ISSN : 2350-0743

www.ijramr.com

International Journal of Recent Advances in Multidisciplinary Research

Vol. 06, Issue 04, pp.4806-4809, April, 2019

RESEARCH ARTICLE

STRENGTH AND MODAL ANALYSIS OF AVIATION TURBINE BASED ON ANSYS

*Yongyue Yuan

Binzhou University, Binzhou, China

ARTICLE INFO

Article History:

Received 20th January, 2019
Received in revised form
26th February, 2019
Accepted 24th March, 2019
Published online 30th April, 2019

Keywords:

Turbine; Strength; ANSYS; Modal.

ABSTRACT

Turbine is an important component of aero-engine. Once a fault occurs, especially when the load exceeds the yield strength, the turbocharger will be damaged in a very short time. Therefore, the structural reliability analysis of turbocharger becomes more and more important. A simplified turbine model is established by Catia. The static strength of the model is analyzed based on ANSYS Workbench finite element analysis software. The distribution of stress and deformation is obtained, and the relationship between rotational speed and maximum stress is studied. In addition, the modal analysis of the model can obtain the natural frequencies and modes, which can provide the basis for the optimization design and dynamic analysis.

INTRODUCTION

Turbocharging is a technology of driving air compressor by using exhaust gas generated by the operation of internal combustion engine. It has good application effect in the field of aero engine. Turbine is the most important mechanical component, and its mechanical properties are very important for the safe operation of the whole engine equipment. For solving this problem, the finite element method is a good means. The finite element method divides the continuous solution domain into a group of elements. The approximation function assumed in each element is used to represent the unknown field function in the solution domain. The approximation function is usually expressed by the numerical interpolation function of the unknown field function and its derivatives at each node of the element. Thus a continuous infinite degree of freedom problem becomes a discrete finite degree of freedom problem. Finite element method can analyze complex structures, deal with complex boundary conditions, ensure specified engineering accuracy, and deal with different types of materials. As typical finite element analysis software, ANSYS Workbench has a bidirectional connection with the solid and surface model of CAD system. Its high success rate of importing CAD geometric model can greatly reduce the error removal time and shorten the design and analysis. In view of its superior processing ability, this paper chooses it as a tool to deal with problems.

Static Strength Analysis

Analytical tasks and requirements: The purpose of static analysis is to check whether the bearing capacity of the structure meets the requirements of strength design. If the strength is excessive, the size of structural bearing parts can be reduced. For structures with cracks, due to the singular stress distribution at the crack tip, the conventional static strength analysis method is no longer applicable, and it is a fatigue and fracture problem. Finite element analysis can verify whether the ability of structure to resist deformation meets the requirements of strength design, and provide structural stiffness characteristics data for dynamic analysis, which is usually carried out under the use of loads or smaller loads. Calculate and check whether members, plates, thin-walled structures and shells will lose stability under load. The aeroelastic mechanics method is used to study the stability of structures with aerodynamic and elastic coupling forces. In addition, it can also calculate and analyze the distribution of stress and deformation and buckling modes of structures under static loads, providing information for other aspects of structural analysis.

Establishment and pretreatment of finite element model: Catia is used to draw turbine model. The three-dimensional model established by Catia is generated into files conforming to IGES standard and imported into ANSYS Workbench. Firstly, the material is defined, then the three-dimensional model is imported into Workbench, and mesh calculation parameters are defined by Mesh software, then the boundary conditions and loads are defined. Finally, the solution calculation is carried out, and the results can be obtained and analyzed. For the definition of material attributes, it is necessary to double-click Engineering Data to enter the material library, select General Materials, and select aluminum alloy as turbine material. Then, add Aluminum Alloy under Material. After the material definition is completed, return to Project and double-click Model to enter the Mechanical interface after importing the model. Under Material, select the material just defined in Assignment. The grid has a large workload and many problems to be considered. The form of the grid directly affects the accuracy of the results and the scale of the model. Therefore, mesh generation is the most critical link in the process of modeling. The type of analysis data should be considered when deciding the number of grids.
In static analysis, if only the deformation of the structure is calculated, the number of meshes can be less.

If the stress needs to be calculated, a relatively large number of grids should be taken under the same accuracy requirements. Similarly, in response calculation, the number of meshes for calculating stress response should be more than that for calculating displacement response. When calculating the inherent dynamic characteristics of structures, if only a few low-order modes are calculated, fewer grids can be selected, and if the calculated modal order is higher, more grids should be selected. In thermal analysis, the temperature gradient inside the structure is not large, and a large number of internal elements are not needed. At this time, fewer grids can be divided. The meshes with different densities are mainly used for stress analysis, while the uniform steel lattice is used for calculating the inherent characteristics.

This is because the natural frequencies and modes depend mainly on the mass and stiffness distribution of the structure, and there is no similar phenomenon of stress concentration. The element difference between the stiffness matrix and the mass matrix of the structure cannot be too large by using uniform grid, and the numerical calculation error can be reduced. After comparing the calculation accuracy and efficiency, Element Size is chosen as 2 mm, the number of meshes is 124807, and the number of nodes is 195263. The adaptive network division method is selected, and the mesh is shown in Fig.1. The load includes boundary conditions and excitation. Turbocharger works in a very harsh environment in practical work, and it acts together with aerodynamic load and centrifugal force. But the aerodynamic force is smaller than the centrifugal force, so the aerodynamic force can be neglected and only the centrifugal force can be considered. Select Static Structural, define the boundary conditions under Supports, and select Fixed Support to select the front and rear end surfaces of turbine shaft. Rotational Velocity is selected under Inertial to analyze and compare the stress state under different rotational speeds, as shown in Fig.2.

RESULTS ANALYSIS

The stress nephogram is shown in Fig.3. It can be known that the position where the maximum and minimum stresses occur is basically unchanged at different rotational speeds. On the axis, the maximum stresses occur at the bottom of the turbine central hole. The reason is that the centrifugal force produced by high-speed rotation generates tension stress in the turbine. There are many materials at the bottom of the turbine. The center of mass is close to the edge of the turbine, and the inertia is relatively large. Therefore, the centrifugal force formed outward is relatively large, which affects the formation of internal high stress zone. On the contrary, the upper part is turbine blade with small inertia and stress. The minimum stress occurs at the tip of the large blade. The material near the axis is pulled by the external centrifugal force. The more material radiates to the edge of the turbine, the closer the center of mass is to the rim, the greater the stress in this area. The maximum stress corresponding to different rotational speeds is shown in Fig. 4. It can be concluded that with the increase of turbine speed, the maximum stress increases in an approximate parabolic law. The deformation nephogram is shown in Fig.5.With the increase of centrifugal force, the blade deforms along the extension direction of radius, and the axle hole of hub extends to the direction of radius enlargement, and the axle hole tends to enlarge. The size of the turbine becomes larger.

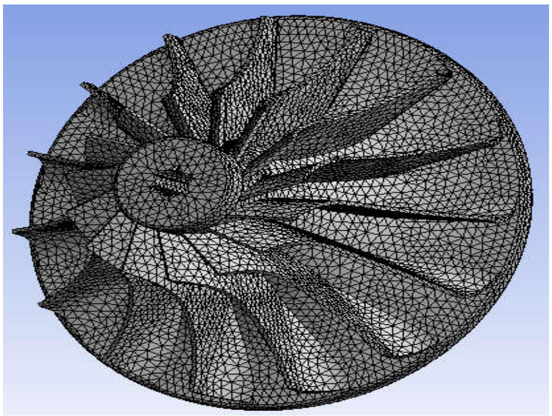

Fig.1 Mesh partition graph

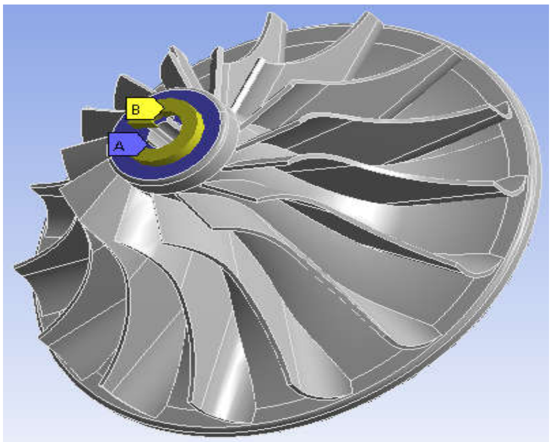

Fig.2 Load and boundary condition

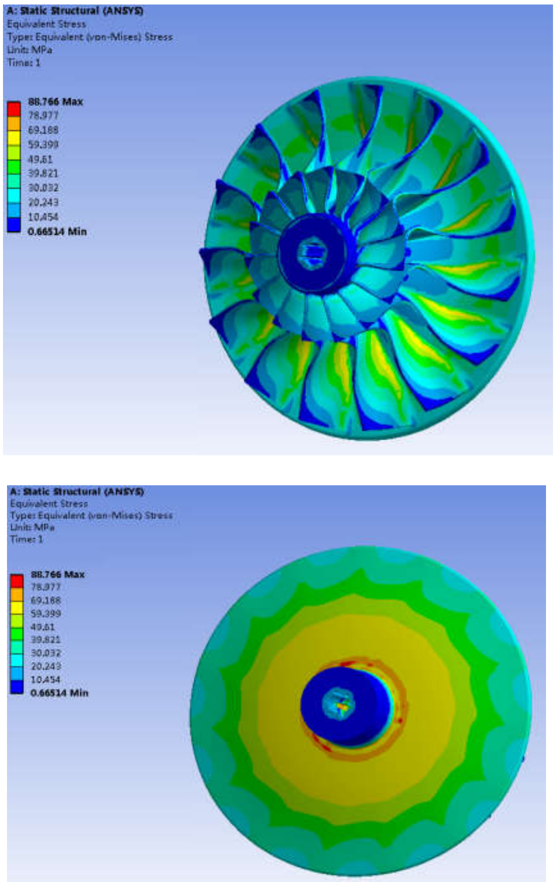

Fig.3. Stress nephogram (5000r/min)

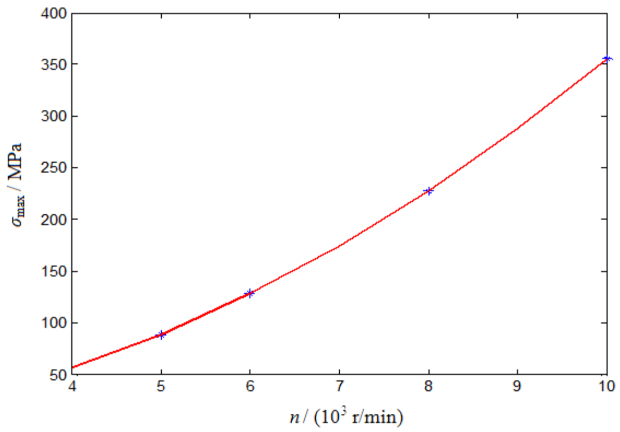

Fig.4 The maximum stress corresponding to different rotational speeds

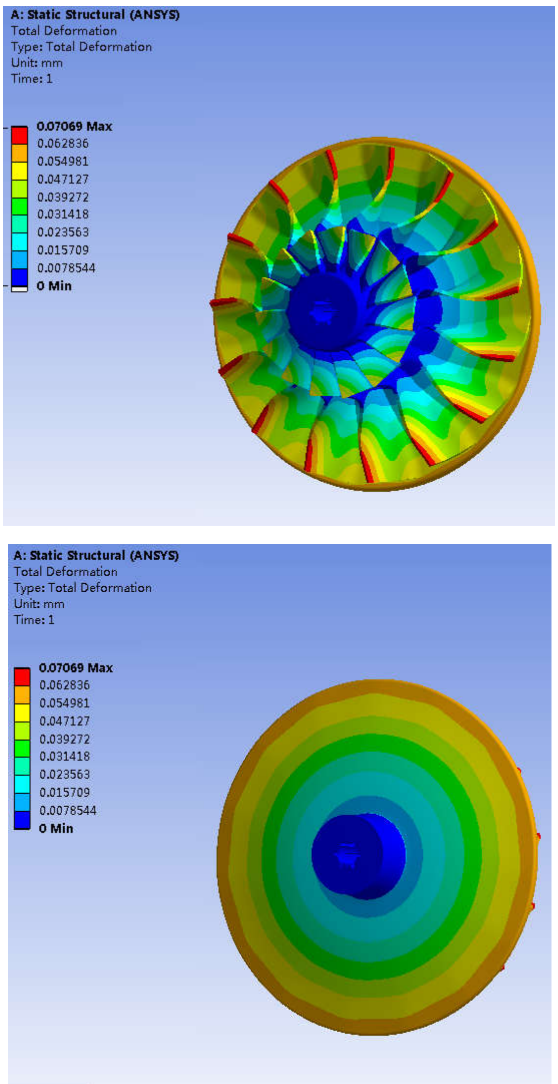

Fig.5 Deformation nephogram (6000r/min)

Table.1 Calculation results of natural frequencies

Order number	1	2	3	4	5
Natural frequency/Hz	3445	6517	6523	6902	7123

Observing the deformation from the bottom of the turbine, we can see that the edge of the turbine expands outward. Because the turbine edge is affected by the blade, the deformation is limited, resulting in wavy deformation.

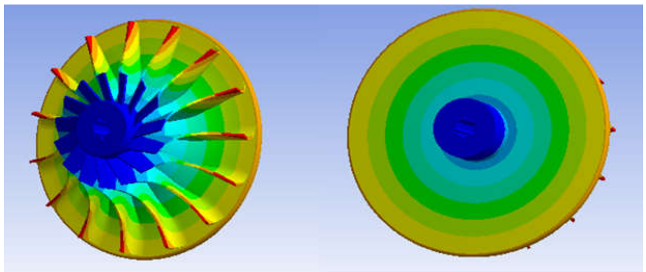

Fig.6. The first order mode diagram

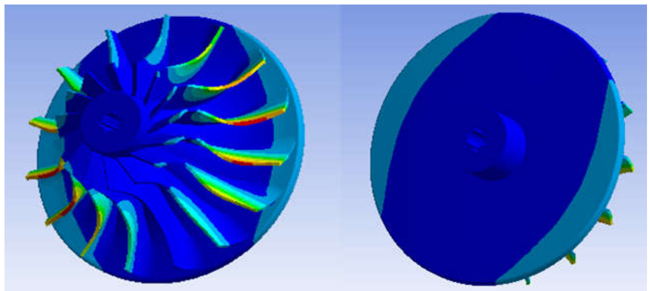

Fig.7. The second order mode diagram

Modal Analysis

Relationship between vibration and modal: Free vibration refers to the simple harmonic vibration of a vibrating object under the action of no alternating external force. The object is not subject to the continuous action of the outside world, and only depends on the elastic restoring force and the inertial force of mass to maintain the vibration. But the vibration is stimulated by external force, and the energy of vibration is given by the external force of accident. The frequency of free vibration is the natural frequency, which is only related to the physical parameters of the system. The physical parameters of a single-degree-of-freedom system are the mass and stiffness of the system. For multi-degree-of-freedom systems, the related physical parameters are the mass matrix and stiffness matrix of the system, which are the boundary conditions, geometric conditions and material properties of the system.

Modal is the inherent vibration characteristic of structure. Each mode has a specific natural frequency, damping ratio and mode shape. These modal parameters can be obtained by computational or experimental analysis, such a process of calculation or experimental analysis is called modal analysis. If the analysis process is obtained by finite element method, it is called computational modal analysis. Vibration modes are inherent and integral characteristics of elastic structures. If the characteristics of the main modes of the structure in a susceptible frequency range are clarified by modal analysis method, it is possible to predict the actual vibration response of the structure under the action of various sources inside or outside the frequency band.

Model setup and analysis: In this paper, modal analysis is carried out at 6000 r/min speed. First select Static Structural in Toolbox, and then select Modal to complete the connection on Solution. Then, double-click Engineering Data to define the material, select the commonly used material aluminum alloy, and then return to Project. Double-click Model to enter the Mechanical interface, and the other steps are similar to static structure strength analysis. Through continuous iteration, the natural frequencies under different order conditions can be obtained as shown in Table 1.

It can be seen that the first and second natural frequencies are the most representative. With the increase of order, the difference of natural frequencies is not large. The first and second modes of the model are shown in Figs. 6 and 7, respectively. The first and second modes of the model are shown in Figs. 6 and 7, respectively. It can be seen that the position of the largest resonance displacement is at the top of the blade, and the stiffness of the central structure is larger, which satisfies the condition of high speed.

Acknowledgement

The paper is supported by the Youth Talent Innovation Project (BZXYG1806).

Conclusion

Considering the centrifugal force, the structural strength calculation and vibration modal analysis of aviation turbine are carried out based on ANSYS Workbench software. Through the static strength calculation based on the finite element method, the maximum stress position and distribution state are obtained. According to the different rotational speed, the internal relationship between maximum stress and rotational speed is obtained. Strength analysis of static results can provide reference for design. The resonance characteristics of the structure are predicted by modal vibration calculation.

REFERENCES

Chen, X.L. 2013. Finite element static analysis of derrick and base system of deep well drilling rig. *Petroleum Machinery*, 41: 19-21.

LI, Y.M. 2013. Nonlinear finite element static analysis of reinforced concrete beams. *Laboratory Research and Exploration*, 32: 48-53.

LI,Y.2013. Finite element static analysis of steel structural joints based on ABAQUS. *Architectural Structure*, 12: 487-492.

Shi, C., YANG, R.B. 2015. Finite element static analysis of excavator of peanut harvester based on UG. *Agricultural mechanization research*, 11: 18-21.

Wang, M. 2014. Finite element modal analysis and structural optimization of sprayer boom. *Journal of Agricultural Machinery*, 45: 112-116.

Yang, Li. 2016. Development of a Thermal Barrier Coating Erosion Testing Device for Turbine Blades of Aeroengine. *Equipment Environmental Engineering*, 13: 48-56.

****