COBEM UBERLÂNDIA 2019

ABCM Associação Brasileira de Engenharia e Ciências Mecânicas

25th ABCM International Congress of Mechanical Engineering
October 20-25, 2019, Uberlândia, MG, Brazil

NUMERICAL MODAL ANALYSIS OF AN AIRCRAFT WING PROTOTYPE FOR SAE AERODESIGN COMPETITION

Victor Emmanuel Liete Gasparetto
Marcela Rodrigues Machado
Department of Mechanical Engineering, University of Brasília, Campus Universitário Darcy Ribeiro, Brasília-DF, Brazil
vicelg96@gmail.com, marcelam@unb.br

Sergio Henrique da Silva Carneiro
Department of Aerospace Engineering, University of Brasília, Faculdade Gama UnB, DF, Brazil
shscarneiro@unb.br

Abstract. *The SAE Aerodesign (Society of Automotive Engineers) competition encourages undergraduate and graduate students from Brazil and Latin America to develop a small-scale cargo transport aircraft, from conception, detailed design, construction and testing, with the aim of completing a pre-established flight mission. It is a project carried to the limit of the structural efficiency, looking for an internal structure of low empty-weight, optimizing it for the required flight conditions. In order to understand the response of the aircraft to dynamic demands and ensure project safety, it is necessary to be attentive to the modal parameters of the structure, such as natural frequencies, damping factors and vibrational modes. This work presents the modal analysis of an aircraft wing prototype designed by the Aerodesign team of the University of Brasília. It is studied the finite element modeling of simplified structures that compose the aircraft, such as the main wing spar, proposing a method for determining equivalent flexural stiffness of beams using the Euler-Bernoulli formulation; the complete structure of the prototype aircraft wing is then modeled using ANSYS APDL software. Numerical results for the wing structure modal analysis returned an average percentual difference of 5,8% to the experimental natural frequencies.*

Keywords: *modal analysis, finite elements modeling, Aerodesign, Euler-Bernoulli beam theory*

1. INTRODUCTION

The SAE Aerodesign (Society of Automotive Engineers) competition encourages undergraduate and graduate students to develop a small-scale radio-controlled cargo transport aircraft with the goal of completing a pre-established flight mission of the competition, minimizing empty weight and maximizing the payload, whose value can reach up to 7 times the empty weight. It is, therefore, a project that is pushed to the limit of structural efficiency.

The characteristics of SAE Aerodesign of low-mass aircraft and internal structure optimized for requests under specific flight conditions (flexible structure), as well as the frequent use of materials with high specific stiffness, such as balsa wood, high performance structural foam, fiber fabric of carbon, aramid or glass for the design of each prototype; make the aircraft more susceptible to aeroelastic phenomena or unwanted vibrations that may alter its stability, limiting the operating envelope (Bisplinghoff, 1996).

For the correct understanding of the response to dynamic inputs, in addition to seeking to ratify safety to the project, it is necessary to be aware of the modal parameters of the structure, such as natural frequencies, damping factors and vibrational modes, obtained using structural dynamic analysis methodologies; being then an important stage for an aircraft design, either at the Aerodesign or industrial level. Therefore, modal analysis is an essential tool in the procedure for the determination of those parameters, based on the fact that the vibrational response of a time-invariant linear dynamic system can be expressed as the linear combination of a set of simple harmonic motions called modes of vibration, which are determined from the physical properties inherent to a specific system and its spatial distributions in geometry (Fu and He, 2001).

The main objective of the work is to determine the modal parameters from the numerical modeling in finite elements of the experimental aircraft's wing structure, designed by the Draco Volans Aerodesign team for the XX SAE Brazil Aerodesign Competition.

2. Background theory

The modeling of discrete systems with n degrees of freedom proposes the solution of a set of n ordinary differential equations (Rao, 1986), given by Eq. 1.

$$[M]\{\ddot{x}\} + [C]\{\dot{x}\} + [K]\{x\} = \{F(t)\} \tag{1}$$

Where [M], [C] and [K] are the $n \times n$ mass, damping and rigidity matrices respectively, $\{\ddot{x}\}$,$\{\dot{x}\}$,$\{x\}$,{F(t)} are $n \times 1$ vectors of acceleration, velocity, displacement, and force.

To solve the modal analysis problem, upon imposing the boundary conditions, the typical eigenvalue problem described in Eq. 2 is solved for the non-trivial solution, where $\{X\}$ is the nodal displacement vector (which is known as the "modal vector"), and ω results in the natural frequency (eigenvalue associated to the modal eigenvector).

$$([K] - \omega^2[M])\{X\} = 0 \tag{2}$$

For the finite element method solution of the modal analysis problem of a discrete structure, the matrices $[K]$ and $[M]$ must be determined for each component element of the discretization adopted. It is determined by Eqs. 3 and 4, obtained from the finite element method variational formulation, as described in Petyt (2010).

$$[M_i] = \int_V [N_i]^T \rho [N_i] dV \tag{3}$$

$$[K_i] = \int_V [B_i]^T [D] [B_i]^T dV \tag{4}$$

Defining,

$$[B_i] = \frac{\partial N}{\partial x_i} \tag{5}$$

Where $[B_i]$ is the deformation-displacement matrix, $[N_i]$ is the shape function matrix, defined for an element i of the global set, and $[D]$ is the elasticity matrix of the material, which depends on Young's modulus and shear parameters, and Poisson's coefficient. The elastic matrix for isotropic and orthotropic materials is described in Eqs. 6 and 7 (Jones, 1998).

$$[D] = \begin{bmatrix} \frac{E}{1-\nu^2} & \frac{E\nu}{1-\nu^2} & 0 \\ \frac{E\nu}{1-\nu^2} & \frac{E}{1-\nu^2} & 0 \\ 0 & 0 & \frac{E}{2(1+\nu)} \end{bmatrix} \tag{6}$$

$$[D] = \begin{bmatrix} \frac{1}{E_x} & -\frac{\nu_{yx}}{E_y} & -\frac{\nu_{zx}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{xy}}{E_x} & \frac{1}{E_y} & -\frac{\nu_{zy}}{E_z} & 0 & 0 & 0 \\ -\frac{\nu_{xz}}{E_x} & -\frac{\nu_{yz}}{E_y} & \frac{1}{E_z} & 0 & 0 & 0 \\ 0 & 0 & 0 & \frac{1}{G_{yz}} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{1}{G_{xz}} & 0 \\ 0 & 0 & 0 & 0 & 0 & \frac{1}{G_{xy}} \end{bmatrix} \tag{7}$$

Where E is the modulus of elasticity, ν is the Poisson coefficient, G is the shear modulus, with respect to the planes xy, xz and yz and their symmetric planes. Since all parameters can be explicitly determined, the equation can be solved directly. A set of natural frequencies for each mode can then be calculated. The eigenvectors of each natural frequency, which are displacements of each node under a given mode can also be calculated. This solves the proposed dynamic problem.

For a continuous system, an infinite number of mode functions can be defined, where, for each, it is associated with a natural frequency. The general equations for the relative parameters are given by Eq. 8 and 9.

$$\begin{aligned} W(x) =& C_1(cos\beta x + cosh\beta x) + C_2(cos\beta x - cosh\beta x) + \\ &+ C_3(sen\beta x + senh\beta x) + C_4(sen\beta x - senh\beta x) \end{aligned} \tag{8}$$

$$\omega = \beta^2 \sqrt{\frac{EI}{\rho A}} = (\beta l)^2 \sqrt{\frac{EI}{\rho A l^4}} \tag{9}$$

Here, C_1, C_2, C_3, C_4 are constants that can be determined by means of the boundary conditions imposed for the solution of the proposed system, just as the value of β, E is the equivalent modulus of elasticity of the material, I is the moment of inertia of the cross section of the beam, ρ is the specific mass of the material, l is the effective length of the beam, A is the cross-sectional area. For the free-free boundary condition, values for the constant βl are presented in the Tab. 1. Isolating E from Eq. 9 and experimentally determining ω, it is possible to estimate the elasticity modulus of a beam structure, principle that will be used in this work.

Table 1: Values for the constant $\beta_n l$

$\beta_n l$	Numerical value
$\beta_1 l$	1,875104
$\beta_2 l$	4,694091
$\beta_3 l$	7,854757
$\beta_4 l$	10,995541

3. EXPERIMENTAL AND NUMERICAL CASE STUDY OF TUBULAR MAIN WING SPAR

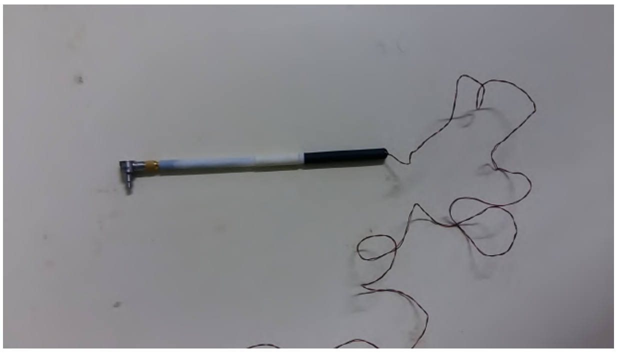
The analytical theory behind obtaining FRFs is applied and a method is proposed for obtaining the equivalent rigidity of beams by means of a case study of a component with simple geometry that is used as the structural part of the aircraft, the main spar. In this study, the modal parameters of vibrational mode and natural frequency inherent to the component are evaluated by analytical, numerical approaches (FEA modeling) and experimental tests by free-free transient excitation, comparing numerical results for a beam element with the finite elements and, finally, determining an equivalent stiffness value for the referent component.

The present case study aims to verify the application of the developed theory for lateral vibration of beams, by the numerical approach, associated with an experimental procedure to determine the FRF of acceleration by transient excitation in a CP that represents the structure of a main spar with geometry and dimensions commonly used by the SAE Aerodesign team of University of Brasília in the manufacturing of prototypes, basing in concepts developed in Ewins (1984). Initially, a simplified material validation method (aluminum tube 6063, CP-01) is analyzed, and a carbon fiber laminated thin-walled tube (composite, CP-02) is subsequently analyzed.

Transient impact vibration tests were performed in the CPs, in free-free boundary condition (suspended in light foam). The input force was applied with an impact hammer (model 084A14) and the acceleration measured using a uniaxial accelerometer (model ICP® 352A21). The sensor and impact hammer are connected to the data acquisition board (VIB-E-220 model) and the Polytec Vibrometer software containing the FFT analyzer, the FRF of the excitation in the structure could be obtained, which, therefore, enables us to determine the first resonance frequencies of the structure, which approximate its natural frequency, as shown in Rao (1986). A set of 5 repetitions of measurement was performed aiming to guarantee consistency in vibration response patterns, as recommended by Avitabile (2017). Figure 1 presents the testing setup.

(a) (b)

(c)



(d)

Figure 1: (a) Experimental set-up for free-transient vibration test at CP-01, (B) Fixing the accelerometer on one end of the CP-02, detail on the use of the foam on the sides of the tube to prevent rotation of the tube, (c) Data acquisition hardware model VIB-E-220, (d) Impact hammer (model 084A14)

Results obtained experimentally for the CP-01 were compared with a numerical analysis using the FEA, in ANSYS APDL, by using linear beam elements (BEAM188) with converged mesh (Brancheau, 2015), as shown in Tab. 2 and Fig. 2. The Young Modulus was obtained by the modeling of the beam using Euler-Bernoulli's formulation (Rao, 1986) and the resonance frequencies associated to the structure. Those frequencies were used in order to determine an average value to the Young Modulus. The maximum percentage error obtained between the peaks was 1,64%. The resulting value for the Aluminum 6063 Young Modulus was $68,9$ GPa, close to the one described in the literature (MATWEB, 2018).

Table 2: Comparison between natural frequencies for CP-01 obtained experimentally and numerically for finite element BEAM188

Frequency	**Experimental [Hz]**	**BEAM188**	
		[Hz]	**ERR%**
1	261.3	265.58	1.64
2	705	716.41	1.62
3	1346	1362.4	1.22
4	2155	2168.8	0.64
5	3098	3103.5	0.18
Number of FE for result convergence		186	

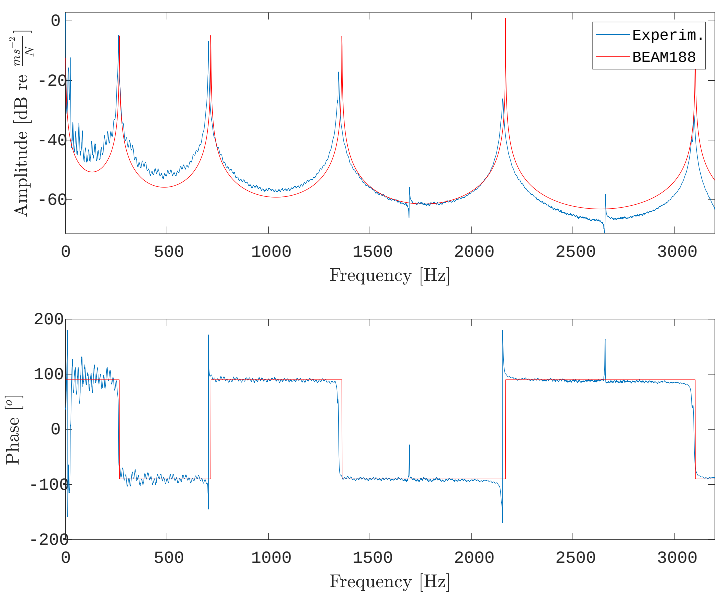


Figure 2: Comparison between experimental and numerical FRF (element BEAM188) for CP-01

Now, for the CP-02, after obtaining its resonance frequencies, the procedure of determining the equivalent modulus of elasticity of the composite beam for flexural behavior was performed. The procedure performed was similar as explained in the CP-01 case study, by obtaining an equivalent mean stiffness value, used as a variable parameter in the fit process of the experimental and numerical FRFs, obtained from a simulation where a forcing was applied harmonic, covering the frequency range determined by the experimental test. The numerical iteration, in which every iteration means varying the equivalent elasticity modulus, followed until a relative error value of less than 2% was determined between the resonance frequency peaks and the natural numerical frequencies. The Young Modulus obtained after the iterations was 24,3 GPa. Although CP-02 is classified as an orthotropic composite, which has directional properties and symmetry planes of the material (Mendonça, 2005), good convergence of the experimental and numerical results was observed for the first two natural frequencies, using element BEAM188 and the material modeled as isotropic. The results after the iterations are presented in Fig. 3 and Tab. 3.

Table 3: Comparison between natural frequencies for CP-02 obtained experimentally and numerically

Ressonance Frequency (experimental) [Hz]	**Natural Frequency (numerical) [Hz]**	**Percentual Error**
288,8	283,0	2,0%
746,3	747,0	0,094%

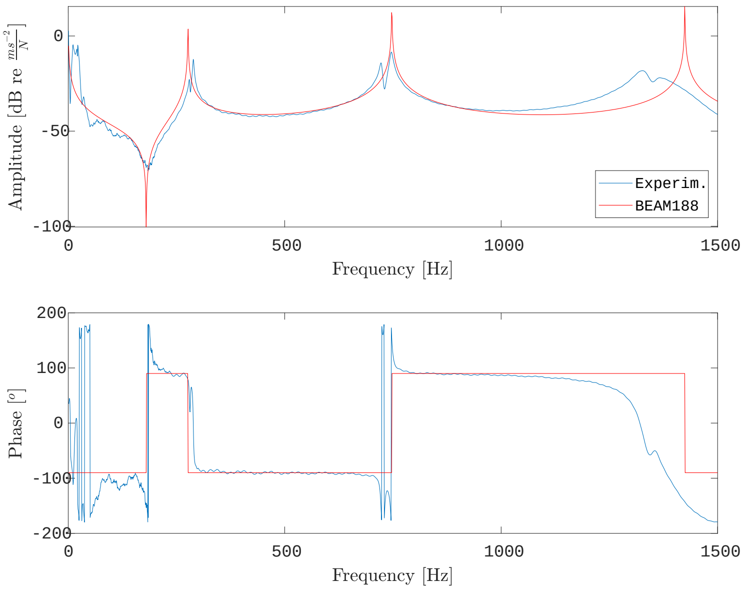


Figure 3: Comparison between experimental and numerical FRF (element BEAM188) for CP-02

4. NUMERICAL ANALYSIS OF THE AIRCRAFT WING PROTOTYPE

The aircraft designed by the Draco Volans team, to meet the specifications established in the regulation of the XIX SAE Aerodesign Competition, assumed a conventional airplane concept, which presents performance advantages in several areas in relation to other aircraft concepts for the imposed regulation. The design is composed of two independent parts, one is the structure of the fuselage and tailboom, the other is composed by the wing, object of study of the work. The following are construction details, the properties of the materials used, the finite element modeling, and finally the results obtained.

The structure of the aircraft has a wingspan of 2126 mm, chord at the root of 496 mm, chord at the tip of 291 mm and was designed to withstand critical situations of loading in flight and forced landing, besides having a compartment to accommodate the embarked avionics and of the load carried. The elements that make up the main structure are the central ribs of laminated sandwich plates of structural foam and carbon fiber, carbon tubes pultruded to the end wing spars, thin wall tube laminated with carbon fiber bidirectional as the main spar, ribs and leading and trailing edges in balsa wood, besides a part of the leading edge is made in Styrofoam F7. The final structural layout is shown in Fig. 4. The main dimensions of the structure are shown in Fig. 5. The components and their locations in the wing are listed and detailed in Tab. 4.

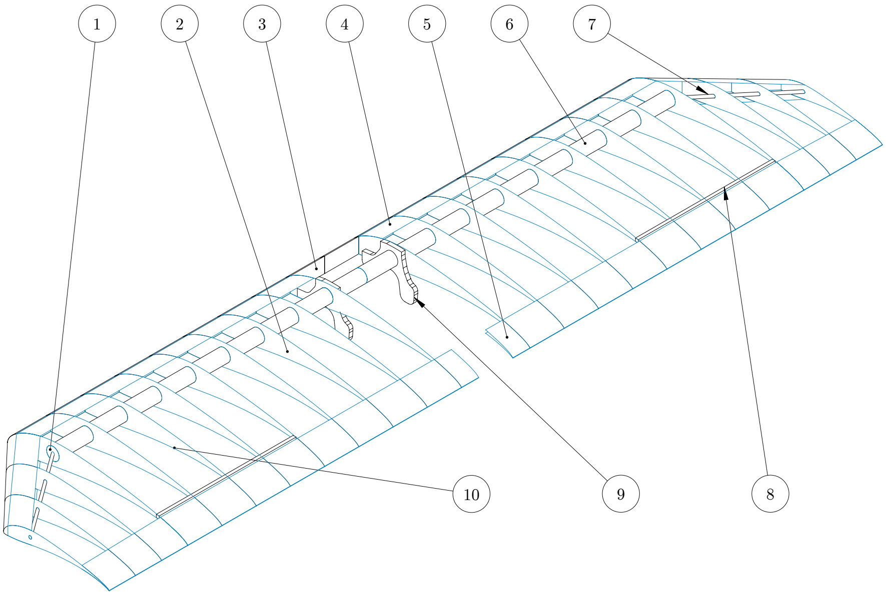


Figure 4: Structural layout and wing detailing designed by Draco Volans Aerodesign team

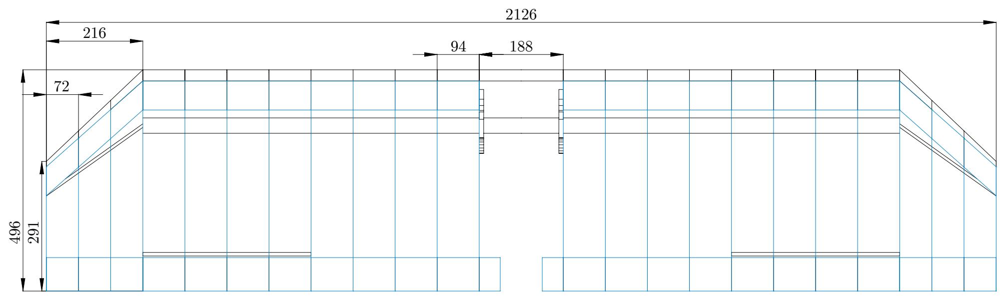


Figure 5: Wing dimensions designed by the Draco Volans Aerodesign team

Table 4: Detailing of prototype components

Item	Specification
1	Sleeve for secondary spars (thickness: 25 mm, 2 units)
2	Balsa wood rib (thickness: 3 mm)
3	Styrofoam F7 leading edge
4	Balsa wood leading edge (thickness: 1 mm)
5	Balsa wood trailing edge (thickness: 1 mm)
6	Main spar (laminated carbon fiber and epoxy resin)
7	Secondary spar (carbon fiber pultruded tube - OD 8 mm ID 5.5 mm)
8	Aileron spar
9	Central rib (2 units)
10	Aileron servo-controller (2 units)

The cover of the wing is made with adhesive plastic called *MicroLite*. All components of the lateral section are fixed by TEKBOND 793 quick curing glue, which fills gaps up to 0.1 mm, according to the manufacturer TEKBOND (2014). The components of the central section were joined by the application of AMPREG A-26 SLOW resin, due to its greater resistance compared to glue.

The objective now is to present the EF modeling, based on the pre-processing steps of a numerical simulation (ANSYS, 2009). Afterwards, the results obtained for the first vibrational modes will be presented.

The mechanical properties of rigidity and specific mass of the materials composing the prototype structure are summarized in Tab. 5. It is important to note that the total plastic cover mass was considered together with the density used for balsa wood. The assumption was made that at each rib has the mass of the plastic cover section of half from each side. Thus, it was obtained a balsa density and plastic cover integrated at the end.

Table 5: Material properties

Material	Elasticity modulus (GPa)	Specific mass (kg/m^3)
Balsa wood (low specific mass)[1]	1,2	160
Microlite cover[1]	-	200
Pultruded carbon-fiber tube[2]	28,0	1500
TekBond glue[3]	-	1,05
Styrofoam F7[4]	2,0	1050
laminated carbon fiber and epoxy resin (bidirectional)	24,3	969

[1] Data obtained from tests performed by the team
[2] Data provided by the manufacturer (ACP, 2018)
[3] Data provided by the manufacturer (TEKBOND, 2014)
[4] Data provided by the manufacturer (KNAUF, 2019)

Three types of elements were selected: BEAM188 (linear element - 2 nodes - 6 degrees of freedom at each node - translation and rotation), SHELL181 (quadrilateral element - 4 nodes - with 2 degrees of freedom in each node - translation only) and MASS21 (structural mass point element with 6 degrees of freedom - translation and rotation). The latter element was associated with different types of components treated as point masses in the wing structure, which are described in the Tab. 6.

Sections have been associated with each part of the geometry to be drawn. One simplification implemented was a mass equivalence study between the balsa wood ribs. They were modeled as continuous rectangular sections, where they remained the same length and thickness as the original structure, varying only the height of the section, conserving constant the overall mass of the component. The same was done with the F7 styrofoam-shaped leading edge (item 3, Fig. 4), now establishing a square section to hold the equivalent mass of the component, one for the straight section and one for the tapered section. The Tab. 6 presents the sections used in the modeling.

Table 6: Sections of the structural components of the modeled wing

Component	Element	Section	Value	Material
Main spar	BEAM188	Circular-tube	17,275(ER) 17,025(IR) mm	Carbon fiber-epoxi laminate
Secondary spar	BEAM188	Circular-tube	4(ER) 2,75(IR) mm	Pultruded carbon fiber-epoxi
Styrofoam leading edge (straight section)	BEAM188	Squared	24,865 × 24,865 mm	Styrofoam F7
Styrofoam leading edge (tapered section)	BEAM188	Squared	12 × 12 mm	Styrofoam F7
Ribs (straight section)	BEAM188	Rectangular	28,99 × 3 mm (height × thickness)	Balsa wood
First rib (tapered section)	BEAM188	Rectangular	24,77 × 3 mm (height × thickness)	Balsa wood
Second rib (tapered section)	BEAM188	Rectangular	19,16 × 3 mm (height × thickness)	Balsa wood
Third rib (tapered section)	BEAM188	Rectangular	13,70 × 3 mm (height × thickness)	Balsa wood
Aileron spar	BEAM188	Squared	10 × 10 mm	Balsa wood
Leading and trailing edge cover	SHELL181	Shell	2 mm (Thickness)	Balsa wood
Aileron servo-controller	MASS21	-	6 g	-
Sleeve for secondary spars	MASS21	-	10 g	-
Central rib	MASS21	-	15 g	-

Table 7 brings the comparison between the mass of the real structure and the mass of the numerical model developed, showing good proximity.

Table 7: Comparison between the actual mass of the structure and the mass obtained in the numerical model

Measured mass (g)	Numerical mass (g)	Err%
645,60	644,78	0,13%

The geometry of the model was entirely made within the ANSYS APDL design modeler. Making the geometry within the finite element software brings advantages in that it can define the numbering of the nodes to be generated in the element mesh, in addition to making the simulation computationally less costly compared to a geometry import in IGES format, STEP, or PARASOLID (ANSYS, 2009). The second is a point of unique importance due to the fact that the developed routine will be used in an iteration algorithm, which will need to use ANSYS APDL as a solver of the FEM equilibrium equations. Thus, the lower the processing time of the simulation, the faster the optimal solution will be determined.

The next step is to create the finite element mesh and establish the level of discretization for convergence of the response minimizing the computational cost. After a mesh convergence study, the graphs of the Fig. 6 were obtained, concluding that a total of 6031 nodes are required to converge the model result to the fourth decimal place of the calculated frequencies.

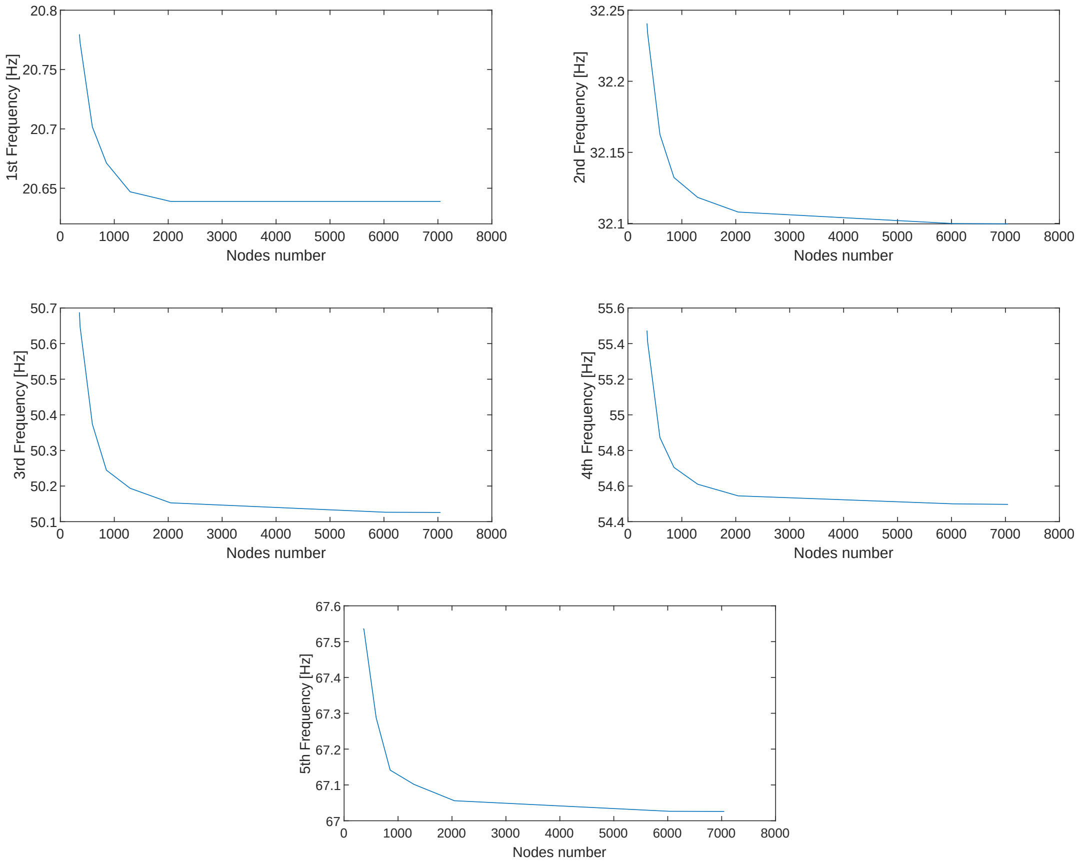
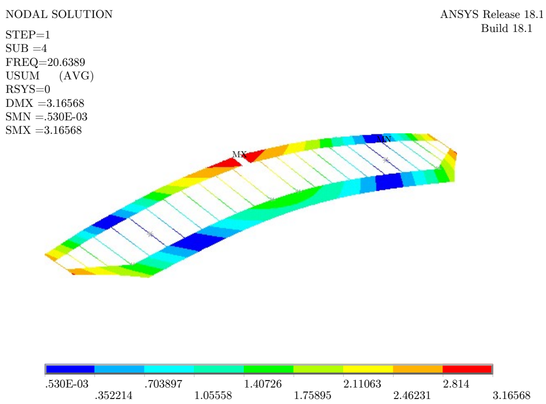
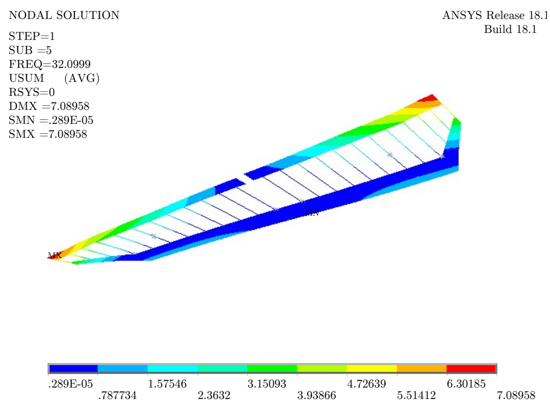


Figure 6: Mesh convergence of the DV-2017 wing model for all numerical frequencies

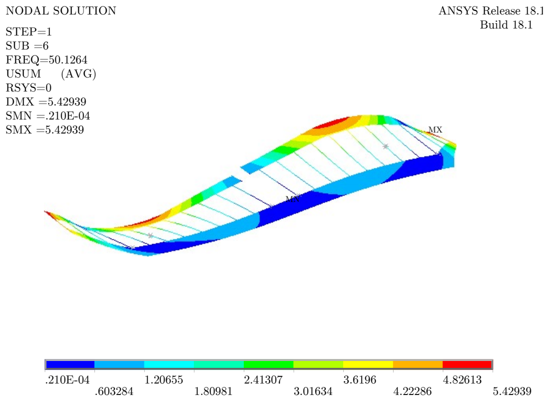
The results are presented for the modal numerical analysis in free-free boundary condition of the developed model. The modes identified are in a frequency range of 0 to 70 Hz, and are also summarized in Tab. 8, which were compared with experimental results from a Ground Vibration Testing (GVT) performed in the aircraft wing structure.



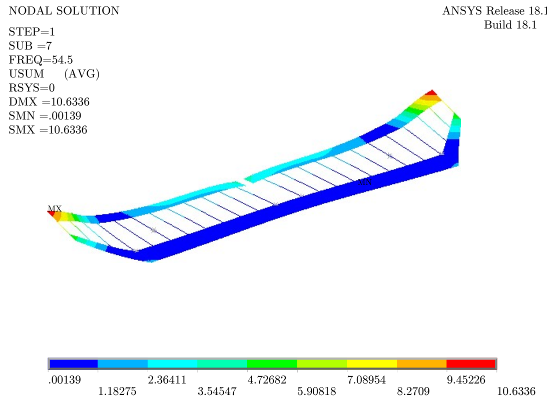
(a) First bending mode



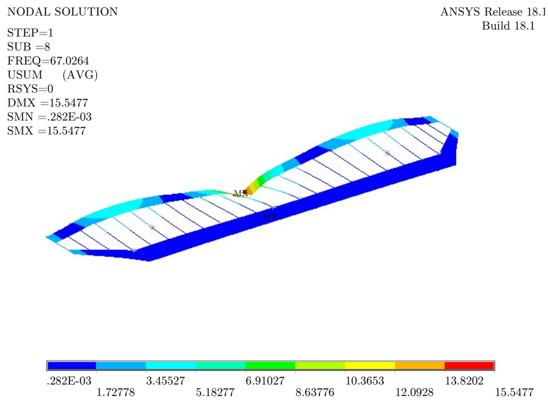
(b) First torsion mode



(c) Second bending mode



(d) Second torsion mode



(e) Third bending mode

Figure 7: Numerical mode shapes and frequencies of the wing prototype studied

Table 8: Results of the modal numerical analysis (from 0 to 70 Hz)

Characteristic mode	Numerical Frequency [Hz]	Experimental Frequency [Hz]	Error %
First bending mode	20,6389	25,928	20,399
First torsion mode	32,0999	31,716	-1,211
Second bending mode	50,1264	47,545	-5,618
Second torsion mode	54,5000	55,462	1,735
Third bending mode	67,0264	67,222	0,292

Thus, the average percentual error obtained was 5,813%, considering the absolute valor in order to calculate the averaged result.

5. CONCLUSIONS

In order to apply the theory developed in the text and establish an algorithm to determine the equivalent stiffness of a structure that behaves as a beam, a case study with two test specimens characterizing structures used in the manufacture of Aerodesign prototypes was performed, comparing results obtained experimentally and numerically, through FEM analysis, considering different types of elements. For this, a transient vibration test was performed by means of impact hammer excitation, where it was possible to obtain a frequency response function for each system. The result of the analysis, after the data treatment, showed that the application of the Euler-Bernoulli beam theory formulation returns an average perceptual error for thr elastic modulus estimation of 0.29 % for the aluminum 6063 tubular specimen and 2 % for the carbon fiber tube laminated with epoxy resin, representing excellent estimates for implementation in numerical modeling. The next stage of the work consisted of making the FE modeling of the complete structure of the prototype wing. For this, geometric simplifications were assumed in order to make the numerical model parameterizable, though considering all the materials and their equivalent mechanical properties present in the structure. One-dimensional beam elements (BEAM188), two-dimensional shell (SHELL181) were used. In addition, localized mass of several elements that compose the wing were considered, in order to make the numerical model the most consistent with the real, through the use of MASS21 elements. The numerically modeled mass of the wing diverged 0.13 % of the actual mass of the structure. It was presented a mesh convergence of the model and the results for a frequency range of 0 to 70 Hz. The averaged percentual difference between the experimental results and the numerical model was 5,813%, with the highest difference observed in the first natural frequency (corresponding to the first bending mode).

6. ACKNOWLEDGEMENTS

The authors are thankful to the support of University of Brasília, specially to the Group of Dynamic Systems, to make this work possible.

7. REFERENCES

ACP, 2018. *Carbon fiber uniroll wrapped and pultruded tubes datasheet*.
ANSYS, 2009. *ANSYS User's Manual: Structural Analysis Guide*.
Avitabile, P., 2017. *Modal Testing: A Practitioner's Guide*. Wiley.
Bisplinghoff, R., 1996. *Aeroelasticity*. Dover Books on Aeronautical Engineering Series. Dover Publications.
Brancheau, J.E., 2015. *Practical Aspects of Finite Element Simulation - A Study Guide*. Altair.
Ewins, D., 1984. *Modal testing: theory and practice*. Mechanical engineering research studies: Engineering dynamics series. Research Studies Press.
Fu, Z. and He, J., 2001. *Modal Analysis*. Elsevier Science.
Jones, R., 1998. *Mechanics Of Composite Materials*. Materials Science and Engineering Series. Taylor & Francis.
KNAUF, 2019. "Eps isopor". URL `https://www.knauf-isopor.com.br/produtos/servicos-em-inovacao/eps/`.
MATWEB, 2018. "Aluminum 6063-o mechanical properties". URL `http://www.matweb.com/`.
Mendonça, P.T.R., 2005. *Materiais compostos e estruturas-sanduíche: projeto e análise*. Editora Manole.
Petyt, M., 2010. *Introduction to Finite Element Vibration Analysis*. Cambridge University Press.
Rao, S.S., 1986. *Mechanical vibrations*. Addison-Wesley Longman.
TEKBOND, 2014. *Ficha técnica TekBond 793*.

8. RESPONSIBILITY NOTICE

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