XLIII CILAMCE
21-25, November, 2022
FOZ DO IGUAÇU - PR - BRAZIL

Numerical modeling for deformation analysis of a cantilever beam

Gilsomaro Barbosa de Melo Silva[1], Rosilda Sousa Santos[1], Diego David Silva Diniz[2], Walber Medeiros Lima[2], Jackson de B. Simões[2]

[1]Departament of Science and Technology, Rural Federal University of the Semi-Arid
RN-233, Zip Code 59780-000, Caraúbas-RN, Brazil.
E-mail: gilsomaro.silva@alunos.ufersa.edu.br and rosilda.santos@ufersa.edu.br
[2] Department of Engineering, Rural Federal University of the Semi-Arid
E-mail: diego.diniz@ufersa.edu.br, walber@ufersa.edu.br and jackson.simoes@ufersa.edu.br

Abstract. The tireless quest for better living conditions makes people use technological resources as manners to balance a work/economy/safety relationship. Those technological resources have been applied more frequently in engineering. This is due to development of simulation software in order to provide greater economic viability in engineering projects. The considerable growth in the infrastructure market, further promoted the application of beams in projects, linked to this is the concern with the deformations that are caused to them. In this sense, the deformation suffered by a beam was evaluated via structure analysis using the ANSYS Mechanical software. The beam is at one extremity and supported by fixed support at the other extremity. To obtain the values of deformations a beam was developed in ANSYS, and after, the deformation was found through finite element methods. To verify the methodology effectiveness proposed in this study the numerical results were compared to data obtained in the tests and theoretical calculations. Through the values obtained, it was concluded that the values obtained in the simulation present deformations very similar to the tests and the theoretical calculation.

Keywords: ANSYS mechanical, deformations, numerical simulation, finite element methods.

1 Introduction

According to Hibberler (2010), beams are considered the most important structural and mechanical elements in engineering due to their wide functionality. Usually, they are widely used in bridge structures, floor support, airplane wings, automobile axles, among other elements that bring immense benefits to society in general.

With the study of these beams, it is possible to verify from the appropriate material to be used, the dimensions that must be placed, as well as the reactions that these materials undergo over time. Based on this, it is important to know the deformation that the beams undergo when undergoing any type of stress or disturbance.

Although there are already multiple techniques for measuring deformations, it is necessary to adopt the one that best suits the characteristics of the structural system, presenting a higher level of accuracy and low cost (SILVA, 2012). For this purpose, a beam was generated in ANSYS Workbench 19.2 and, through the numerical simulation performed in this software, the variable of interest was verified, that is, the measurement of the deformation suffered by the cantilevered beam. With the numerical results obtained, it is intended to compare with data reported in the experiment and in the literature.

It is important to highlight that this work was developed at a time when numerical simulations are gaining increasing importance, and where the use of numerical simulators is more integrated in the improvement of the use of beams in constructions. Therefore, it is important to verify the deformation suffered by a beam through the ANSYS software.

2 Methodology

To perform the research and analyze the deformation of the beam, the software ANSYS Workbench R19.2

was used. To run the simulation in ANSYS software, a Samsung notebook was used, with 4GB RAM and 500 GB HD. In addition, "Origin" was used as an auxiliary tool to manage the data and plot the graph (Force x Strain). Due to the large amount of values, it was necessary to use the Excel program in order to apply statistical studies.

2.1 Geometry of the object of study

In order to use the finite element method and the calculations required to find the deformation of a beam when subjected to a force, it was initially necessary to characterize and size the object of study. Thus Figure 1 represents the beam designed in the software as well as its dimensions. Considering that these dimensions were collected from the works of Araújo (2017) and Santos (2018).

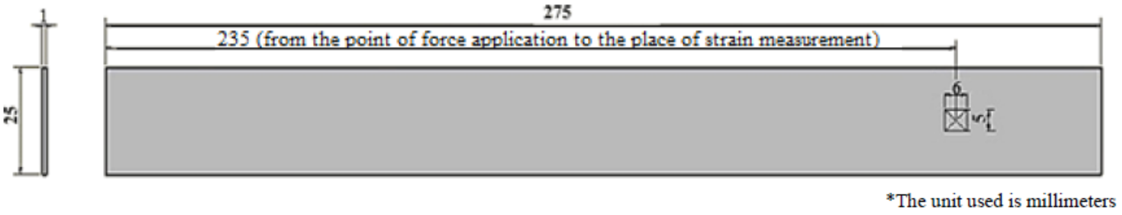

Figure 1. Projection of the beam and its dimensions

2.2 Mesh generation

After the construction of the geometry, we started the methodology of construction of the numerical mesh used in the simulations for this work. To build this object it was necessary to have knowledge about the material used in the experiment (stainless steel), the modulus of elasticity approximately (208.4 Gpa), the length of the beam (275mm), the width (25mm), the height (1mm), the force that was being applied (Fi = 0.187 N varying up to Ff = 1.587 N), and the distance from the point of force application to the measurement equipment reading location (235mm).

All parameters mentioned are the same as those entered in ANSYS Mechanical, except for the beam material and modulus of elasticity, which consist respectively of standard structural steel and a modulus of elasticity equal to 200 Gpa.

In addition, the mesh was refined so that the values found for the displacements, stresses and strains were the closest to the analytical values. The parameters changed to perform the refinement were: changing the number of nodes of the mesh in order to increase the number of elements (parts) in the geometry and finally, to conclude that the results are as close to reality, the mesh was submitted to the convergence test, for each force applied to the beam a new convergence test was performed and soon after the verification that the mesh was converging, the value of the strain was noted.

According to Cunha (2014), it is important to have a control of the mesh quality in order to have a control of the discretization errors. In other words, to obtain more accurate numerical solutions it is important to reduce their source of errors. Numerical uncertainty errors cannot be avoided, but they can be minimized. In view of minimizing these errors, mesh refinement is important. In summary, to obtain satisfactory results it was necessary to refine the mesh, increase the number of nodes in the mesh, and perform the convergence test. Figure 2 shows the final model of the geometry after refinement containing 393 nodes and 40 elements.

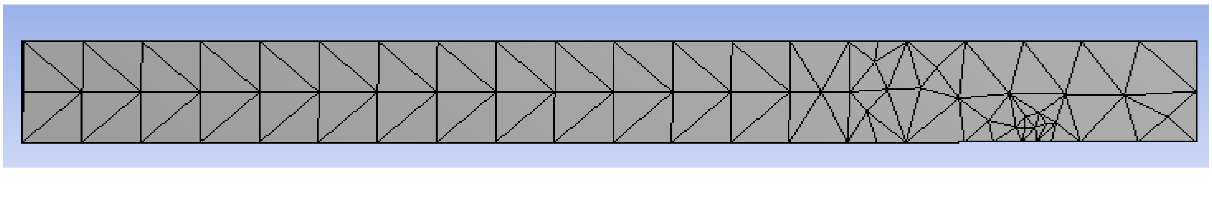

Figure 2. Refined mesh

2.3 Case study

In general, for the elaboration of the structural arrangement, the following general aspects were considered:

a) The beam is being crimped by a support fixed at one end so that the force is applied at the opposite end;

b) The standard material used by the software consists of a structural steel whose modulus of elasticity (E) is 200 Gpa;

c) $F_i = 0.187\ N$ was used, varying up to $F_f = 1.587\ N$ being applied in the y direction;

d) After the implementation of the mesh, it was used a measuring instrument that could measure the deformation of the beam from the force application point to the reading location. This instrument, called "probe", serves to precisely measure the deformation of the beam in order to provide better accuracy of the results. This measurement in which the deformation values were collected is due to the experiment that was performed and in which a strain gauges was inserted in order to find the value of the deformation suffered by the beam, and this distance between the force application point to the reading location is 235 mm.

3 Results

The Figures 3 and 4 illustrate the evolution of the stress and strain gradient of the beam for a force (F=0.187N) applied in the y direction, this scale indicates the minimum and maximum values of stresses and strains.

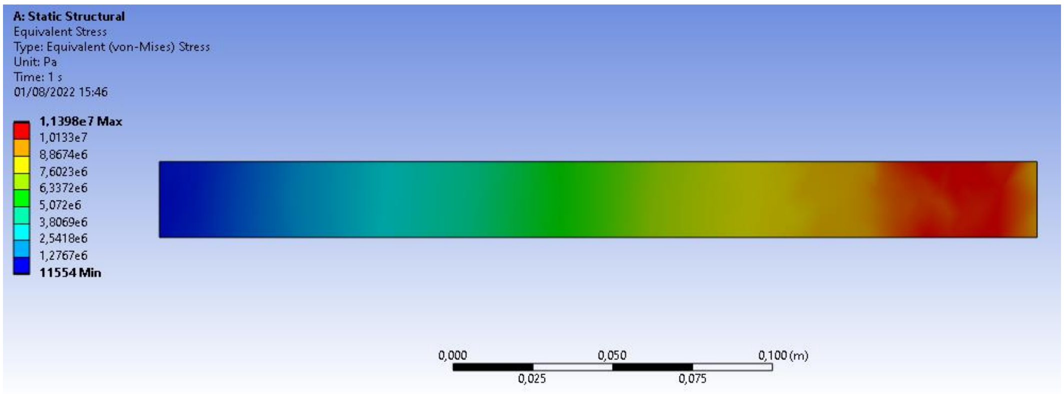

Figure 3. Stress color analysis

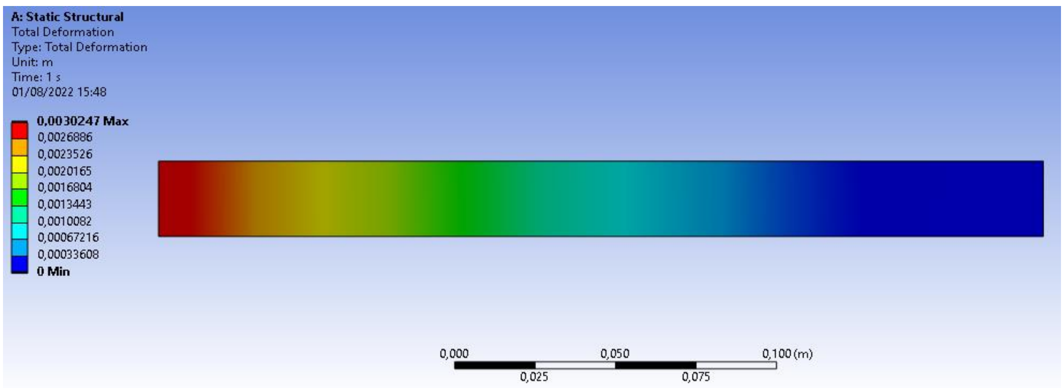

Figure 4. Color analysis of total deformations

A closer inspection of Figures 5 and 6 allows visualizing the color gradient of the variable of interest, which is the analysis of the deformation suffered by a beam when it is subjected to a force, in addition, the Figures also allow a detailed visualization the distance between the point of application of the force to the location of the deformation reading.

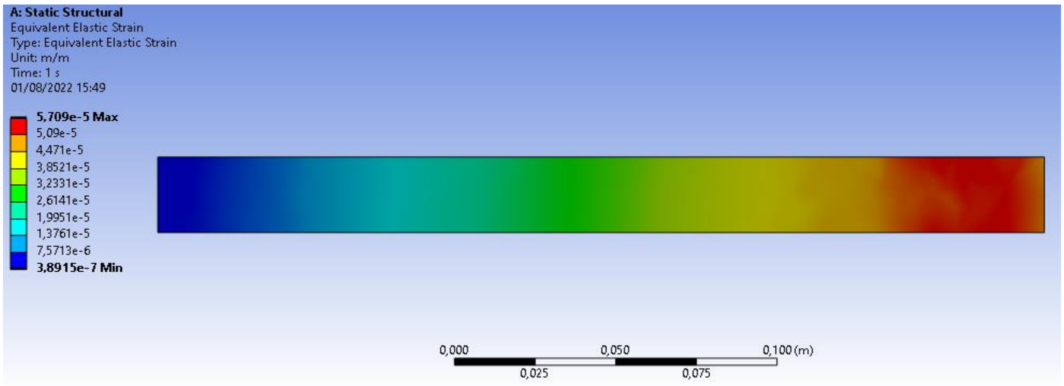

Figure 5. Strain color analysis, mesh for data collection

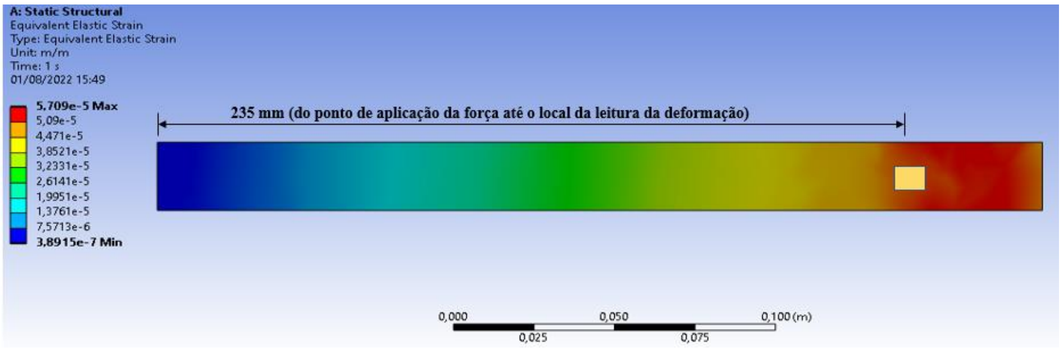

Figure 6. Distance from data collection

From these simulations and using the "probe" tool at the midpoint of the rectangle (representation formatted in PowerPoint illustrating how the results were collected) that characterizes the reading location, it was possible to plot a graph that allows a comparison between the experimental results, theoretical with the simulation results. For this, Figure 7 demonstrates this comparison between these values.

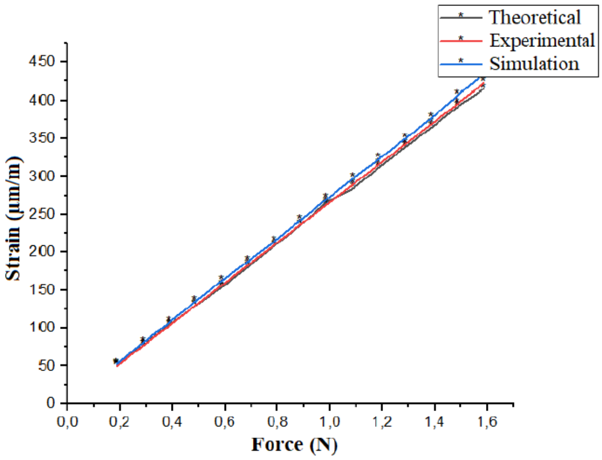

Figure 7. Comparison between the values of experimental and theoretical measurements

The graph shows that in both methods the strain values had similar behavior. Therefore, the three methods are reliable and can be used in strain measurements. According to Table 1, it is possible to verify the statistical data and from these it is concluded that the errors were low considering that the average margin of error of the experimental results in relation to the simulation results is approximately 3.74%.

Table 1. Averages of experimental and theoretical strains compared to simulation

F (N)	ε_{exp} (μm/m)	ε_{teo} (μm/m)	$\varepsilon_{Simulation}$ (μm/m)	Error (%)
0.187	050.76	049.84	052.67	3.75
0.287	076.95	076.46	080.49	4.60
0.387	104.08	103.10	107.57	3.36
0.487	129.44	129.70	135.12	4.38
0.587	153.55	156.30	162.19	5.63
0.687	180.59	182.90	187.96	4.08
0.787	208.30	209.50	213.92	2.70
0.887	235.29	236.10	242.17	2.93
0.987	266.27	262.70	269.96	1.39
1.087	284.10	289.30	297.15	4.59
1.187	311.71	316.00	323.06	3.64
1.287	338.89	342.60	350.00	3.28
1.387	364.70	369.20	377.04	3.38
1.487	392.12	395.80	406.52	3.67
1.587	414.97	422.40	434.55	4.72

4 Conclusions

Considering the results obtained through numerical simulation, based on the finite element method (FEM) it was concluded that the values obtained in the simulation present deformations very similar to the experimental tests and the theoretical calculation. Thus, it was observed that the computational methodology used proved to be very efficient for the simulation of cantilevered beams and can be used to study the phenomena and parameters existing in this type of application.

References

[1] A. L. Cunha; J. S. Souza; S. R. Farias Neto; A. G. B. Lima; E. S. Barbosa. "Separation Process by Porous Membranes: A Numerical Investigation". *Advances in Mechanical Engineering* (New York), v. 2014, p. 1-9, 2014.

[2] P. G. M. Flávio. "Simulação numérica de banco veicular dianteiro: impacto traseiro", University of Brasilia, 2015. Accessed on May 5, 2022. Available at: <https://fga.unb.br/articles/0001/0188/Paulo_Marques_TCC2.pdf>.

[3] R. C. Hibbeler. Resistência dos Materiais. 7ª Edition. São Paulo: PEARSON, 2010.

[4] W. M. Lima. "Plataforma para análise comportamental de atuadores de ligas com memória de forma e para o controle de deformação de uma barra flexível". Thesis (Master's degree). Campina Grande: Federal University of Campina Grande, 2008.

[5] P. R. de Araújo. "Instrumentação virtual para medição do deslocamento linear de uma barra flexível atuada por fio de liga com memória de forma: *LabVIEW^{TM}*". 2017. Monography – Electrical engineering, Rural Federal University of the Semi-Arid – UFERSA, Caraúbas, 2017.

[6] N. F. Queiroz. Tutorial Ansys Workbench 11.0: Simulação com CFD de escoamento de fluxo ao redor de um edifício de 10 andares. Federal University of Rio Grande do Norte – UFRN, 2008.

[7] W. J. S**.** Silva. "Monitoração estrutural e instrumentação virtual aplicados ao ensino experimental de engenharia civil"**.** In: XL Brazilian Congress of Engineering Education (XL COBENGE), 2012.

[8] M. A. B. Santos. "Plataforma para análise da deformação de uma viga simplesmente engastada". 2018. Monography – electrical engineering, Rural Federal University of the Semi-Arid – UFERSA, Caraúbas, 2018.