STRUCTURAL ANALYSIS OF A RIVETED JOINT AIRCRAFT USING THE FINITE ELEMENT METHOD

Alexandra Alvim Arbex, alexandra.arbex@gmail.com
Fernando de Azevedo Silva, fazevedo@feg.unesp.br
Universidade Estadual Paulista, Faculdade de Engenharia de Guaratinguetá, Departamento de Mecânica, Av. Dr. Ariberto Pereira da Cunha, 333, Pedregulho, 12516-410, Guaratinguetá, SP, Brasil.

Abstract. *Aerospace structure surface are mostly united with rivet joints. Joints are the most common sources of failure in aeronautical structures, so they should be considered one of the most important components of a project, requiring great care, as structural strength and fatigue life, during the use of the aircraft. The finite element method is being increasingly used for calculation and dimensioning of parts in mechanical design, ensuring a reliable result for complex problem, reducing the number of prototypes and design time. The objective in this article is to reproduce, using the finite element software Abaqus 6.10, in a satisfactory manner, the behavior of a riveted lap joint with fifteen rivets distributed in three equally spaced rows when subject to monotonic traction stress. With this study is possible to determine which variables should be assigned to the finite elements model for better represent the riveted joint, so use a method where you can get a better cost / benefit analysis of this nature.*

Keywords*: Riveted joints, finite elements, structural analysis*

1. INTRODUCTION

Riveted joints are widely used on Aeronautical Industry, therefore the mechanical resistance of the components are often studied, according to recent work (Spinelli, 2004). Usually, the riveted joints of aeronautical structures are made of Aluminum Alloys. The finite element method has been used more frequently for this kind of calculation, predicting failures and the structure behavior with high reliability. Many parameters must be considered for the correct modeling of the structure, such as material's mechanical properties, contact conditions, boundary conditions, residual tensions caused by the riveting process, and others.

This paper has as purpose, through the correlation with an experimental test of monotonic traction, to make the simulation, using the software Abaqus version 6.10, and verify the main variables to be considered in a way to obtain an optimized result.

2. EXPERIMENTAL TEST

A single specimen of the riveted joint was subjected to a monotonic traction test, as shown at work (Spinelli, 2004) and has the following dimensions, represented in Fig. 1:

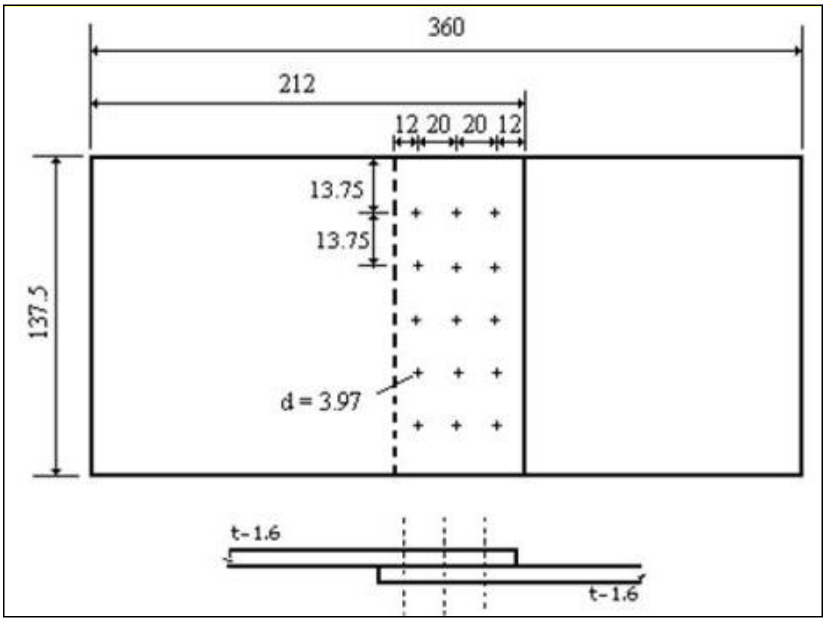

Figure 1. Speciment's dimensions [mm]

The material used for the sheets was Aluminum Alloy 2024-T3 and for the rivets was the Aluminum Alloy 2117-T4. The mechanical properties of both materials are at Tab. 1:

Table 1. Mechanical properties

Mechanical Properties	2024-T3	2117-T4
Modulus of Elasticity [GPa]	73,1	71
Poissons ratio	0,33	0,33
Tensile Stength, Ultimate [MPa]	483	296
Tensile Stength, Yield [MPa]	345	165
Elongation at Break	18%	27%

The monotonic traction test was made at “Laboratório de Ensaios Mecânicos”, as shown in (Spinelli, 2004) using a servo-hydraulic universal machine MTS 25’’, as shown at Fig. 2:

Figure 2. Universal machine MTS 25’’

The sample of the lap joint was fixed at the ends with claws with five security screws each, that are necessary in case the friction is not enough for the sample to be fixed on the machine. The sample is aligned to minimize the second boundary effect. The monotonic traction test was done as follow: a single specimen of the lap joint, was submitted to traction's loadings, applied slowly in a range of 0 to 15600 N, and then was unloaded, always oriented vertically, arbitrated as coordinated “X” axis . In order to study the strain distribution along the lap joint, 13 uniaxial’s strain-gauges were used according to Fig. 3:

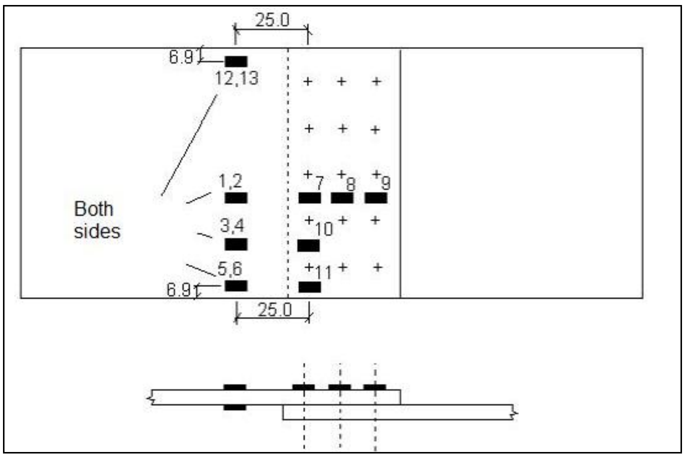

Figure 3. Strain-gauges distribution

3. SIMULATION

Using software Abaqus 6.10, a simulation of the traction test was made, in order to define which variables must be assigned to the finite element model, to better represent the riveted joint. The analysis can be described checking the modeling techniques, boundary conditions and loading in each Step:

3.1. Modeling

Due to the symmetry of the model between the rivets rows, a “strip” was modeled with 1/5 of the joint’s width, as shown in Fig. 4. The benefit of using 1/5 of the full model’s width is to make the modeling process simpler, reducing the analysis time.

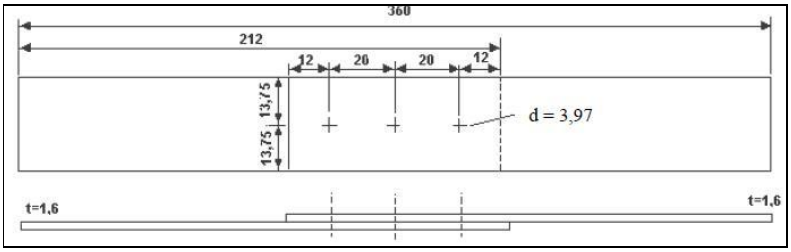

Figure 4. Dimensions of the solid riveted joint’s model [mm]

Thus, using the drawing tool from Abaqus/CAE, both sheets and rivets were modeled as solid parts. For the simulation of the rivets, only the body was considered, because this region that transmits the traction’s efforts of one sheet to another, during the monotonic traction test.

The joint’s assembly is made according to Fig. 5:

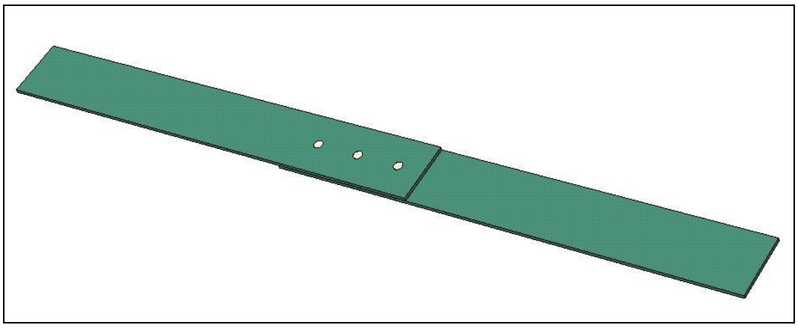

Figure 5. Assembly

The meshing was made with C3D8R elements (8-node linear brick, reduced integration, hourglass control), through the sweep method, with mapped meshing on the most critical areas. The size of the elements changes by region, according to the degree of importance for the results. On the endings was considered an element size of 3 mm, with a control of the elements number at the thickness, which must be at least 4 to get a better representation of bending, as shown at Fig. 6. At the rivets region was used a mesh control to get a compatible mesh between the part instances (rivets and sheets), according to Fig. 7, and there’s a higher mesh refinement in this area, as can be seen at Fig. 8:

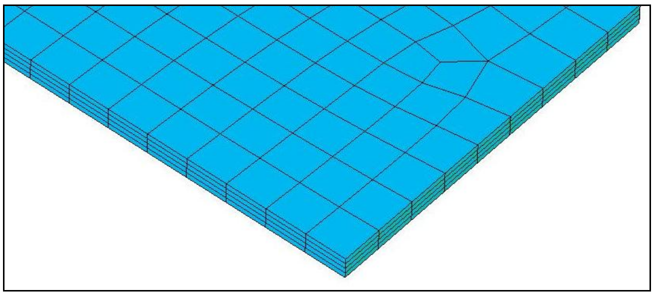

Figure 6. Elements at thickness

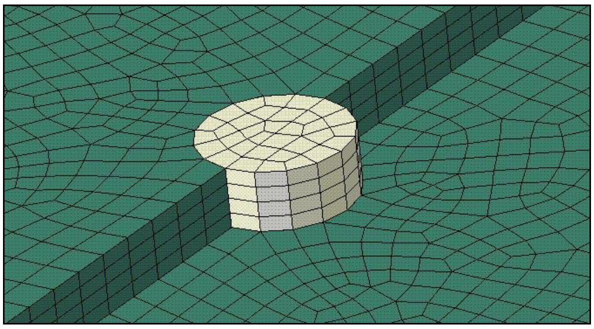

Figure 7. Compatible mesh between the part instances

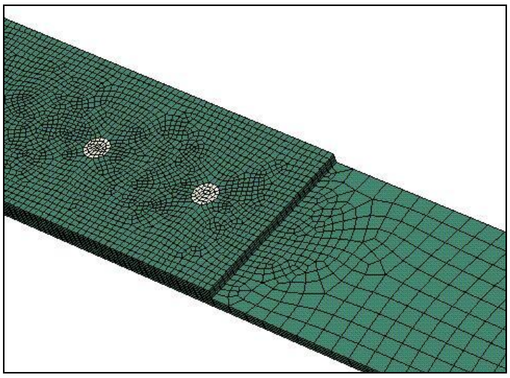

Figure 8. Mesh refinement

3.2. First step

At the first Step an interference fit analysis was made, between the rivets and the plate's holes, in a way to simulate the riveting process. The importance of this Step is the residual stress caused by the riveting process which, on the next Steps, will modify the mechanical behavior at the rivets area, in both static and dynamic situations. In order to obtain a valid correlation from the riveting process, published work (Fung and Smart, 1997a) studied that, to simulate the assembly after the process, the average axial stress at the rivet's shank – which will guide the squeeze force between the sheets – must be about 0,1 times the yield stress of the rivet's material (Al 2117-T4), and the stress that the rivet's shank causes on the face holes at the sheets must be 0,03 times the yield stress of the sheet's material (Al 2024-T3). On this paper, this condition was achieved through interference between the rivet and the hole, and empirically, the references values cited above were found. To obtain these results an interference of 0,002 mm was considered.

The boundary conditions are the encastres at the two endings of the riveted joint, and the restriction on the face of the rivets, which do not allow both rotation and translation on the Z axis. This restriction is necessary to interference fit's analysis, so the nodes on the rivet's shank can meet the nodes of the plate's holes, in a planar movement, so the software is able to calculate the stress. The compatible mesh between the part's instances is indispensable to obtain right results and, in some cases, for the convergence of the analysis. The contact conditions are defined by the type surface-to-surface between the hole's face and the rivet's face. The property of this contact is frictionless, due to the type of analysis as specified at Abaqus Documentations (Abaqus Analysis User's Manual, 2010).

To create an analysis only to calculate the interference between parts, no loadings are included, and in the Contact Module the option "Interference fit" is activated. Inside the "Interference fit", the condition for the calculation is to gradually remove slaves nodes overclosure during the step, which are the nodes from the rivets. The overclosure's adjustment is defined to "automatic shrink fit", that is the default when the analysis must be uniform.

So, during this first Step only the adjust of the rivet's nodes to the hole diameter's nodes will occur, and the result will be the residual stress.

It is possible to verify this condition at Fig. 9, Fig. 10 and Fig. 11:

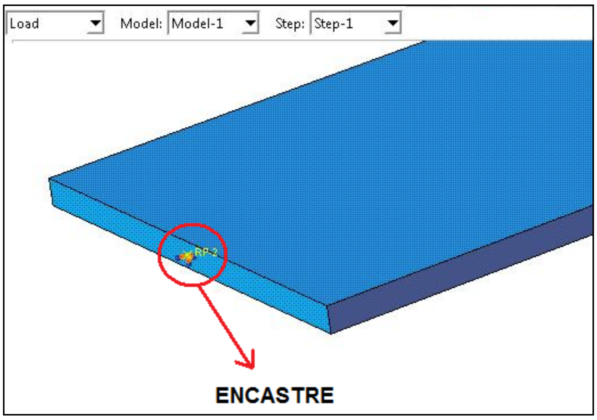

Figure 9. BC Step1: Ending encastre

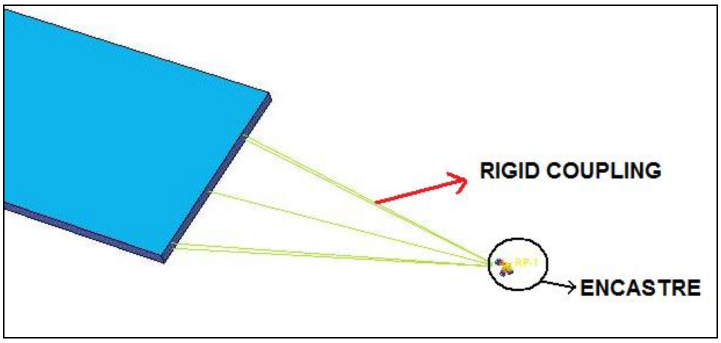

Figura 10. BC Step1: Ending encastre

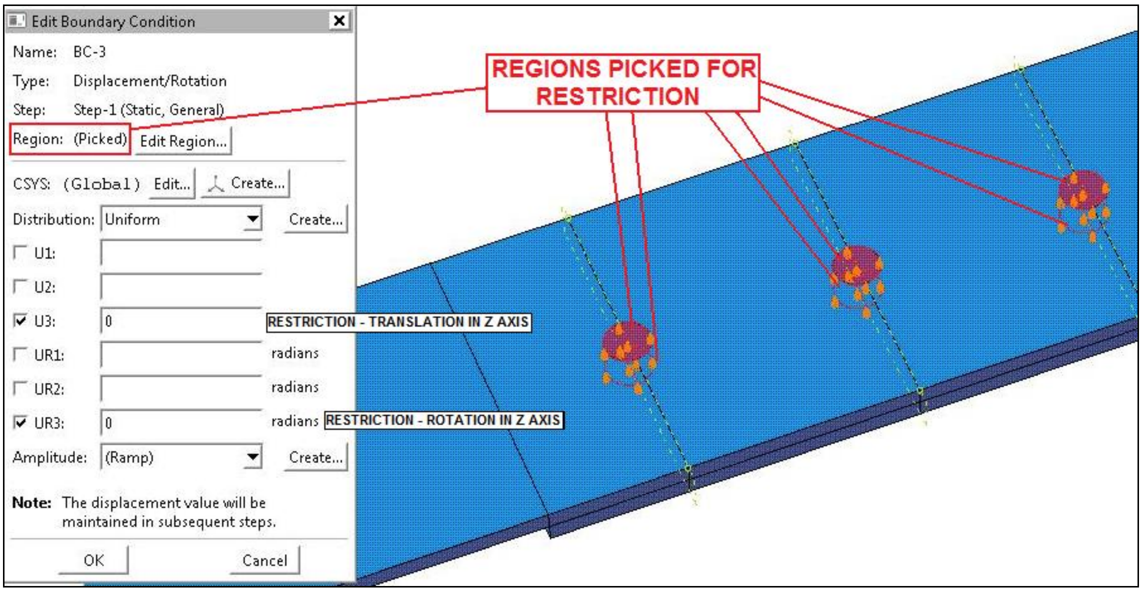

Figura 11. BC Step1: Restrictions

3.3 Second Step

On the second Step, after the interference fit analysis, the boundary conditions are modified. Now there's an encastre applied in one end, and in the other one a displacement of 1mm is applied, through a rigid coupling, on the X axis direction, as shown in Fig. 12 and Fig. 13, respectively. The displacement's application in order of a loading application is made due to the better convergence of the analysis. An output of the reaction force is requested to observe the values concerning to the strain measured by the stain-gauges when the force achieves 3120 N (1/5 of the total force), as in the experimental test.

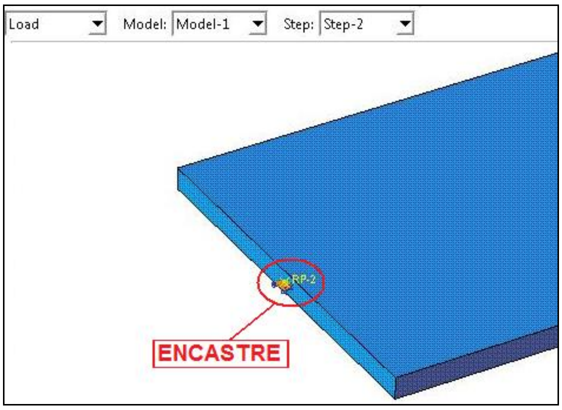

Figure 12. BC Step2: Ending encastre

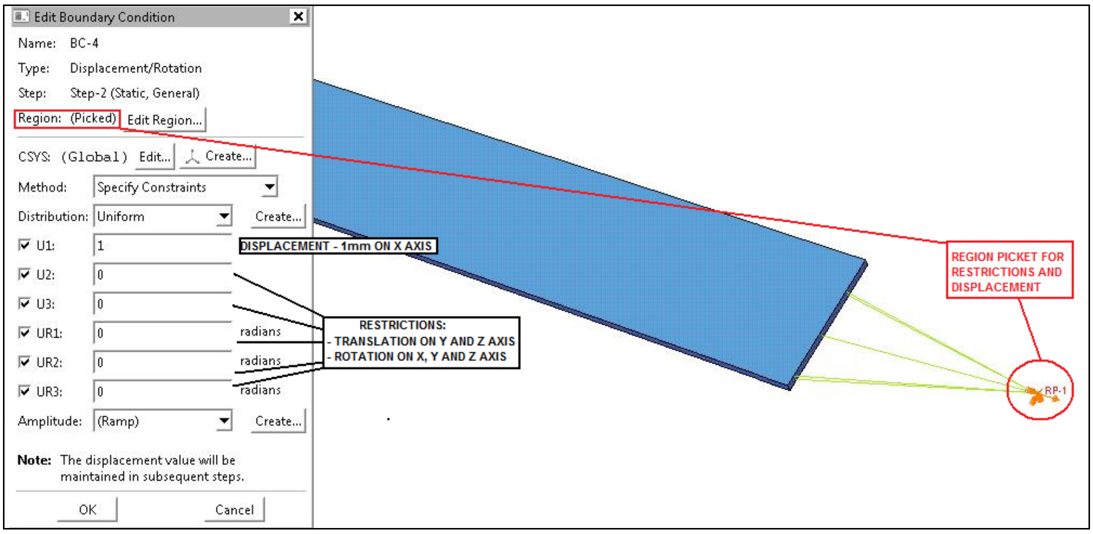

Figura 13. BC Step2: Ending displacement

As the boundary conditions, the contact conditions were modified too. In this Step is used a coefficient of friction of 0,2, according to the study about rivet joints (Müller, 1995). The interference fit is deactivated, and because of the sequence of steps, the residual stresses caused by the first step are automatically considered.

4. RESULTS

In the first Step results, it's possible to observe on Fig. 14, the residual stress caused by the simulation of the riveting process.

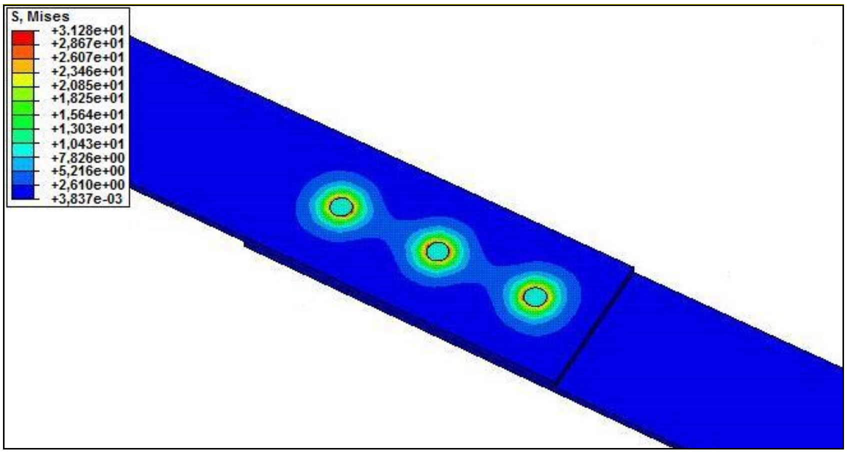

Figure 14. Residual Stress

The founded values for the axial stress at the rivet's shank and the pressure on the face between the rivet's shank and the face of the sheet's hole correspond to the expected values, as defined before. At Fig. 15, is possible to observe that the stress on the rivet's shank is 16,4 MPa. The Eq. (1) shows this relation:

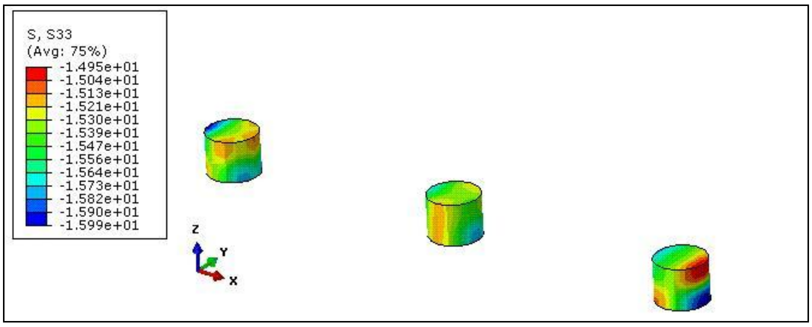

Figure 15. Axial Stress

$$\sigma_{rivetshank} = 0{,}1 \cdot \sigma_{yield-2117T4} = 0{,}1 \cdot 165 = 16{,}5MPa \cong 16MPa \quad (1)$$

At Fig. 16, the result of the pressure at the interface of the plate's holes with the rivets is obtained, and the relation between this value and the yield stress of the material is 0,04. The Eq. (2) shown the relation defined for this tension. It's possible to note by the Eq. (3), that the results are very close to the expected.

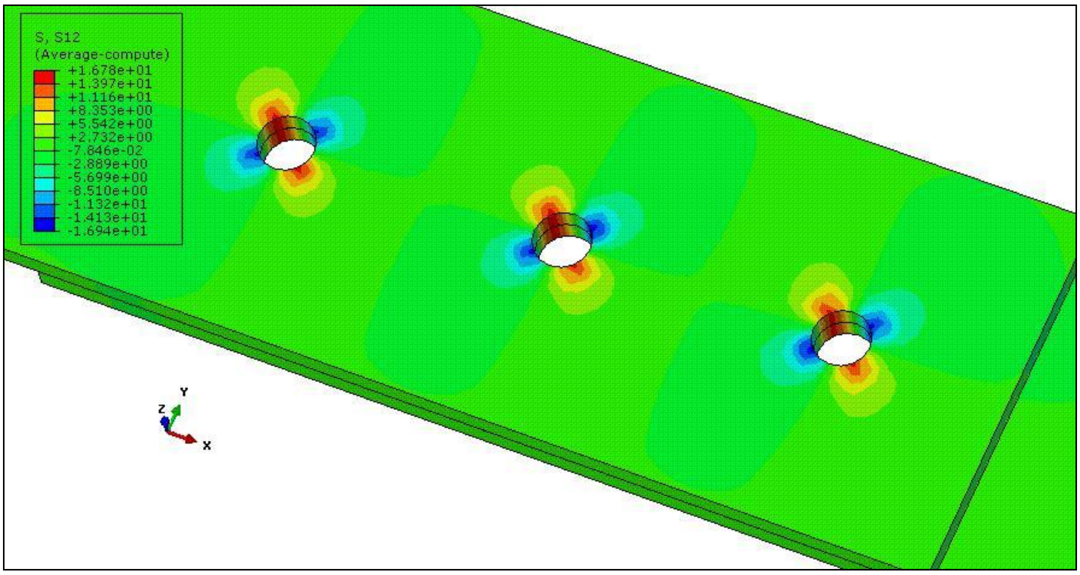

Figure 16. Stress at the interface

$$0{,}03 \cong \frac{\sigma_{platehole}}{\sigma_{yield-2024T3}} \quad (2)$$

$$0{,}04 \cong \frac{16}{345} \quad (3)$$

On the results from the second Step, the strain values obtained by the strain-gauges at the experimental test are compared to the values obtained by the simulation. These values at the simulation were found from the results of the "elastic strain components" for X direction, defined by "EE1". The results were found on the elements that corresponded to each strain-gauge's location. Thus it is possible to validate the finite element model. Table 2 shows the comparison between the experimental and simulation results for the main strain-gauges, measured at the force tested. The most significant error, at strain-gauge 2 is due to edge effect.

Table 2. Strain measures

	EXPERIMENTAL	FEM	ERROR
Strain-Gauge 2	1,25E-03	9,91E-04	20,68%
Strain-Gauge 7	6,32E-04	6,51E-04	3,10%
Strain-Gauge 8	5,53E-04	5,93E-04	7,36%

5. CONCLUSIONS

Through the results that were obtained, was possible to observe the main parameters to be used in this kind of analysis. The modeling using solid elements, although requiring a longer analysis total time, gives more accurate results, for being more detailed and for giving a better representation of the real model. The first Step must be done, by interference fit as done in this paper, or other techniques as thermal analysis, or the complete simulation of the riveting process, (which requires a higher processor and a much longer time of analysis), to consider the influences of the process on resulting residual stress next to the region of the rivets, which will be subjected to subsequent loadings.

The coefficient of friction of 0,2, found in the literature for Aluminium, must be used in all the parts that are in contact in order to represent the effects of interaction between them, that affects the results.

With the experimental test, and the measure of the strain-gauges, was possible to obtain the correlation of the finite elements model with the real model. With this right correlation, it's possible to validate these main parameters used in this analysis.

6. REFERENCES

Abaqus Documentantion - Abaqus Analysis User's Manual, SIMULIA Dassault Système, 2010.

C-P Fung and J Smart : Riveted single lap joints. Part 1: a numerical parametric study, Proc. Instn Mech. Engrs, Vol 211 Part G (1997)

Müller, R.P.G. An Experimental and Analytical Investigation of the Fatigue Behavior of Fuselage Riveted Lap Joints. The Significance of the Rivet Squeeze Force, and a Comparison of 2024-T3 and Glare 3, Delft University of Technology, 1995.

Spinelli, H.A., Análise Estrurual de Juntas Rebitadas de Uso Aeronáutico, UNESP, Guaratinguetá, 2004

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.