Modal Analysis of Titan Cantilever Beam Using ANSYS and SolidWorks

Pavol Lengvarský[*], Jozef Bocko, Martin Hagara

Department of Applied Mechanics and Mechatronics, Technical University of Košice, Košice, Slovakia
*Corresponding author: pavol.lengvarsky@tuke.sk

Abstract In this paper are given basic steps of the modal analysis simulation. The modal analysis is carried out on the titan cantilever beam. The cantilever beam is designed in the graphical environment of the ANSYS and SolidWorks. The cantilever beam was fixed on one end and all degrees of freedom on this end were taken, beam cannot move and rotate. Mode shapes and natural frequencies are computed in programs ANSYS and SolidWorks with numerical formulation of the direct solver including the block Lanczos method.

Cite This Article: Pavol Lengvarský, Jozef Bocko, and Martin Hagara, "Modal Analysis of Titan Cantilever Beam Using ANSYS and SolidWorks." *American Journal of Mechanical Engineering* 1, no. 7 (2013): 271-275. doi: 10.12691/ajme-1-7-24.

1. Introduction

The modal analysis deals with the dynamics behavior of mechanical structures under the dynamics excitation. The modal analysis helps to reduce the noise emitted from the system to the environment. It helps to point out the reasons of vibrations that cause damage of the integrity of system components. Using it, we can improve the overall performance of the system in certain operating conditions. We know two basic methods of the modal analysis, namely the numerical modal analysis and the experimental modal analysis. The experimental modal analysis deals with measurement input data from which a mathematical model is derived. However, it has to take different levels of analysis, from which the model is constructed. The main part of this analysis consists of determining the curve FRF [1].

2. Cantilever Beam

In Figure 1 is shown design of the cantilever beam with its dimensions. It is made of Titanium alloy: Ti - 5Al - 2.5SN, with material properties $E=1.103e^{11}$ Pa, $\mu=0.31$, $\rho=4480$ kg/m^3, $Rm=861e^6$ Pa, $Re=827e^6$ Pa.
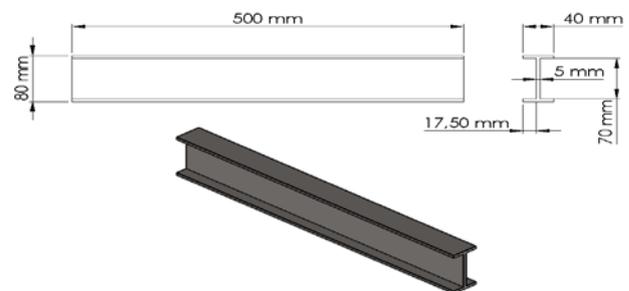

Figure 1. Design of cantilever beam

2.1. Modal Analysis with ANSYS

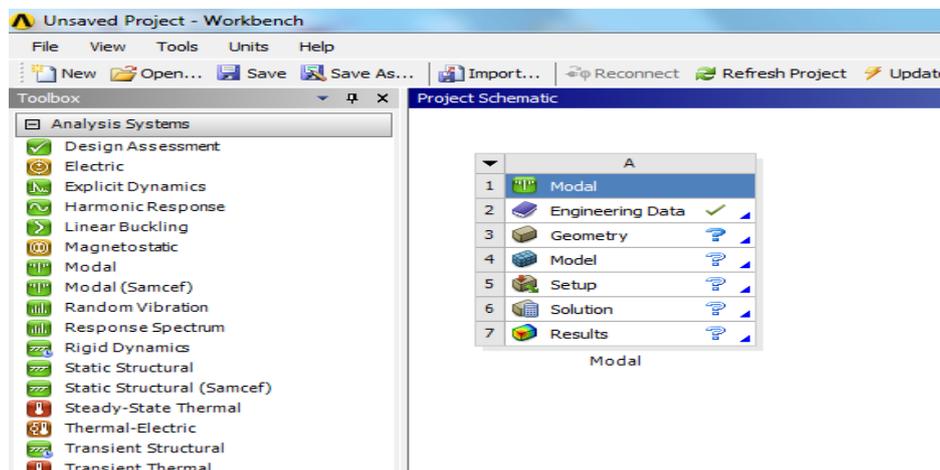The modal analysis for the cantilever beam Figure 1. is executed by ANSYS Workbench Figure 2.

Figure 2. Graphical environment of ANSYS Workbench

2.1.1. ANSYS

ANSYS is universal software, which is used on simulation of the interactions in physics structures, vibration, fluid dynamics, thermal transfer and electromechanics for engineers [7].

We can simulate with ANSYS structures and then test them in the virtual environment.

ANSYS can import CAD data and sketch of the geometry. ANSYS Workbench is a platform, which integrates simulation technologies and parametric CAD systems with unique automation and performance [7].

2.1.2. Boundary Condition

The material properties are assigned to the beam and boundary conditions are defined. The beam's all degrees of freedom on surface are taken. They are denoted with the blue flag Figure 2. This condition prevents the movement of the surface in a space.

Figure 3. Boundary conditions

2.1.3 Mesh of Finite Elements

Mesh on the beam is generated automatically by ANSYS, while is used the spatial element SOLID187 Figure 4. The element is defined by 10 nodes while each node has three degrees of freedom. The SOLID187 has a quadratic shifting behavior and is suitable for modeling of the finite element irregular mesh.
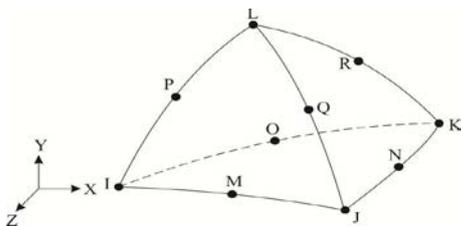

Figure 4. Spatial element SOLID187

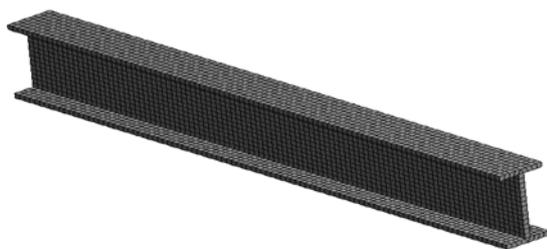The maximum size of the element is 5 mm. The mesh in Figure 5. is created of 3200 elements and of 22459 nodes.

Figure 5. Mesh of finite elements on beam

2.1.4. Analysis of Cantilever Beam

We can influence computational time of the analysis, when a range of frequencies or number of mode shapes is specified. The type of solver and the solution method in program ANSYS is selected automatically. For this modal analysis the direct solver including the block Lanczos method is used. The first five mode shapes are shown in Figure 6, Figure 7, Figure 8, Figure 9, Figure 10 and the first five natural frequencies are shown in Table 1.

Table 1. Natural frequency of particular mode shapes

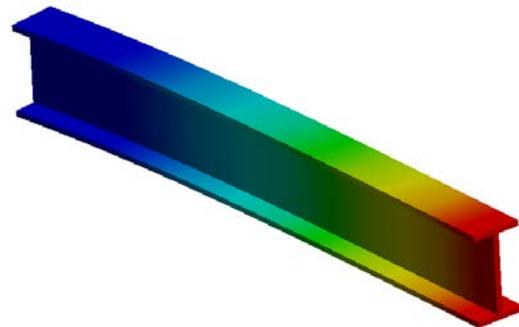| Mode shape	1.	2.	3.	4.	5.
Frequency (Hz)	94.02	207.65	324.32	570.48	824.08

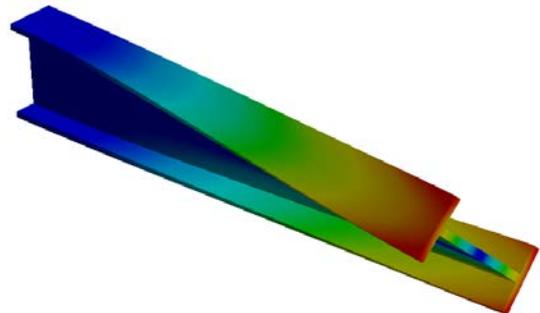**Figure 6.** The first mode shape at natural frequency 94.02 Hz

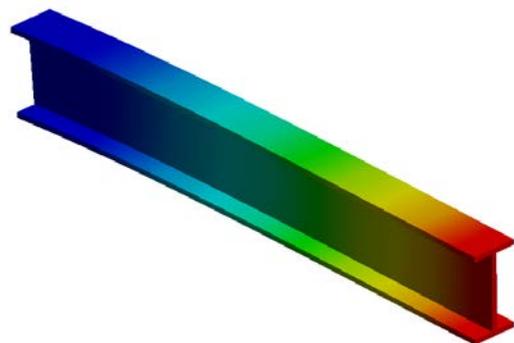**Figure 7.** The second mode shape at natural frequency 207.65 Hz

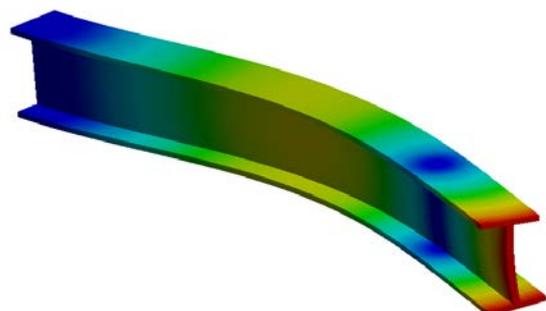**Figure 8.** The third mode shape at natural frequency 324.32 Hz

Figure 9. The fourth mode shape at natural frequency 570.48 Hz
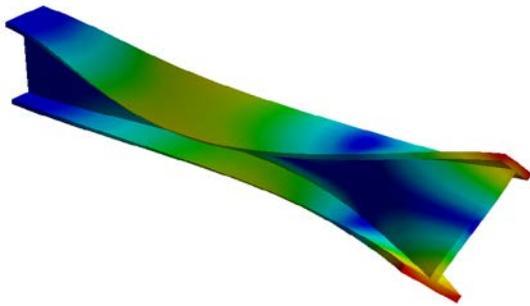
Figure 10. The fifth mode shape at natural frequency 824.08 Hz

2.2. Modal Analysis with SolidWorks

The modal analysis for the cantilever beam Figure 1 is executed by SolidWorks Figure 11.

2.2.1 SolidWorks

The SolidWorks is engineering 3D CAD software for Microsoft Windows. It has three degrees of function for the needs of organizations. The SolidWorks Standard is suitable for fast modeling, design in 2D and in 3D. The SolidWorks Professional is the superstructure of SolidWorks Standard. It improves of the efficiency and innovating with solutions that are used by millions of designers. It contains additional extension modules as Animator, PhotoWorks, etc. The SolidWorks Premium is the most comprehensive software. It combines the capabilities of the SolidWorks Professional with simulations. The Part of the SolidWorks Professional is the SolidWorks Simulation, which provides basic simulation tools for testing stress, strain, analyzing the kinematics, dynamics and it simulations conditions of the real world [9].
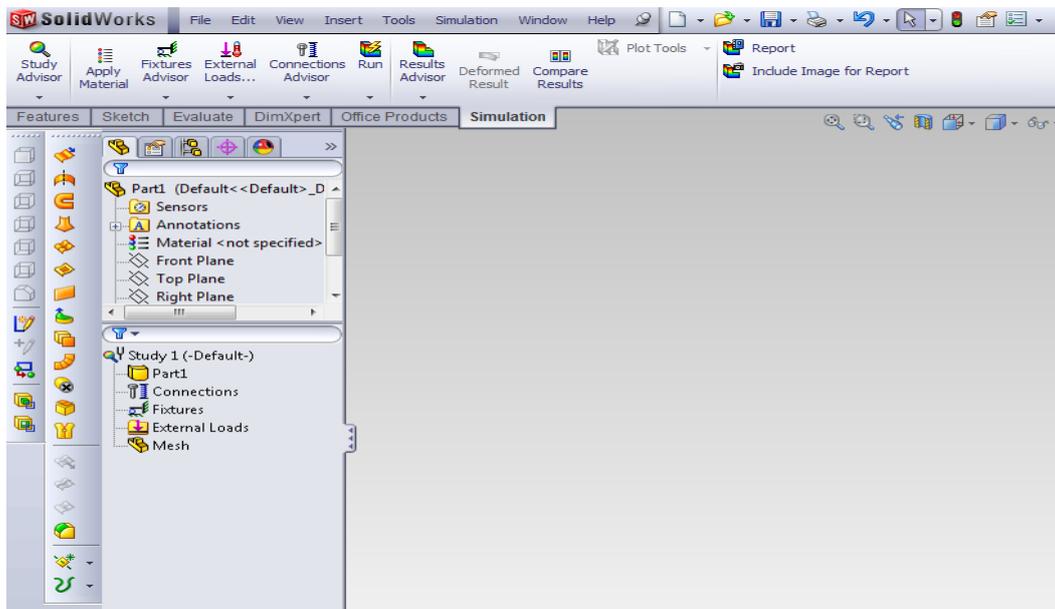

Figure 11. Graphical environment of SolidWorks

2.2.2. Boundary Condition

The material properties are assigned to the beam and boundary conditions are defined. The beam's all degrees of freedom on surface are taken. They are denoted with the blue flag Figure 12. This condition prevents the movement of the surface in a space.
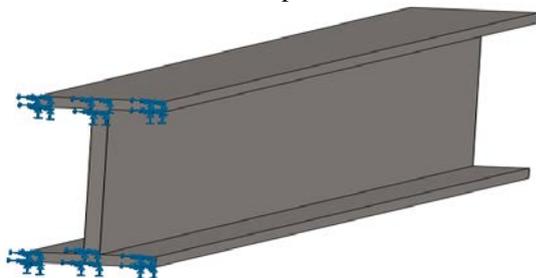

Figure 12. Boundary condition

2.2.3. Mesh of Finite Elements

Mesh on the beam is generated automatically by SolidWorks, while is used the spatial element SOLID187 Figure 3. The element is defined by 10 nodes while each node has three degrees of freedom at each node. The SOLID187 has a quadratic shifting behavior and is suitable for modeling of the finite element irregular mesh.

The maximum size of the element is 5 mm. The mesh in Figure 13 is created of 31358 elements and of 54977 nodes.

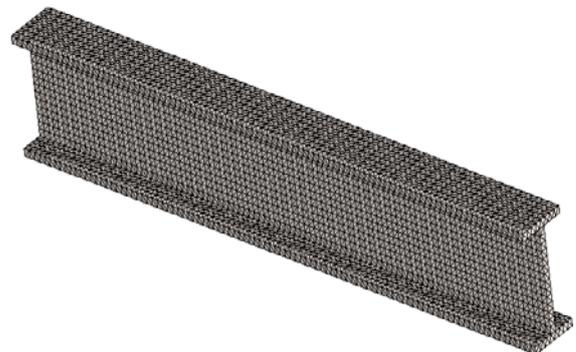**Figure 13.** Mesh of finite elements on beam

2.2.4. Analysis of Cantilever Beam

The modal analysis is carried out by SolidWorks and mode shapes and natural frequencies are also calculated. For this modal analysis the direct solver including the block Lanczos method is used. The first five mode shapes are shown in Figure 14, Figure 15, Figure 16, Figure 17 Figure 18 and the first five natural frequencies are shown in Table 2.

Table 2. Natural frequency of particular mode shapes

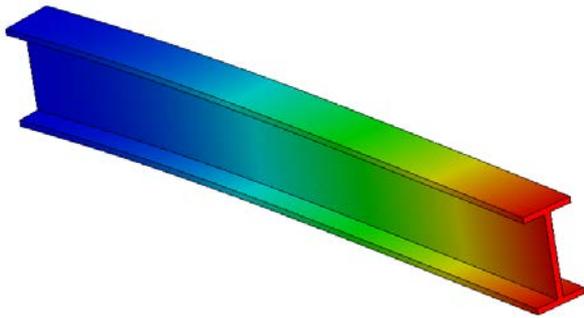| Mode shape	1.	2.	3.	4.	5.
Frequency (Hz)	94.02	207.58	324.32	570.47	823.64

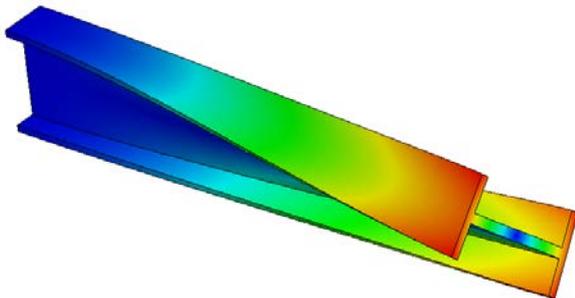**Figure 14.** The first mode shape at natural frequency 94.02 Hz

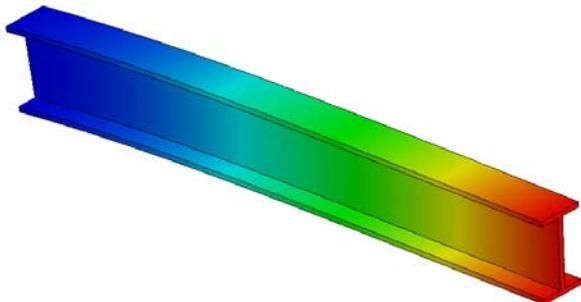**Figure 15.** The second mode shape at natural frequency 207.58 Hz

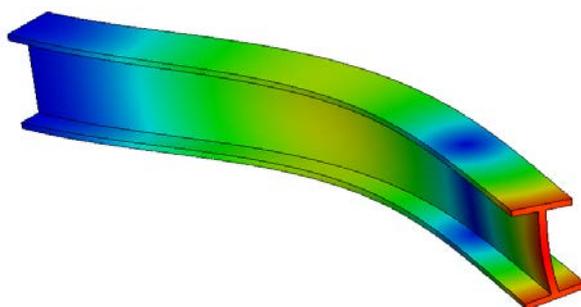**Figure 16.** The third mode shape at natural frequency 324.32 Hz

Figure 17. The forth mode shape at natural frequency 570.47 Hz
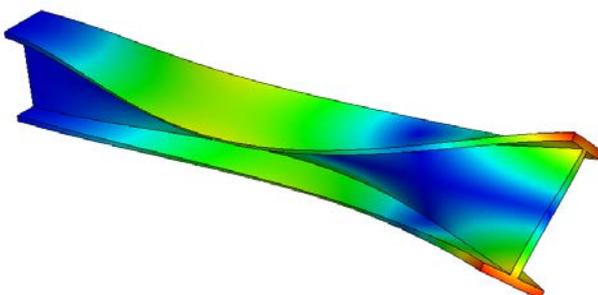

Figure 18. The fifth mode shape at natural frequency 823.64 Hz

3. Conclusion

The model of the titan cantilever beam was modeled in the programs ANSYS and SolidWokrs. In both programs the equal mesh (SOLID187) was used, whereas in the ANSYS it was automatically generated regular in the SolidWorks it was automatically generated irregular. It has resulted in a larger number of elements and nodes and longer and more complex calculation. Then the modal analysis of the titan cantilever beam was executed and the first five mode shapes and their natural frequencies were computed. Mode shapes of the titan cantilever beam are identical for both programs. Natural frequencies are almost the same for both programs. Numerical deviations of natural frequencies were created by different arrangement of the finite element mesh, which was automatically generated by the program. The program ANSYS has better options of finite element mesh creating. In Figure 19 are shown details and modifications of finite element mesh, which was generated in the program ANSYS. In Figure 20 are shown possibilities of control and creation of finite element mesh.

Figure 19. Details of mesh

Figure 20. Mesh control

Acknowledgement

References

[1] EWINS, D.J., *Modal Testing: Theory, Practice and Application.* England: Wiley, Second Edition. 2000.

[2] LENGVARSKÝ. P., *Štrukturálna a modálna analýza súčiastok z polymérov používaných v domácich spotrebičoch,* Košice, 2000.

[3] TREBUŇA, F. - ŠIMČÁK, F. *Príručka experimentálnej mechaniky*, SjF TU, Košice, 2007.

[4] ŽIARAN, S., *Kmitanie a akustika. Ochrana človeka pred kmitaním a hlukom,* STU Bratislava., 2001.

[5] KOLOUŠEK, V., *Dynamics in Engineering Structures*. Praha: Academia, 1973.

[6] MEAD, D., *Passive Vibration Control*. University of Southampton, UK: Wiley, 2000.

[7] ANSYS: *Products,* [online], <http://ansys.com/Products>.

[8] SolidWorks: *Products,* [online], <http://www.solidworks.com/-sw/products/10141_ENU_HTML.htm>.

[9] SolidWorks *Simulation,* [online], <http://www.solidworks.com/-sw/products/10169_ENU_HTML.htm>.