RESEARCH ON TURBINE BLADE STRENGTH AND STEADY STATE RESPONSE BY FINITE ELEMENT METHOD

Jian Sun[*], Rong Tang, Wei Liu
Beijing FEAonline Engineering Co.,Ltd, China

KEYWORDS -
Turbine Blade, Strength, Steady State Response, ANSA, ABAQUS

ABSTRACT -
The strength and steady state response analysis about a type of turbine blade is analyzed in this paper based on the finite element method(FEM). In order to obtain the perfect simulation results the pre-processing was conducted in ANSA software including the geometry clean-up, the meshing, the loading, and all other solving settings. Then, the computational model was calculated in the ABAQUS software. The simulation results show an agreement between the experimental results indicating that the finite element analysis can provide a reference for turbine blade design and optimization.

1. INTRODUCTION

With the development of computer technology, FEM has attracted more and more applications in various high-technology industries due to its advantage of high-precision and low-cost. As the pre-processing is an important and time-consuming part during the simulation of a complex engineering project, ANSA software, with its high performance in preprocessing, has been already applied widely. Recently, many engineers have solved engineering problems using this software for finite element simulations. In this paper, the strength and steady state response of a type of turbine blade was analyzed using ANSA software for pre-processing and ABAQUS software for the calculations.

2. FINITE ELEMENT MODEL

2.1 Single turbine blade model

ANSA offers has high performance in meshing, quality control, geometry cleanup, and model set-up the single turbine blade model set-up was conducted in this software. The mesh refinement was performed at the transition zone between tip shroud and body. Taking the precision and cost limitations into account, the model with 70000 C3d8I most-hexahedron elements was selected for simulation. The blade rotated with 3000r/min, after fixing the flank of blade base and X direction of the fourth cog. The meshing and loading model are shown in Fig.1.

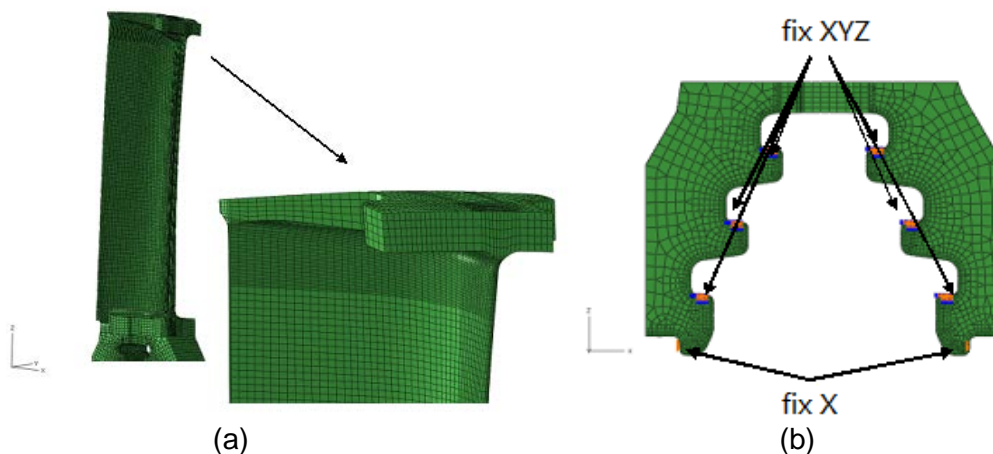(a)   (b)
Fig.1 Finite element model (a) meshing (b) contacting

2.2 Whole model

The whole model with the 100 thousand most-hexahedron elements includes the blade, the blade platform, the blade base, and the wheel. Using the ANSA software, the mesh refinement was performed at the contact region of blade base and the wheel. The interaction region between the blade base and the groove was added with the friction of 0.1, as illustrated in Fig.2.

The whole model rotated with 3000r/min, while the temperature increasing from 20°C to 80°C, after fixing the surface of wheel bore. Cyclic symmetry boundary condition was applied on the surface of one pitch of a wheel.
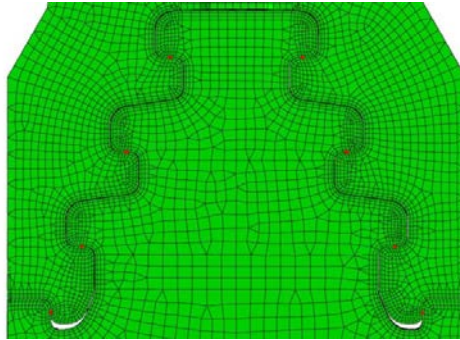

Fig.2 Interaction region

3. STRENGTH ANALYSIS

3.1 Strength of single turbine blade

After pre-processing in ANSA software, the single turbine blade model was calculated in ABAQUS software. The Stress and deformation distributions of the blade body were analyzed intensively in post-processing.

The Mises stress and the displacement distribution of the blade body are shown in fig.3.

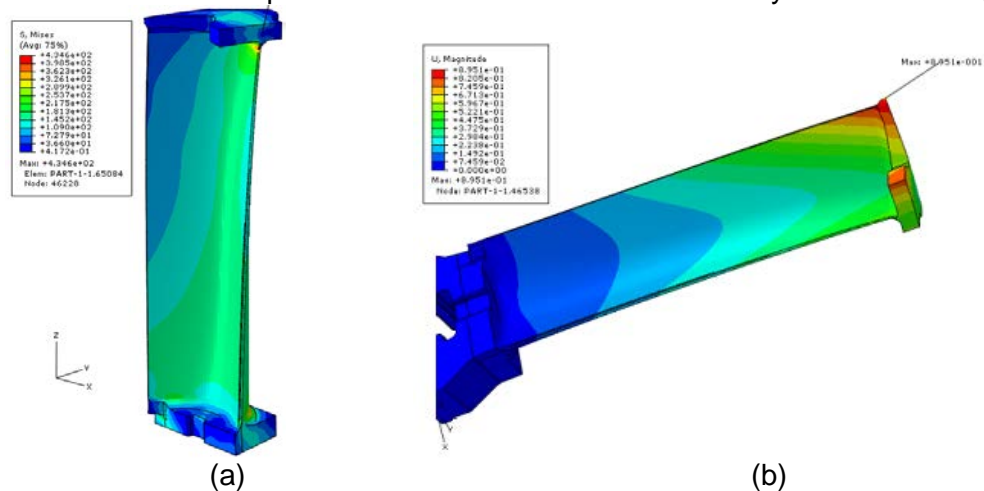(a)   (b)
Fig.3 Stress and displacement distribution (a) mises tress (b) displacement

According to the stress and displacement distribution of the working part, there was an obvious stress concentration at the transition zone between the tip shroud and the body. The radial deformation was dominant and maximum displacement value was 0.895mm.

3.2 Strength of whole model

3.2.1 Strength and displacement of whole model

The maximum mises stress was distributed at the contact region of the blade base and the wheel, and the maximum value was 755.8Mpa, as shown in fig.4.
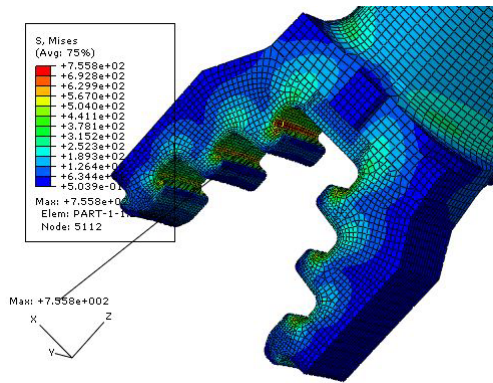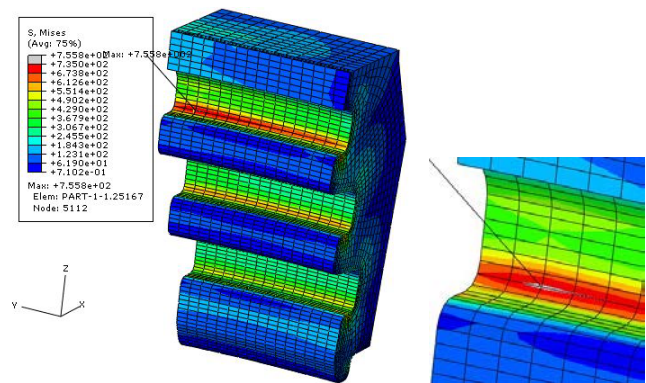
Fig.4 Mises stress distribution   Fig.5 Small scale yield distribution

According to Fig.5, when the upper limit of stress contour was 735Mpa, as the yield stress of blade material, a small scale yield was distributed at the region of blade base fillet.
The Maximum displacement value of the whole model was 0.895mm.

3.2.2 Strength and displacement of contact region

As mentioned before, a small scale yield was distributed at the region of blade base fillet. The Mises stress distribution on other side of blade base is shown in Fig.6 and the maximum value was 719.1Mpa.
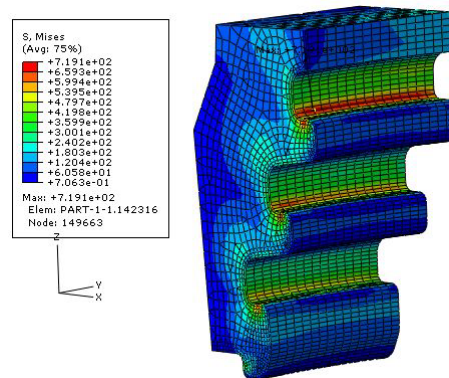

Fig.6 Mises stress distribution

Fig.7 showed the mises stress distribution of both sides of wheel, and the maximum value was 691.7MPa and 680.8MPa.
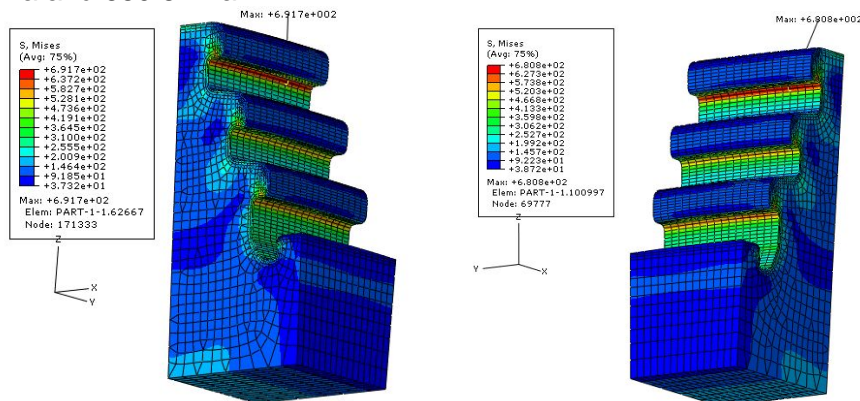

Fig.7 Mises stress distribution

Fig.8 showed the contact pressure distribution of both sides of wheel, and the maximum value was 817.1MPa and 851.9MPa.
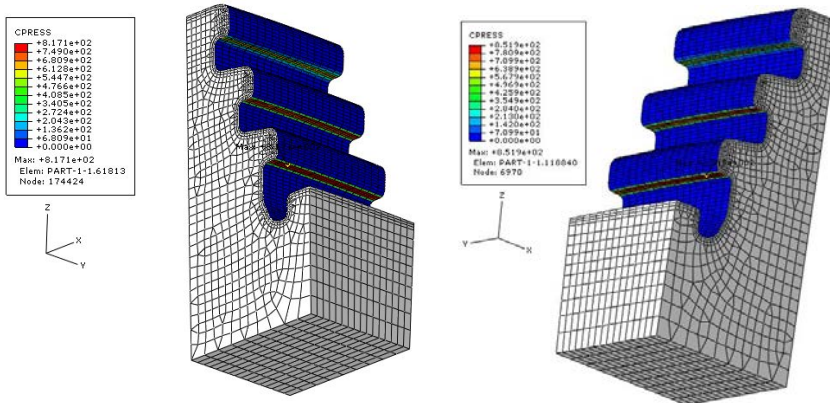
Fig.8 Contact pressure distribution

The distortion of blade base and wheel is shown in fig.9, and the maximum displacement value of the blade base was 0.895mm while that of the wheel was 0.8948mm.
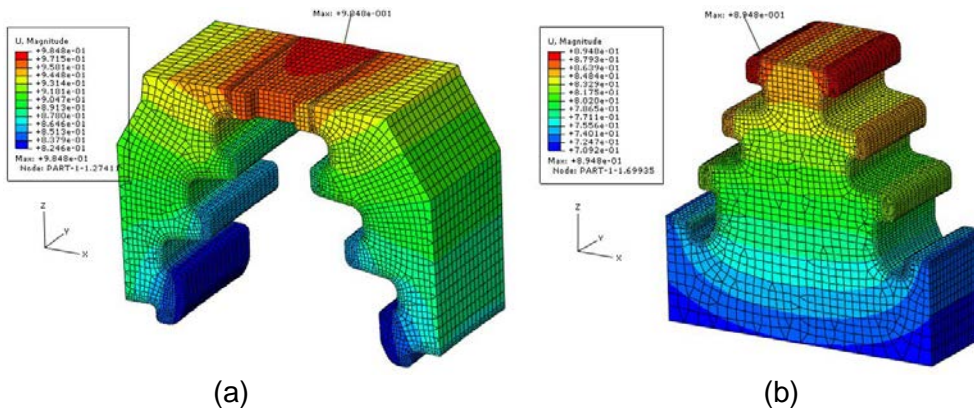

(a)   (b)
Fig.9 Displacement distribution (a) blade base (b) wheel

4. STEADY STATE RESPONSE ANALYSIS

The structure response was analyzed with pressure applied on the blade body and was calculated using Fluent software. Based on the cyclic symmetry boundary condition, the structure response when the pitch diameter was 0、1、7、8、9 was analyzed by a modal superposition procedure. This involves calculating the vibration frequency of whole blade firstly, then selecting the range of frequency sweeping and node for every pitch diameter, and lastly calculating the steady state response.

To study the steady state response systematically, a response curve of 10 representative nodes of blade body was generated, as selected nodes illustrate in Fig.10.
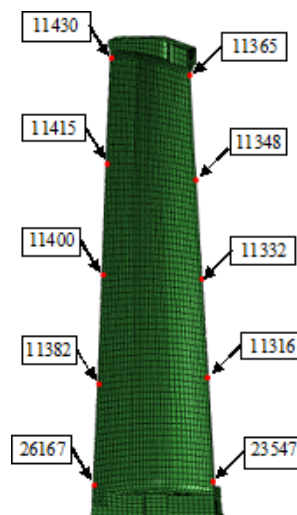
Fig.10 Representative nodes of blade body

According to vibration frequency of the whole blade, the frequency of 260-360Hz was swept. Fig.11 showed the response curve when pitch diameter was 0.
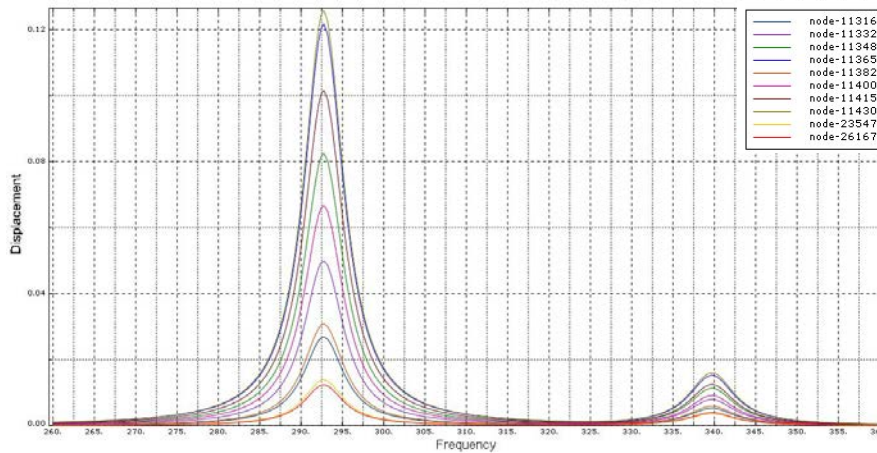

Fig.11 Response curve

It was discovered from modal analysis that the first two order frequency of pitch diameter 0 were 292.75Hz and 339.71Hz. And according to the response curve, the maximum response value distributed at the same frequency value. Fig.12 depicts the stress and displacement distribution of the blade body at 292.75Hz.
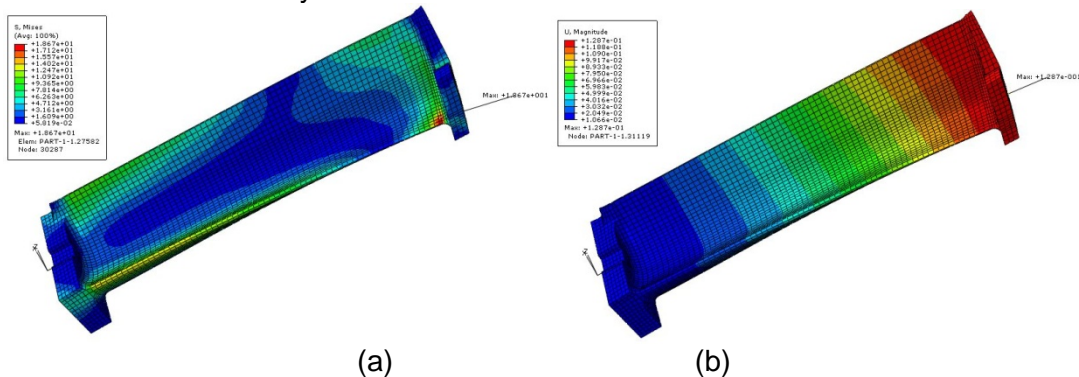

(a)   (b)

Fig.12 Stress and displacement distribution at 292.75Hz (a) mises stress (b) displacement

Maximum stress distributed at node 30287 and response curve of this node was shown in fig.13.

Fig.13 Response curve of node 30287

Fig.14 depicts the stress and displacement distribution of blade body at 339.71Hz.

(a)   (b)

Fig.14 Stress and displacement distribution at 339.71Hz (a) mises stress (b) displacement

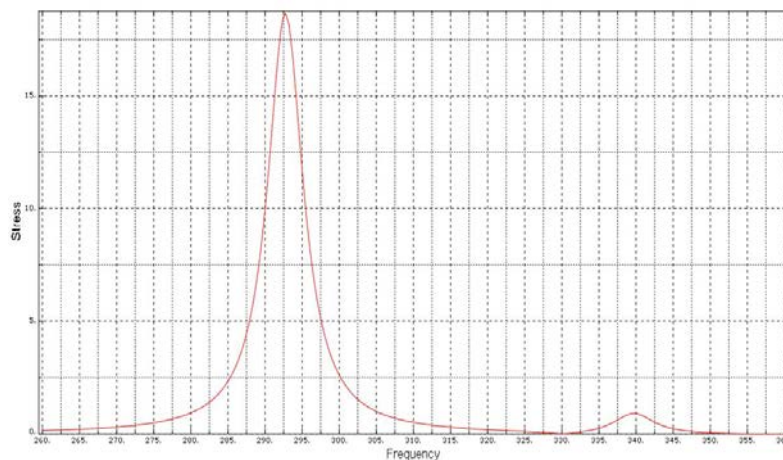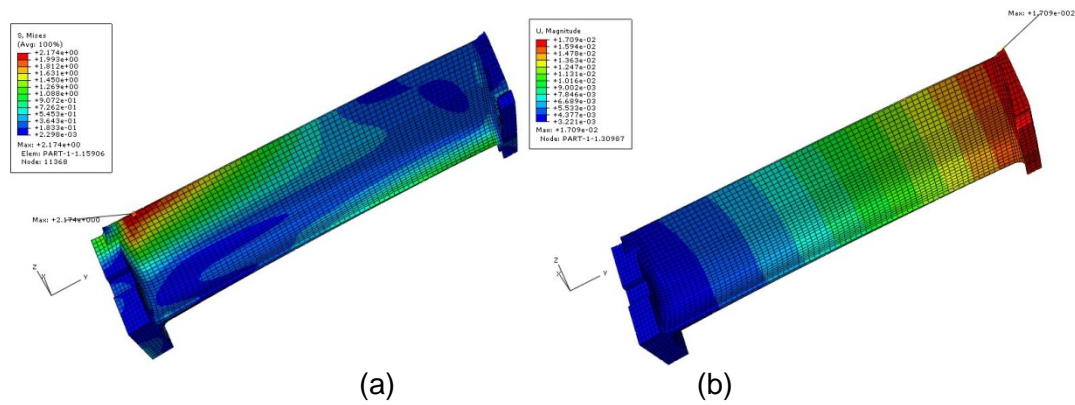Maximum stress distributed at node 30287 and response curve of this node is shown in fig.15.
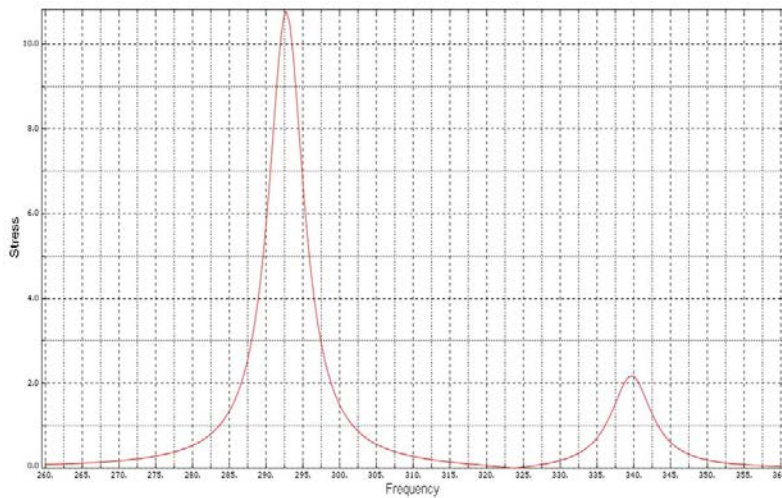

Fig.15 Response curve of node 30287

For pitch diameters of 1, 7, 8, 9, a similar procedure was applied to gain the steady state response. After experimental researching, it was discovered that all the simulation results show good agreement with the experimental results.

5. CONCLUSIONS

1) In ANSA software, the geometry cleanup, the meshing, the loading and other solving settings of a single turbine blade and whole model were conducted, which indicated that ANSA has good performance in pre- process of finite element model.
2) Computational models were calculated in ABAQUS software to analyze the strength and steady state response.
3) For single turbine blade, stress concentration distributed at the transition zone between tip shroud and body. For whole model, small scale yield was distributed at the region of blade base fillet. The structure response with different pitch diameter was analyzed by modal superposition procedure. The simulation results show good agreement with the experimental results.

REFERENCES

(1)   ANSA version 14.1.0 User's Guide, BETA CAE Systems S.A., April 2013
(2)   ABAQUS Analysis User's Manual, Version 6.12. Pawtuchet, RI, USA, Hibbit, Karlsson and Sorensen, 2012