Tungsten Cantilever Beam Using Ansys (Modal Analysis)

[1,] Mr.k.vijaykumar (asst.prof), [2,] Mr.G.shivaraju (asst.prof), [3,] Mr.U.sreekanth (asst.prof), [4,] Mr.aluri david (asst.prof),

[1, 2, 3,] *Department of mechanical engineering, Kasireddy Narayan Reddy College of Engineering & Research (KNRCER), Hyderabad, Andhra Pradesh, India*
[4,] *Department of mechanical engineering, Nalla Malla Reddy Engineering College (NMREC), Hyderabad, Andhra Pradesh, India*

---ABSTRACT--
In this paper are given the modal analysis simulation in particular block Lanczos method. The modal analysis is carried out on the tungsten cantilever beam. The cantilever beam is designed in the graphical environment of the ANSYS and Solid Works. 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 Solid Works with numerical formulation of the direct solver including the block Lanczos method.

KEYWORDS: cantilever beam, ANSYS, Solid Works.

I. 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 measurements 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].

II. CANTILEVER BEAM

In Figure 1 is shown design of the cantilever beam with its dimensions. It is made of Tungsten (w) with material properties E=4.05e5 Pa, μ=0.28, ρ=19.3 kg/m
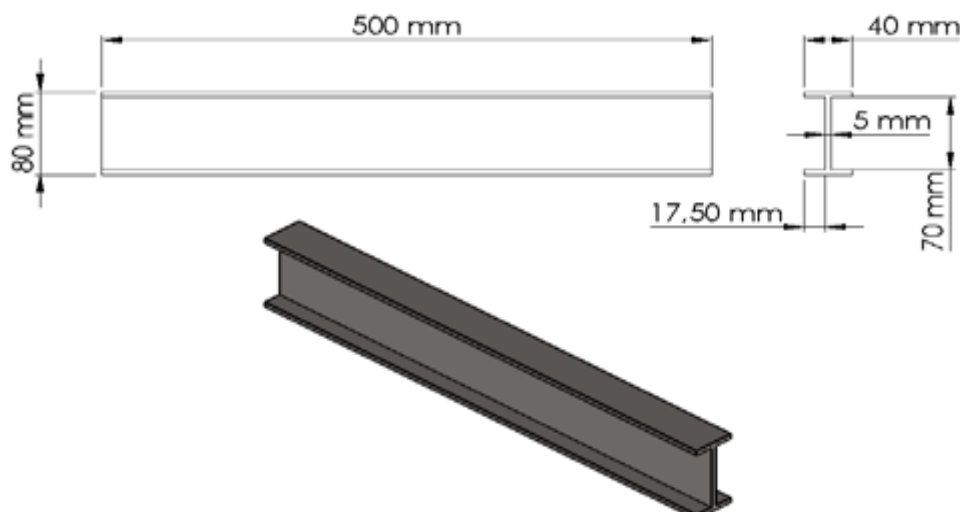

Figure1. Design of cantilever beam

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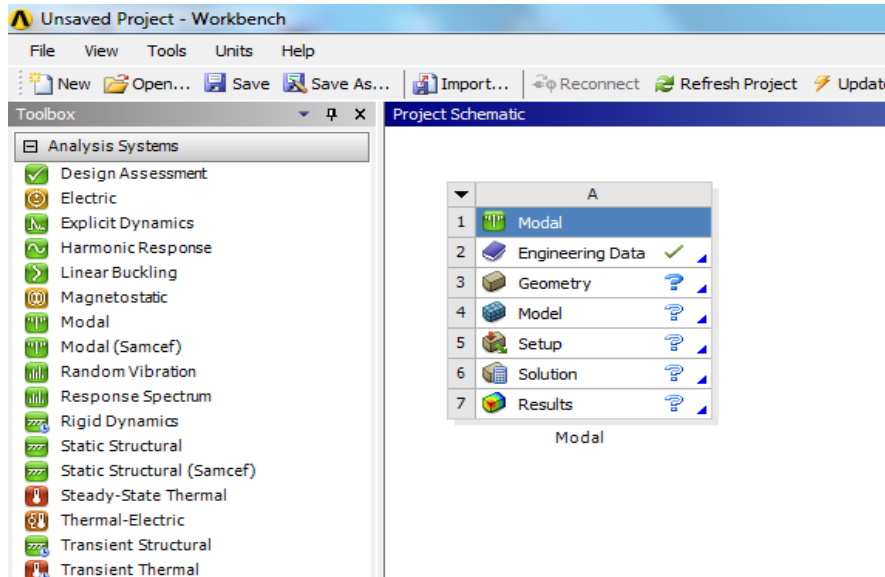The modal analysis for the cantilever beam Figure 1. is executed by ANSYS Workbench Figure 2.

Figure2. 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 electro mechanics 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.
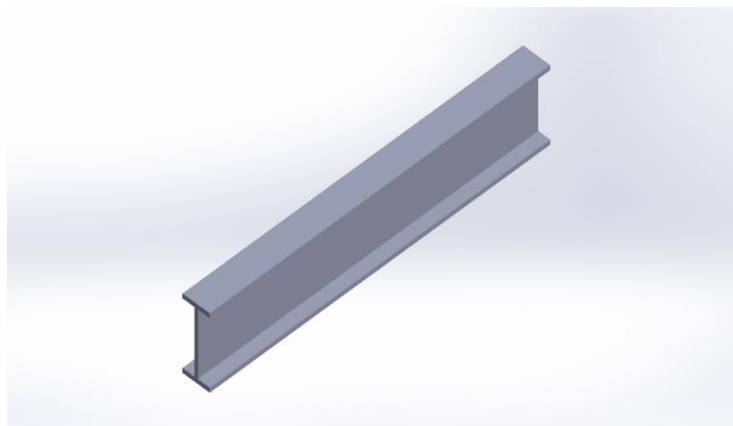

Figure3. Boundary conditions

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.
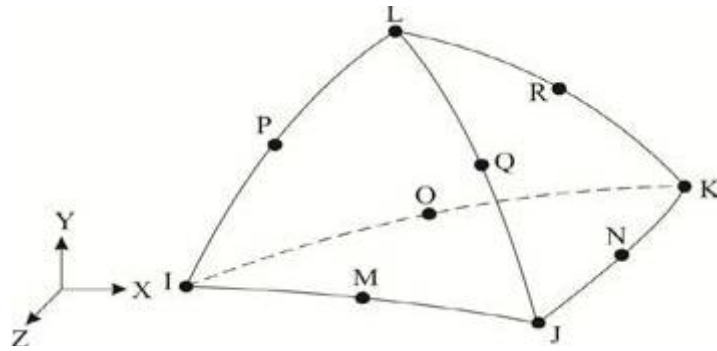
Figure4. Spatial element SOLID187

The maximum size of the element is 5 mm. The mesh in Figure5. Is created of 3200 elements and of 22459 nodes.
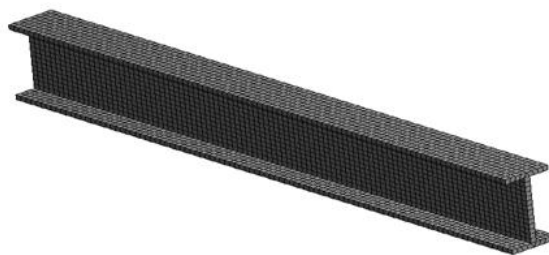

Figure5. Mesh of finite elements on beam

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.

Mode shape	1	2	3	4	5
Frequency(hz)	94.02	201.65	344.32	575.478	804.08

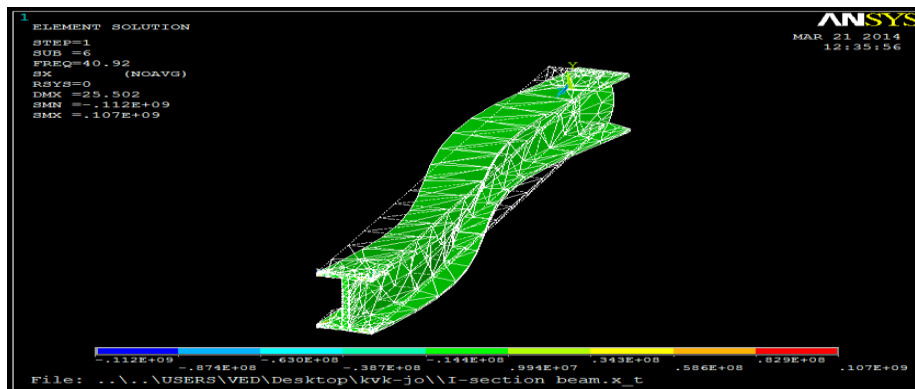**Table 1**. **Natural frequency of particular mode shapes**

Figure6. The first mode shape at natural frequency 94.02 Hz
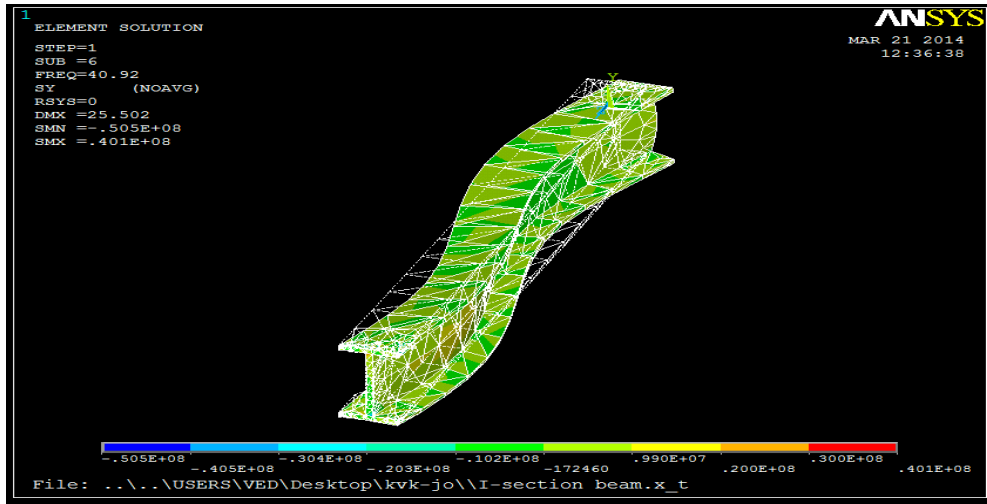
Figure7. The second mode shape at natural frequency 201.65 Hz
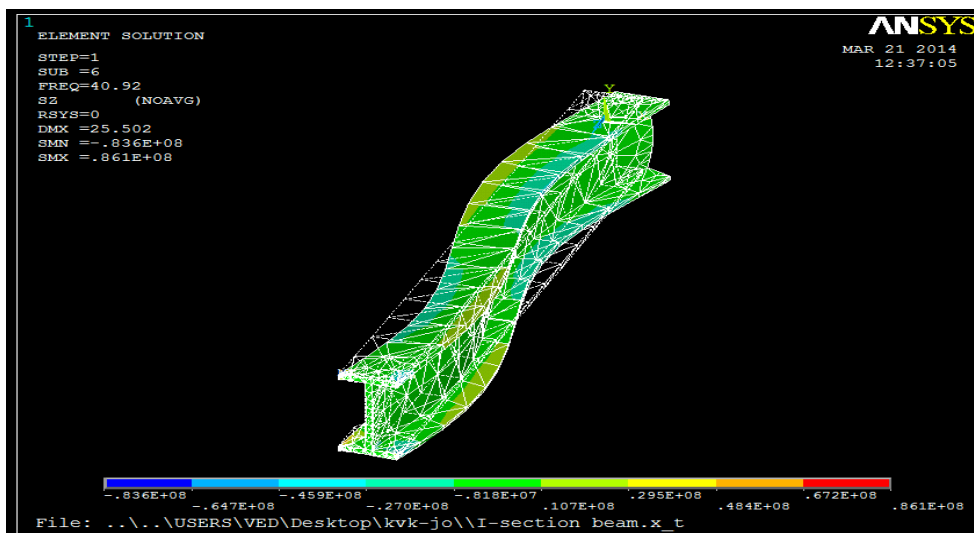

Figure8. The third mode shape at natural frequency 344.32 Hz
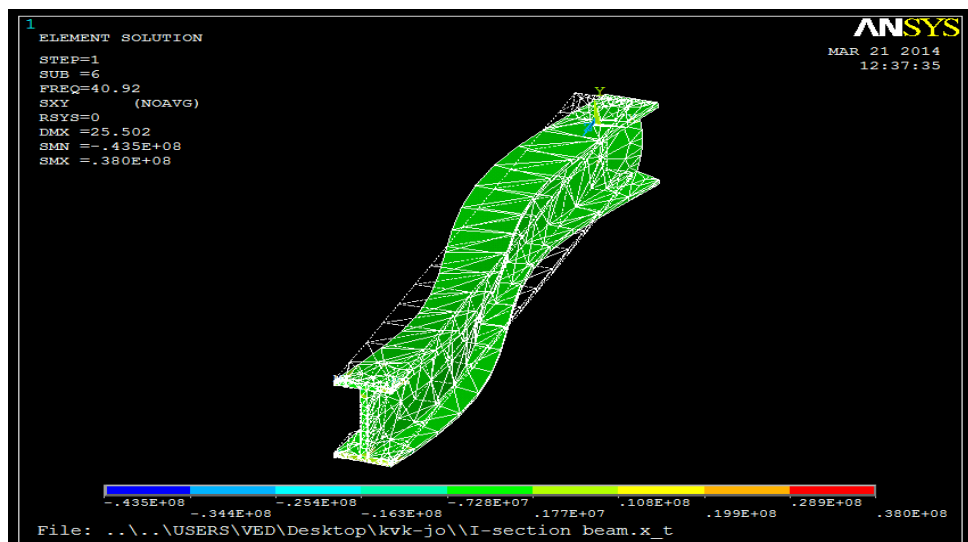

Figure9. The fourth mode shape at natural frequency 575.478 Hz

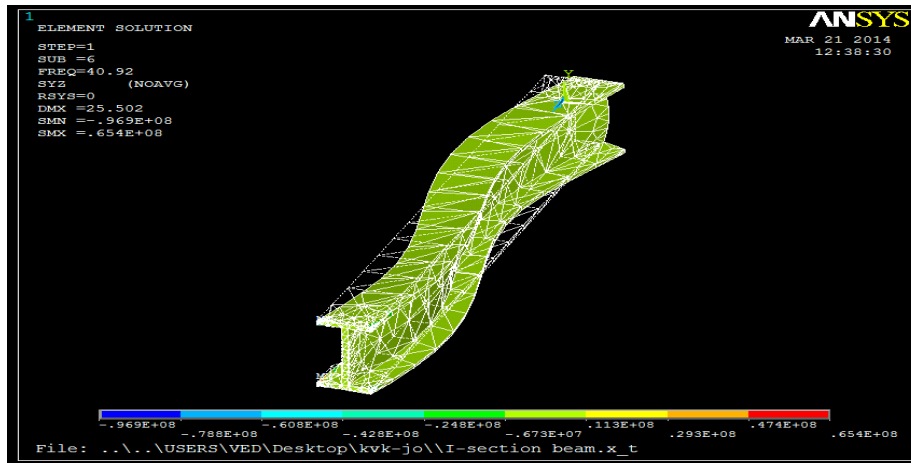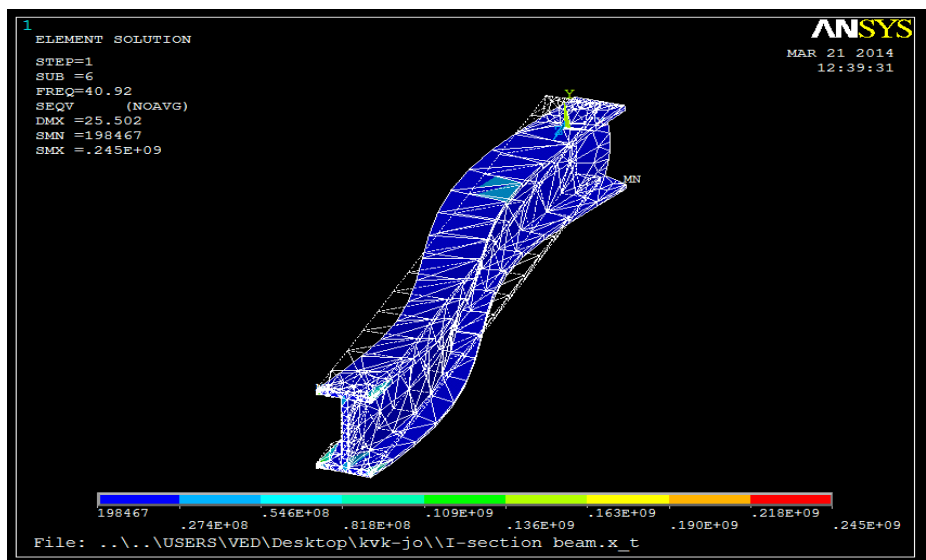**Figure10.** The fifth mode shape at natural frequency 804.68 Hz

Figure11. Von mises stress

2.2. Modal Analysis with Solid Works

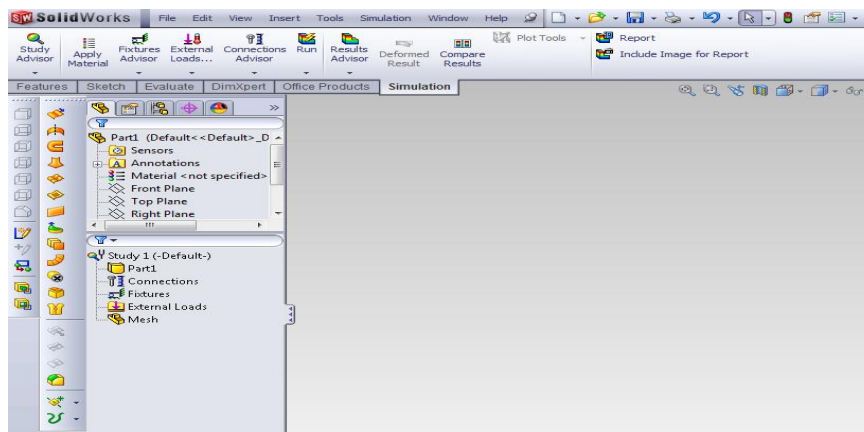The modal analysis for the cantilever beam Figure 1 is executed by Solid Works Figure 11.

Figure12. solid works

2.2.1 Solid Works

The Solid Works is engineering 3D CAD software for Microsoft Windows. It has three degrees of function for the needs of organizations. The Solid Works Standard is suitable for fast modeling, design in 2D and in 3D. The Solid Works Professional is the superstructure of Solid Works Standard. It improves of the efficiency and innovating with solutions that are used by millions of designers. It contains additional extension modules as Animator, Photo Works, etc. The Solid Works Premium is the most comprehensive software. It combines the capabilities of the Solid Works Professional with simulations. The Part of the Solid Works Professional is the Solid Works Simulation, which provides basic simulation tools for testing stress, strain, analyzing the kinematics, dynamics and it simulations conditions of the real world [9]. The maximum size of the element is 5 mm. The mesh in Figure 13 is created of 31358 elements and of 54977 nodes.
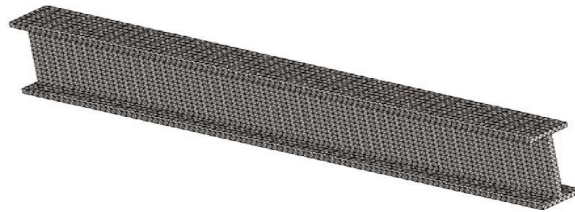

Figure13. Mesh of finite elements on beam

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
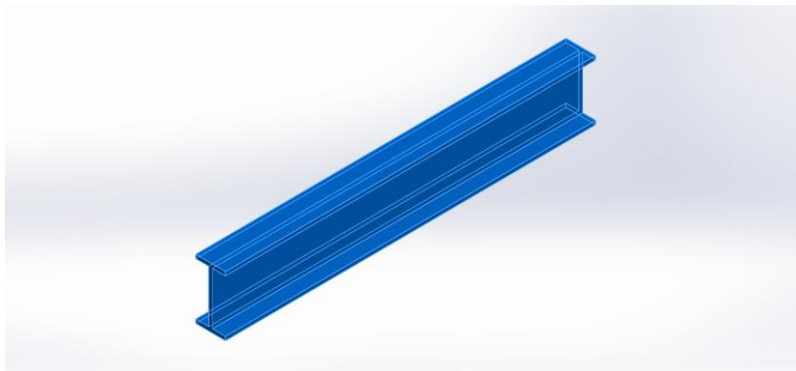

Figure14. Boundary condition

2.2.3.Mesh of Finite Elements

Mesh on the beam is generated automatically by Solid Works, 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

2.2.4. Analysis of Cantilever Beam

The modal analysis is carried out by Solid Works 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.

Table2. Natural frequency of particular mode shapes

Mode shape	1	2	3	4	5
Frequency(hz)	94.02	201.65	344.32	575.478	804.08

III. FACTOR OF SAFETY

Design factor and safety factor

The difference between the safety factor and design factor (design safety factor) is as follows: The safety factor is how much the designed part actually will be able to withstand (first "use" from above). The design factor is what the item is required to be able to withstand (second "use"). The design factor is defined for an application (generally provided in advance and often set by regulatory code or policy) and is not an actual calculation, the safety factor is a ratio of maximum strength to intended load for the actual item that was designed.

$$\text{Factor of Safety} = \frac{\text{Material Strength}}{\text{Design Load}}$$

- Design load being the maximum load the part should ever see in service.

By this definition, a structure with a FOS of exactly 1 will support only the design load and no more. Any additional load will cause the structure to fail. A structure with a FOS of 2 will fail at twice the design load

$$\text{Factor of Safety} = \frac{\text{Material Strength}}{\text{Design Load}}$$

Table 3. Factor of safety

stress	Yield strength of steel (N/MM^2)	Obtained max stress with 300 N of the load (N/MM^2)	Factor of safety	Min stress (N/MM^2)	Max stress (N/MM^2)
X-component of stress	940	107	8.785	-112	107
Y-component of stress	940	40.1	23.44	-50.5	40.1
Z-component of stress	940	86.1	10.917	-83.6	86.1
XY-shear stress	940	38	24.736	-43.5	38
YZ -shear stress	940	65.4	14.373	-96.9	65.4
XZ -shear stress	940	32.8	28.658	-120	32.8
Von mises stress	940	245	3.836	198467	245

IV. CONCLUSION

The model of the Tungsten cantilever beam was modeled in the programs ANSYS and SolidWokrs12. In both programs the equal mesh (SOLID187) was used, whereas in the ANSYS it was automatically generated regular in the SolidWorks12 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 Tungsten cantilever beam was executed and the first five mode shapes and their natural frequencies were computed. Mode shapes of the Tungsten 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 5 are shown details and modifications of finite element mesh, which was generated in the program ANSYS. In Figure 11 are shown possibilities of control and creation of finite element mesh. Factor of safety 3.836 along von mises stress so breaking chances will be their applying of displacement place.

V. 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
[10]. Robert D. Cook, *Concepts* and Applications of Finite Element Analysis, John Wiley and Sons, Inc.,200