several interactions of analysis are done to define the critical stress state of the structure submitted to load conditions, because it is a complex structure that needs to be often improved and updated (considering the requirement of assembly/disassembly simplicity). There are several components in an aircraft attached to each other by the use of fasteners, rivets or nuts made of different materials (aluminum, steel, Inconel among others). In fact, it is not to easy to obtain stress state of the aeronautical structure for real loading conditions. Further, it is also difficult to calculate the load acting on each one of the joints. Several studies were already performed in order to obtain the correct understanding of how actual loads is distributed through the joints. The present work aims to compare results in aircraft joints (focusing in spar and skin regions), considering three levels of fidelity to understand the differences in the structural response using different type of modeling approach. Finite Element Analysis modelings made using Nastran software were performed. Preliminary results show good response agreement even for high and intermediate detail levels. Low detail level present promisor response as a tool for predesign.

Keywords: Hierarchical models; Aircraft joints, Finite Element Methods, Aircraft Fasteners.

1 Introduction

One of the best known methods for a structural analyst is the Finite Element Method. Through this method, it is possible to divide the structure into a finite number of elements that best represents the structure's geometry and simulate all its loading conditions, considering interpolation functions for response fields (stress, strain, displacement). The choice of method alone is not sufficient for the design of the structure. It is also necessary to define what level of detail a given structure needs in order to obtain satisfactory results.

Several authors have already studied the best way to obtain the loading in a joint, considering it homogeneous or hybrid (when it involves metallic parts joined to composite materials). Rutman et al. [1] , for example, suggest the use of 1d elements with Euler formulation connected by spring elements that are able to check the structure's st iffiness in the axial and shear directions, as it can be seen in the figure 1.

Askri et al. [2] uses a high-fidelity joint model to build a low fidelity one, using spring models between surfaces using a multi-fidelity approach [3]. In another study presented by Martins et al. [4], it is possible to see the implications of to not be careful while modeling joints, since the load distribution can be erroneously determined. At another point, Bedair and Eastaugh [5] studies the importance of taking into account the secondary moment that happens in overlapping joints.

On the opposite hand of the detailed models that require a high computational cost, in addition to a significant time demand for construction, there are models with a low level of detail. The figure below illustrates the level of detail a joint can have, all using the Finite Element Method. In this paper, 3 levels of fidelity are compared in prototype wing. The different fidelity levels are shown in figure 2.

Considering the importance to the safety of joints used in the aeronautical industry, one cannot fail to em-

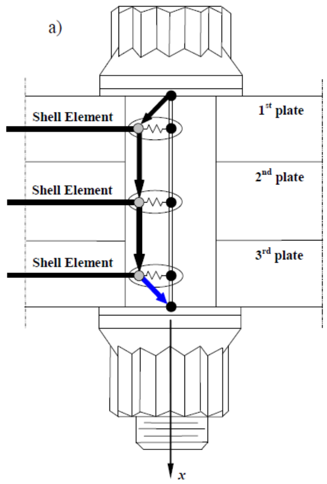

Figure 1. Rutman 1d fastener model. ref. [1]

phasize the fact that the competitiveness of the industry is directly linked to the efficiency of its engineering team. Thus, simulating all loading conditions assertively, without spending a lot of time is vital.

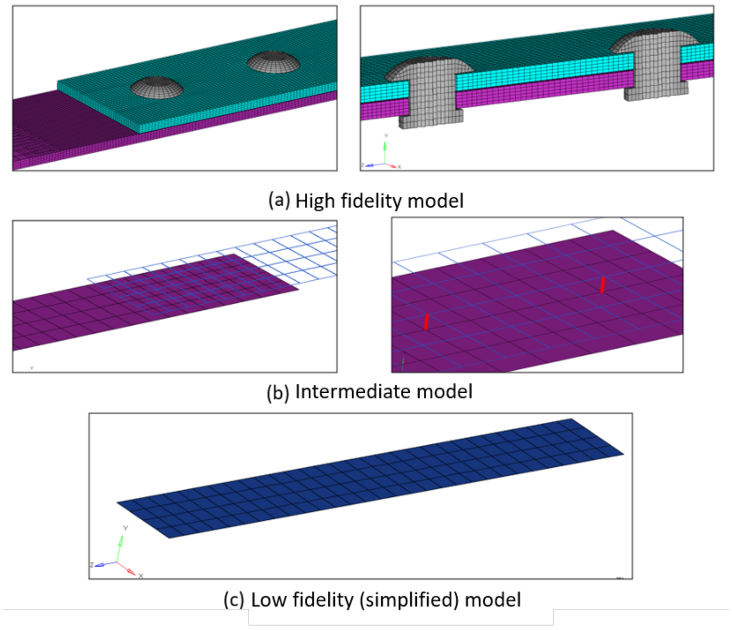

Figure 2. Different levels of finite element fidelity models.

2 Methodology

The prototype model of the work will be the main box of a wing . Three types of riveting configurations will be studied, which are:

- Single Joint
- Double Joint
- Staggered Joint

Those types of joints are shown in figure 3. The correct understanding of the load distribution in joints used in aeronautical components is of paramount importance for their design, as discussed by Chaves and Fernandez [6]. In addition, having means to correctly obtain the margin of safety for each of these joints accurately becomes crucial, since any complication during a field operation, it is necessary to use these safety margin values for evaluation, and correct modification, if necessary.

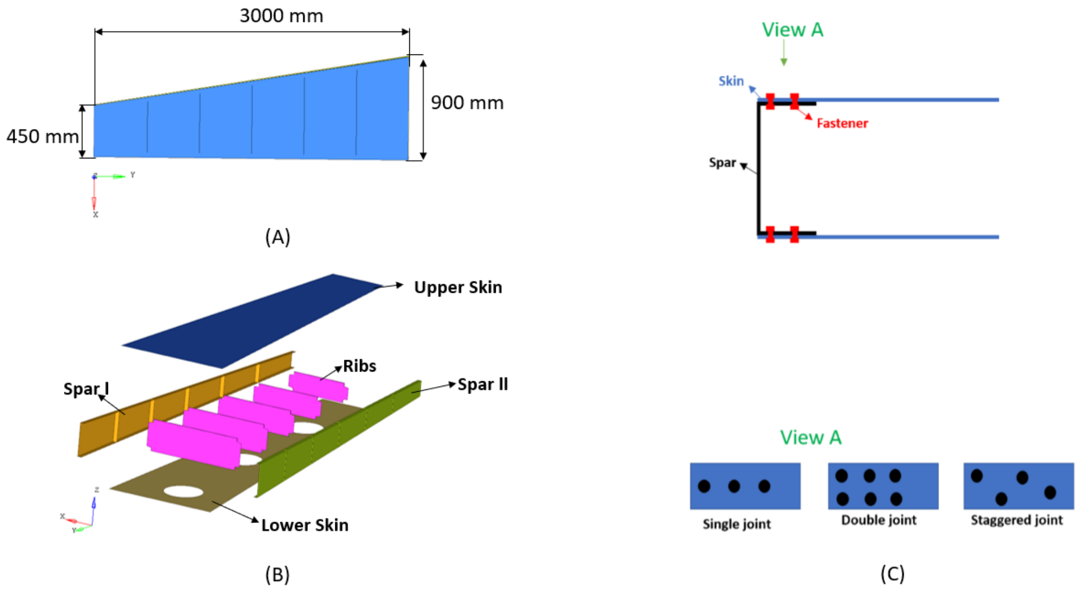

Figure 3. Wing model and Type of Joints.

The mathematical model to be used is only a didactic wing built for study purposes, comprising of upper skin, lower skin, front and rear spars and ribs. Figure 3 (B) shows the mathematical model with its dimensions and the exploded view.

Riveted joints are going to be simulated considering some typical conditions of connections in an airplane using the three simplifications mentioned above and to compare the results, in order to identify the level of reliability of the use of very low fidelity models using analytical methods to obtain the flow of loading. Table 1 presents the material properties (ref. [7] for aluminum and ref. [8] for fasteners), thickness for the sheets and diameter of fasteners.

Table 1. Mesh sensibility analysis

Model properties [mm]		
Component	Thickness /Diameter [mm]	Material
Skin	3.00	AL 7475-T761
Spar	4.50	AL 7475-T761
Ribs	3.75	AL 7475-T761
Fasteners	3.97	Titanium

Figure 4 shows these models where (A) refers to model with low fidelity, (B) model with mid fidelity and (C) the model with high fidelity.

For the model with low fidelity approach, isoparametric (CQUAD4 OR CTRIA 3) elements with 4 integration points are going to be used to simulate the wing skin, ribs and spar web while bar elements considering Euler formulation (ref. [9]) are going to be used to simulate spar cap. For the mid fidelity the whole model are built with those same isoparametric elements but the connections, in which it used the combination with bar and spring elements according to described in ref. [1] and showed in figure 1.

The High fidelity model is built with solid element CPENTA (with 5 grid points) and CHEXA (with 8 grid points) all of them considering reduced integration points, which means 2x2x2 points.

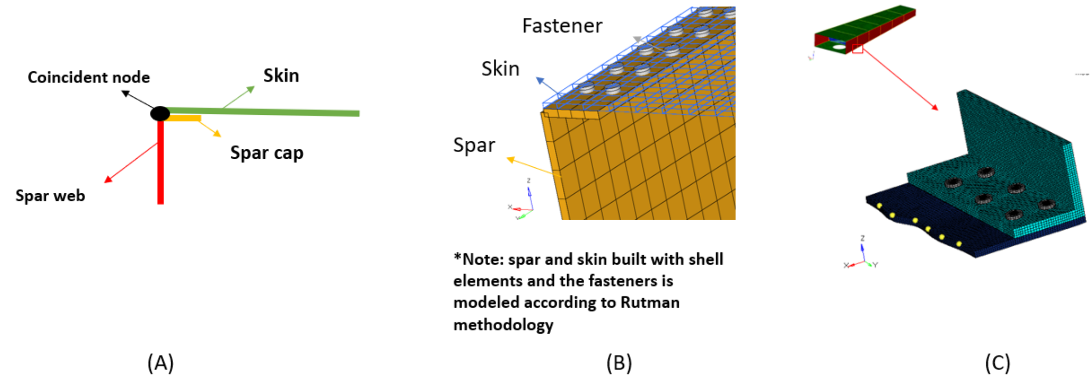

Figure 4. Mathematical Models.

2.1 Initial and boundary Conditions

To build the finite element model, it was used the Stender Method [10] to simplify a load distribution along the wingspan and the forces and moment where applied through a rigid element connected in the wing. The load distribution was made such as the total load equals 2000 daN. The aerodynamic moment (considered being at 25% of the chord) was calculated using [11]:

$$M_A = \frac{1}{2}\rho V^2 S_w \bar{\bar{c}} C_m \tag{1}$$

where:
M_A is the aerodynamic moment,
ρ is the air density,
V is the reference speed,
S_w is the plain view wing area,
$\bar{\bar{c}}$ is the mean aerodynamic chord and
C_m is the moment coefficient.

Using eq. and with the value of -0.2 to C_m (NACA 006 [11]) it was found a value of 223.747 daN.mm. The distribution of load (Stender) was used to the aerodynamic moment.

A total of 7 models were built, three of them regarding to mid fidelity model, three for high fidelity model and one for low fidelity model and the level of fidelity follows from figure 2. Both the low and mid-fidelity models are linear elastic. Thus, it is possible to represent the whole wing with these approaches. On the other hand, the high fidelity model involves the calculation of the non-linear bearing stresses and can't be solved for the whole wing at once.

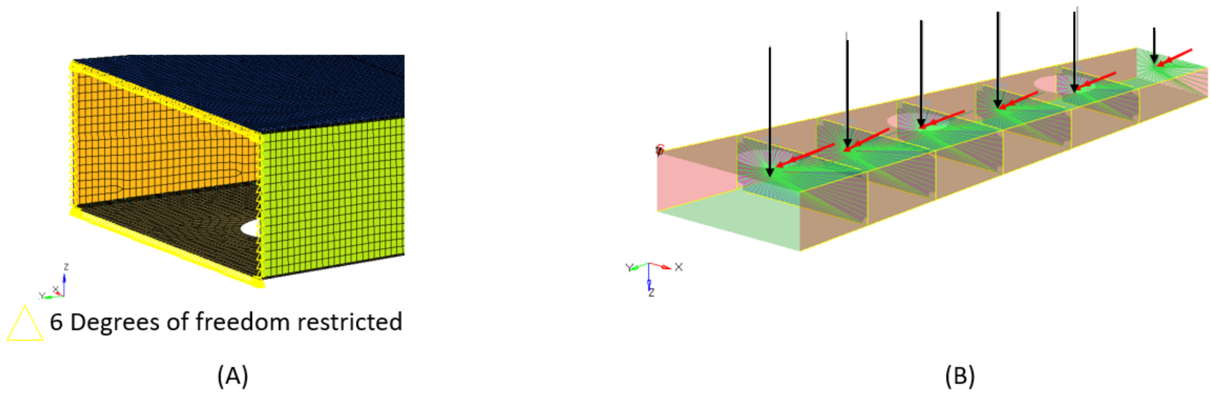

Figure 5. Boundary Conditions.

2.2 Fasteners loads for low fidelity model

Regarding low fidelity model, only one model are going to be used in order to obtain the fasteners forces for each one of the three configuration. The membrane shear force provided for the QUAD4 and TRIA3 element is used to obtain those forces.

For each one of the attachments it was took account its coordinate system and it was verified if those fastener is inside of the element and the load of that element was assigned for the fastener, if there are more than one fastener inside the element the load is divided by two and so on. In order to guarantee the node are inside the element a vector product using the node analyzed and the nodes of the element were performed as it can be seen on 6 (B).

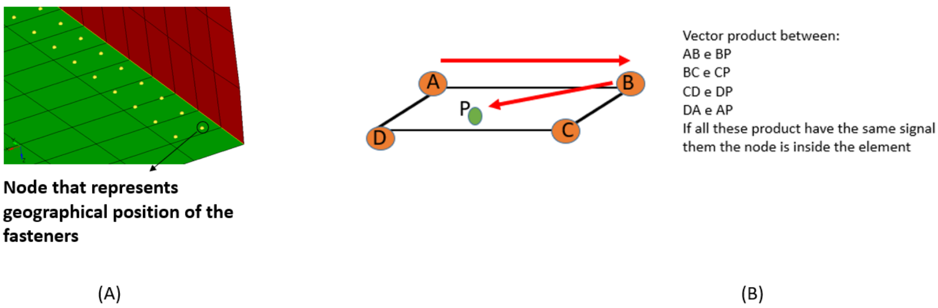

Figure 6. Low fidelity model analysis.

3 Results

3.1 Mesh sensibility analysis

In order to check the mesh sensibility, two kinds of refinement were considered as it can be seen in table 2, where the refinement 1 refers to the model used in this study and refinement 2 is a model with the elements with half of size of the first one. As it can be seen, there is no significant modification in the results if the size of elements used is less and this behavior can be expected no matter the refinement used.

For the low fidelity model the average size of elements regarding to refinement 1 is 30mm and for mid fidelity model the average size of the elements is 15mm. All the models were built following the best practice discussed in ref. [12].

Table 2. Mesh sensibility analysis

Total Displacement [mm]		
Model	Refinement 1	Refinement 2
Low Fidelity	45.20	45.60
Mid Fidelity	40.30	40.96

3.2 Low fidelity vs mid-fidelity comparison

Figure 7 shows the result comparison along the span of the low and mid fidelity models. It can be seen that, although the results are not equal, they follow a same pattern of each of the joints configurations. The results of the low fidelity tends to be conservative on all configurations.

The last figure shows a display of normalized margin of safety for the single joint model. Suppose that the maximum load found in all fasteners for the low and mid fidelity models and take it as the joint allowable.

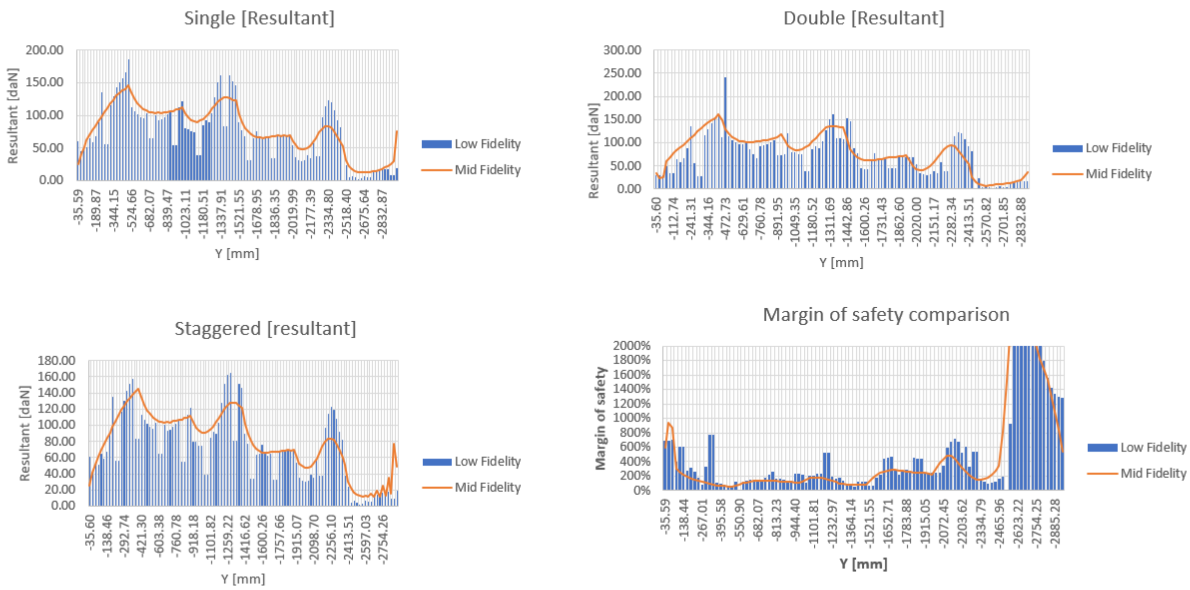

Figure 7. Loads comparison - Through the span wise.

3.3 High fidelity model comparison

The next step is to compare the load distribution prediction of each fidelity model approach. To model the high fidelity model, just a portion of the wing was modelled (from $y = -480mm$ to $y = -410mm$ and from $x = 70mm$ to $x = 100mm$). The model is attached to the low fidelity model using a sub-modelling approach [13].

From figure 8, it can be shown that for single, double or staggered joints the low-fidelity is the most conservative in a overall point of views. It means that, if the joint is design to withstand the critical load, it is most likely to be safe. Although the low fidelity model presents a conservative value, the load distribution between the fasteners is not always similar considering the fidelity, it occurs only to single joint. For double and staggered the load distribution do not follow a pattern as it is observed in single joint.

The high fidelity model was built using enforced displacements. Since low fidelity models tends to be more rigid than high fidelity ones [4], using the conventional sub-modelling approach can lead to an underestimation of the load distribution. A way around of this effect could be the usage of nodal forces instead of nodal displacements.

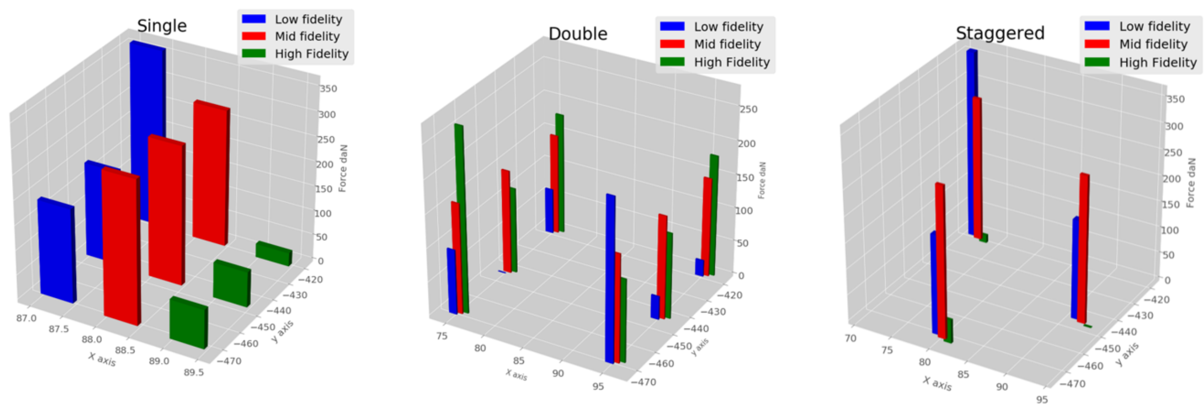

Figure 8. Loads comparison - High fidelity.

4 Conclusions

According to presented in figure 7 the load distribution through the spanwise is very similar even considering different types of attachment configuration as well as fidelity model, being evident that the low fidelity model can represent the structure with confidence and provide a conservative result. The margin of safety comparison has also shown that it is possible to dimensioning a wing attachment region considering a low fidelity model and uses those data to optimize the region far from the root that it is less loaded.

The high-fidelity model (figure 8) comes to corroborate the insight that the low fidelity model is stiffer, which gives fasteners more loaded and consequently more conservatives results during the dimensioning process.

Another interesting issue to be observed is that in the high fidelity model for single and staggered joints there were a significant decrease in the load distribution regarding the other models, those fact can be explained because of these configuration gives some degrees of freedom for the joint allowing some kind of rotation which let the load stored by means of strain energy, besides the contact area of these configuration is bigger which let the load flow through contact force.

It is also highly important to understand the usability of low-fidelity model, which provide good results without spent to much time modeling while considering a static analysis but provide a poor data with very low confident to perform a fatigue or crack propagation analysis in which the accuracy of the analysis is proportionally linked to the level of fidelity of the model.

Acknowledgements. The authors are grateful for the financial support granted by Fundação de Amparo à Pesquisa do Estado de Minas Gerais (FAPEMIG), Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPq) and Programa de Pós-Graduação em Engenharia de Estruturas (PROPEEs-UFMG).

Authorship statement. The authors hereby confirm that they are the sole 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] Rutman, A., Viisoreanu, A., & Parady, J. A., 2000. Fasteners modeling for MSC.Nastran finite element analysis. *SAE Technical Papers*.

[2] Askri, R., Bois, C., Wargnier, H., & Lecomte, J., 2016. A reduced fastener model using Multi-Connected Rigid Surfaces for the prediction of both local stress field and load distribution between fasteners. *Finite Elements in Analysis and Design*, vol. 110, pp. 32–42.

[3] Bathe, K. J., 2016. *The Mechanics of Solids and Structures - Hierarchical Modeling and the Finite Element Solution*.

[4] Martins, R., Dos Santos, M. T., & Palma, E. S., 2018. Fastening analysis using low fidelity finite element models. *31st Congress of the International Council of the Aeronautical Sciences, ICAS 2018*, , n. September, pp. 0–10.

[5] Bedair, O. K. & Eastaugh, G. F., 2007. A numerical model for analysis of riveted splice joints accounting for secondary bending and plates/rivet interaction. *Thin-Walled Structures*, vol. 45, n. 3, pp. 251–258.

[6] Chaves, C. E. & Fernandez, F. F., 2016. A review on aircraft joints design. *Aircraft Engineering and Aerospace Technology*, vol. 88, n. 3, pp. 411–419.

[7] Niu, M. C. Y., 2011. *Airframe Stress Analysis and Sizing (Third edition)*.

[8] https://jet tek.com/, 2018. Jet-tek.

[9] Falsone, G. & Settineri, D., 2011. An Euler-Bernoulli-like finite element method for Timoshenko beams. *Mechanics Research Communications*, vol. 38, n. 1, pp. 12–16.

[10] Iscold, P. H., 2002. *Introdução às cargas nas aeronaves - UFMG*.

[11] Abbott, I. & Von Doenhoff, A., 1959. *Theory of Wing Sections, Including a Summary of Airfoil Data*. Dover Books on Aeronautical Engineering Series. Dover Publications.

[12] ASME, 2016. *Guide for Verification and Validation in Computational Solid Mechanics*.

[13] Zarzalejos, J. M., Fernández, E., Caixas, J., Bayón, A., Polo, J., Guirao, J., García Cid, J., & Rodríguez, E., 2014. Bolted Ribs Analysis for the ITER Vacuum Vessel using Finite Element Submodelling Techniques. *Fusion Engineering and Design*, vol. 89, n. 7-8, pp. 1790–1794.