http://ansinet.com/itj

ITJ

ISSN 1812-5638

INFORMATION TECHNOLOGY JOURNAL

ANSInet
Asian Network for Scientific Information
308 Lasani Town, Sargodha Road, Faisalabad - Pakistan

Information Technology Journal 13 (2): 286-293, 2014
ISSN 1812-5638 / DOI: 10.3923/itj.2014.286.293

Numerical Analysis for Blade Loading of Centrifugal Compressor under Multi-operating Conditions

Rong Xie, Liang Guan and Muting Hao
School of Energy and Power Engineering, Dalian University of Technology,
Liaoning, Dalian, 116024, China

Abstract: Now-a-days the impeller blade crack accidents has already become an ignorable problem and this study was primarily aimed to find causes of one blade crack accident in a large-scale centrifugal compressor enterprise. A numerical study was performed to investigate unsteady flow interactions between the rotatable inlet guide vanes and the rotor blades on the semi-open impeller of a centrifugal compressor, which has went through accidents in practice. In order to analyze Von-Mises stress and deformation of the impeller blade in a large-scale centrifugal compressor to find the accident causes, the fluid-solid coupling technology is adopted to solve the equation of fluid-solid coupling under the design condition and the winter condition in this study. The stress on rotor blades mainly includes tensile stress caused by the centrifugal force and bending stress caused by the aerodynamic load in theory. The result of numerical simulation indicated that the distribution characteristics of the Von-Mises stress and deformation under variable conditions well coincide with each other and the loading of impeller blade is larger under the winter condition. There is an error to analyze dangerous work face with single mechanical analysis. The result of computation indicated that the position of high Von-Mises stress in simulation is in agreement with that of the reality. And this provides a certain numerical simulation basis for the accident analysis in practice.

Key words: Centrifugal compressor, numerical simulation, fluid-solid coupling

INTRODUCTION

As the significant equipment of petrochemical enterprises, whether large centrifugal compressors operate safely is directly related to the security of the entire chain of petrochemical products and the economic benefits of petrochemical enterprises as well. Large centrifugal compressor units generally used variable inlet guide vanes to adapt to the modification of the centrifugal compressor flow under off-design conditions in the present. But the interference effect between the stator and rotor is then caused. It will make the internal flow of compressor more complicated, which has been researched as well (Elhadi and Keqi, 2002; Gao *et al*., 2005; Zhou *et al*., 2008; Xi *et al*., 2008). The change of the operation conditions of the impeller blades increased the possibility of blade fracture accidents.

The method based on flow analysis or based on the structural analysis individually is applied in practice, with its research validated as well (Mao *et al*., 2008; Peng *et al*., 2011). This method cannot reflect the interaction in various physical fields, leading to a decrease of simplicity and accuracy in numerical analysis. Multi-physics coupled theory has been widely used with the development of computational fluid dynamics theory and computer technology, with its application validated as well (Xing *et al*., 1997). Schafer and Teschauer (2001) has tried to apply this theory to engineering analysis through the paper in 2001. Song applied the fluid-structure method to analyze an axial aero-engine compressor, shown as paper (Li, 2010). Chen *et al*. (2010) applied this method to analyze the blade stress of the reactor coolant pump of 300 MWe nuclear power plant as the paper in 2010 and also Peng *et al*. (2011) applied this approach to analyze the stress and deformation of twin-spiral scroll plates. Recently, an impeller blade fracture accident of a large centrifugal compressor unit occurred under the winter operating condition. The investigation of its material defects, manufacturing defects had been conducted by manufacturing companies. The vibration characteristics of the impeller have also been commissioned by the relevant units.

This study applied the fluid-structure method to compute the semi-open impeller section of the centrifugal compressor on the basis of experience of these scholars, with the platform of the CFX, namely the Computational

Corresponding Author: Rong Xie, School of Energy and Power Engineering, Dalian University of Technology, Liaoning, Dalian, 116024, China

Fig. 1: Picture of blade failure

Fluid Dynamics (CFD). Efforts have been made to analyze the flow and structural characteristics of the section and to explore the factors that may trigger blade fracture from the view of aerodynamic analysis according to the calculation results. Figure 1 is a fracture accident picture of semi-open impeller blades in a large centrifugal compressor. As can be seen from the Fig. 1, the fracture position of the blade is at the leading edge of the impeller blade root.

NUMERICAL MODEL AND METHOD

Geometry model and mesh generation: In this study, the geometry model contains the rotatable inlet guide vanes and the semi-open impeller with rotor blades. The main structural parameters of the inlet guide vane and the rotor blade on the semi-open impeller can be seen in Table 1.

Based on the technical drawings from the factory, geometry models containing the rotatable inlet guide and the impeller were generated shown in Fig. 2. The fluid domain consists of the following four parts: the inlet extension segment, the guide vane passage, the impeller passage and outlet extension segment. The solid domain contains the impeller blades.

Simulation method: The fluid flow in the fluid domain is solved via the average Reynolds N-S control equation (Versteeg, 1995; Zhang *et al*., 2012).

The finite element analysis method was adopted to analyze the blade stress-strain. The dynamic equilibrium equation of the linear elastic structure can be written as follows:

$$[m]\{\ddot{u}\}+[c]\{\dot{u}\}+[k]\{u\} = \{F\} \tag{1}$$

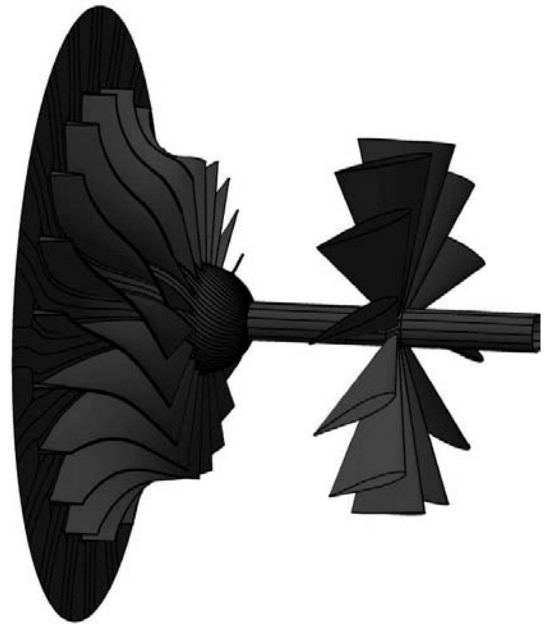

Fig. 2: Geometry model of the variable inlet guide and the impeller

Table 1: Main structure parameters of the centrifugal compressor

	Variable inlet guide vane	Rotor baldes on the semi-open impeller
Inlet diameter (mm)	485	565.92
Outlet diameter (mm)	485	117.09
Blade number	12	19.00

where:

- $[m]$ = Mass matrix
- $[c]$ = Damping matrix
- $[k]$ = Stiffness matrix
- $\{\ddot{u}\}$ = Acceleration column vector of the limited node
- $\{\dot{u}\}$ = Velocity column vector of the limited node
- $\{u\}$ = Displacement column vector of the limited node
- $\{F\}$ = Loading vector (pressure, gravity, centrifugal force) of the limited node

The total stiffness matrix is calculated through the virtual work principle and then the stress σ of each element can be calculated. At last strength check is implemented based on the fourth strength theory:

$$[\sigma] \geq \sqrt{\frac{1}{2}\left[(\sigma_1-\sigma_2)^2+(\sigma_2-\sigma_3)^2+(\sigma_3-\sigma_1)^2\right]} \tag{2}$$

FLUID-STRUCTURE CALCULATION

Coupling method: The fluid-structure method can be divided into two parts according to the method of solving

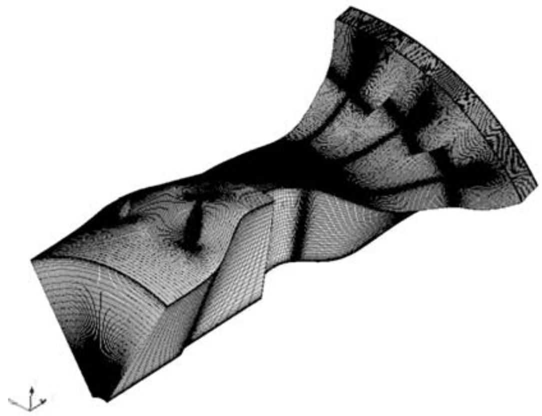

Fig. 3: Mesh of fluid domain of the model

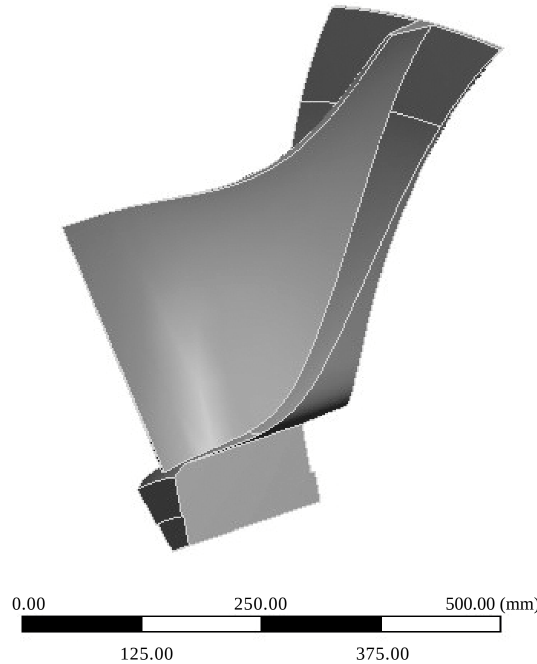

Fig. 4: Solid domain of the centrifugal compressor

the governing equation: One is strong coupling method and the other is weak coupling method. In the strong coupling method, the governing equations of the fluid and solid which need to be solved at the same time are placed in one unified approximate equation, requiring more computer capacity. In the weak coupling method, the fluid and solid equations are alternately solved in time dimensionality and space dimensionality, so it is also called alternately solving method. The analysis focuses on the stress and strain of the centrifugal compressor impeller blades, so the weak coupling method was adopted. At first, the analysis was conducted in the fluid domain and then the results were loaded in the solid domain to calculate the stress of the impeller blade.

Calculation of fluid domain: Domain scaling method was used in the simulation in order to save computer time and memory. The original physical model with 12 guide vanes and 19 guide vanes were scaled into a simulation model with 2 guide vanes and 3 guide vanes. In the fluid domain, the finite volume method was used to conduct the simulation. Hexahedral grids were generated in the fluid domain and there were 4287744 elements in the fluid domain. The turbulence model was adopted when solving the average Reynolds equation. The mesh near the wall is refined to meet the requirements of the turbulence model Y + value. The fluid domain mesh is shown in Fig. 3.

Accidents of compressor often happen under winter condition, which deviates from the design condition and is close to the rotating stall. The outlet pressure of the centrifugal compressor does not vary with the change of flow when working near the rotating stall condition. So it is important to capture the flow variation information. Above all, in the fluid domain simulation, the inlet total temperature, total pressure and outlet mass flow are set (Ding *et al*., 2006). The walls is set as non-slip adiabatic ones. And the interface between the rotor and stator adopts frozen rotor method.

Table 2: Comparison between the results of the simulation and the real monitoring data

	Total temperature (K)	Total pressure Pa
Results of the simulation	349.363	198967.00
Real monitoring data	349.437	201325.00
Relative error (%)	0.031	1.17

The first stage was simulated using the above model and settings to verify the correctness of this method. The results of the simulation are given in Table 2. Calculation was conducted at the heading part of centrifugal compressor based on the real monitoring data as shown in Table 2. A low relative error of less than 3% between the simulation results and real monitoring data was obtained and the accuracy can be accepted.

Coupled simulation method: The solid domain of the centrifugal compressor is shown as Fig. 4. The solid domain numerical calculation only involved the impeller blades, ignoring stress of rotatable inlet guide vanes. Hexahedral grids were generated in the solid domain of impeller blades. Overall, the mesh of the blade contains 13320 nodes and 2409 elements.

The contact part between the impeller blade and the hub is set as a fixed constraint. The pressure surface and suction surface of the blade are set as the fluid-structure surfaces. The result of the fluid domain calculation is loaded on the interface. The numerical simulation method is the sequential fluid-structure interaction method.

RESULTS AND DISCUSSION

Results and discussion of fluid domain: The density of the gas will increase due to the decrease of the inlet temperature with the constant design pressure under the winter condition according to the gas state equation. Volume flow will be reduced, when the real gas mass flow is constant. Thus, a trend towards the rotating stall point from the operating point is inevitable, in accordance with the characteristics of the centrifugal compressor curve. To meet the requirement of the volume flow decreasing under the winter condition, the opening degree of the guide vanes in simulation is adjusted to positive 30°.

Figure 5 shows the fluid domain streamlines maps according to the calculation settings. From the Fig. 5, it can be seen that the overall flow of the centrifugal compressor operates in good condition. The channel flow will produce some rotation angular due to the high-speed rotation of the impeller blades and it combined with the pre-rotation of guide vanes, which determines a more complex internal flow.

Figure 6 is Mach distribution of the impeller blade section. The black dot in the Fig. 6 represents the maximum Mach position and the maximum Mach number is 1.056. The reasons why the transonic region appears were that the linear velocity of impeller outermost is largest and that the impeller made real gas accelerate its motion. The synthesis rate is a little more than 1 Mach.

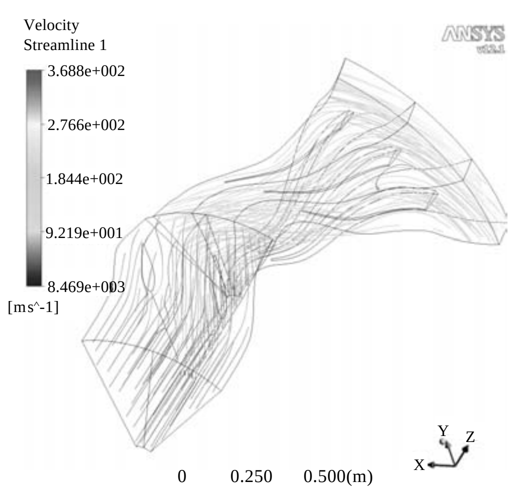

Fig. 5: Stream lines in the computational fluid domain

But the transonic region is relatively small. Because the unevenness of flow increased in the flow channel, the flow loss increased.

Figure 7 is a static pressure envelope curve in the impeller blade root section, the middle section and the top section. It shows that the overall changes of pressure on the blade pressure surface and the suction surface at the same axial position were coincident, except the leading edge and the trailing edge. That indicated the force on the impeller blade surface was uniform and the flow was in good condition within the flow channel. The guide vane wake flow induced strong fluctuations of static pressure at the leading edge of impeller. With the increase of impeller height, the distance between the trailing edge of the guide vane and impeller blade decreased. And the static pressure fluctuation at the leading edge increased. It is bound to increase the bending stress, caused by the aerodynamic loads at the leading edge of the blade root eventually.

Results and discussion of solid domain: Under winter condition, the stress of the impeller blade mainly included: the tensile stress caused by the centrifugal force acting

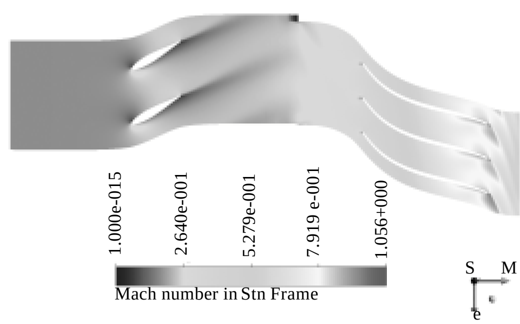

Fig. 6: M distribution of the impeller blade section

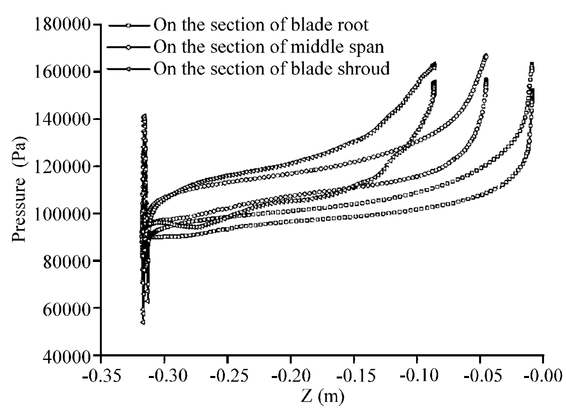

Fig. 7: Static envelope curve of different sections in the impeller blade

on the blades due to the rotating of the impeller, the blade bending stress generated by airflow force and the blade thermal stress caused by the variation of temperature.

The maximum Von-Mises stress region of the impeller blade is the root of the blade, when the load translating to the unity of aerodynamic load, centrifugal force load and fluid-structure. With different loads changing, the value of the stress and the deformation is different. The maximum deformations are 1.3622, 1.2972, 2.4283 mm. Table 3 shows that the centrifugal force is the main factor of the stress acting on the blade. The effect of aerodynamic load is weaker than that of centrifugal force. However, it cannot be ignored. The maximum of deformation is not the algebraic sum of that caused by the aerodynamic load and centrifugal force load, due to the different directions of the maximum deformation.

Figure 8-10 show the stress cloud of impeller blade when the load is defined as aerodynamic load, centrifugal force load and fluid-structure. The Fig. 8-10 show that the maximum Von-Mises stress area on the impeller blades appears in the root region of the blade pressure surface. Tensile stress caused by the centrifugal force is combined with the bending stress caused by aerodynamic load in the fluid-structure interaction effect, so that the Von-Mises stress at the leading edge on the blade pressure surface of blade root increases. So a judgment can be made that the dangerous position is at the leading edge of the blade root region. Figure 10 shows that there is a large Von-Mises stress region at the top of the impeller. It may result in the second flow in the semi-impeller, as shown in Fig. 5.

Table 3: Maximum deformation and Von-Mises stress of variable loading

	Maximum deformation/mm	Maximum Von-Mises/MPa
Aerodynamic load	1.3622	105.65
Centrifugal force load	1.2972	498.99
Fluid-structure	2.4283	541.55

Figure 11 is the stress cloud of impeller blade under the fluid-structure interaction of the design condition. Von-Mises stress which is at the root of leading edge on the impeller pressure surface has improved significantly due to the fluid-structure interaction. Under off-design condition, as shown in Fig. 10, the Von-Mises stress value which is at the root of leading edge on the blade pressure surface is significantly higher than that under the design conditions. As the enlarged view of Fig. 10 and 11 shown, there is a vortex at the root of the blade on the leading edge, so the flow is choking and the pressure is high. This is coincided with the high Von-Mises stress under the fluid-structure interaction. It can be infered that the root of blade on the leading edge is the dangerous working position under off-design condition.

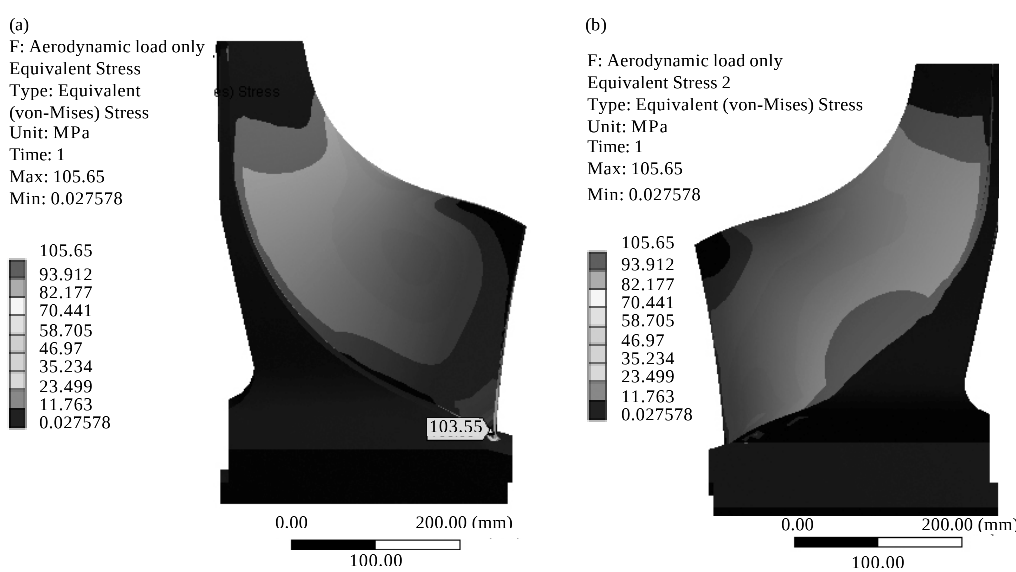

Fig. 8(a-b): Stress cloud of the impeller blade under the aerodynamic load on the (a) Blade pressure side and (b) Blade suction side

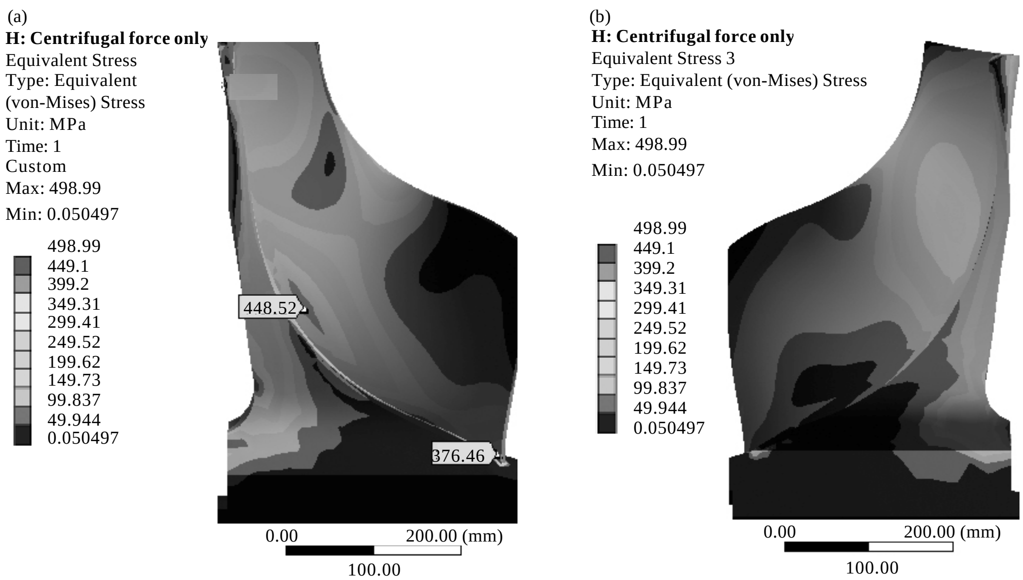

Fig. 9(a-b): Stress cloud of the impeller blade under the centrifugal force load on the (a) Blade pressure side and (b) Blade suction side

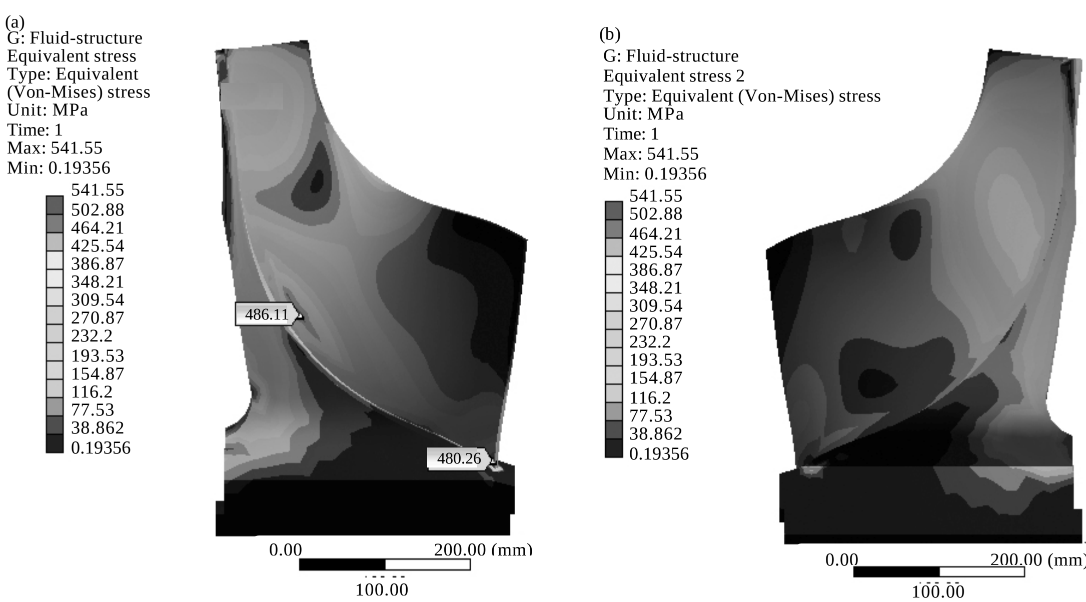

Fig. 10(a-b): Stress cloud of the impeller blade under the fluid-structure interaction on the (a) Blade pressure side and (b) Blade suction side

In summary, the Von-Mises stress of the impeller blade is the largest under the fluid-structure interaction. The tensile stress caused by the centrifugal force in the centrifugal compressor is the main factor for the stress problem. There is an error of considering the aerodynamic load when checking the stress only. The result of computation indicated that the position of high Von-Mises stress is in conjunction with that of the real, which provides a certain numerical simulation basis for the accident analysis in practice.

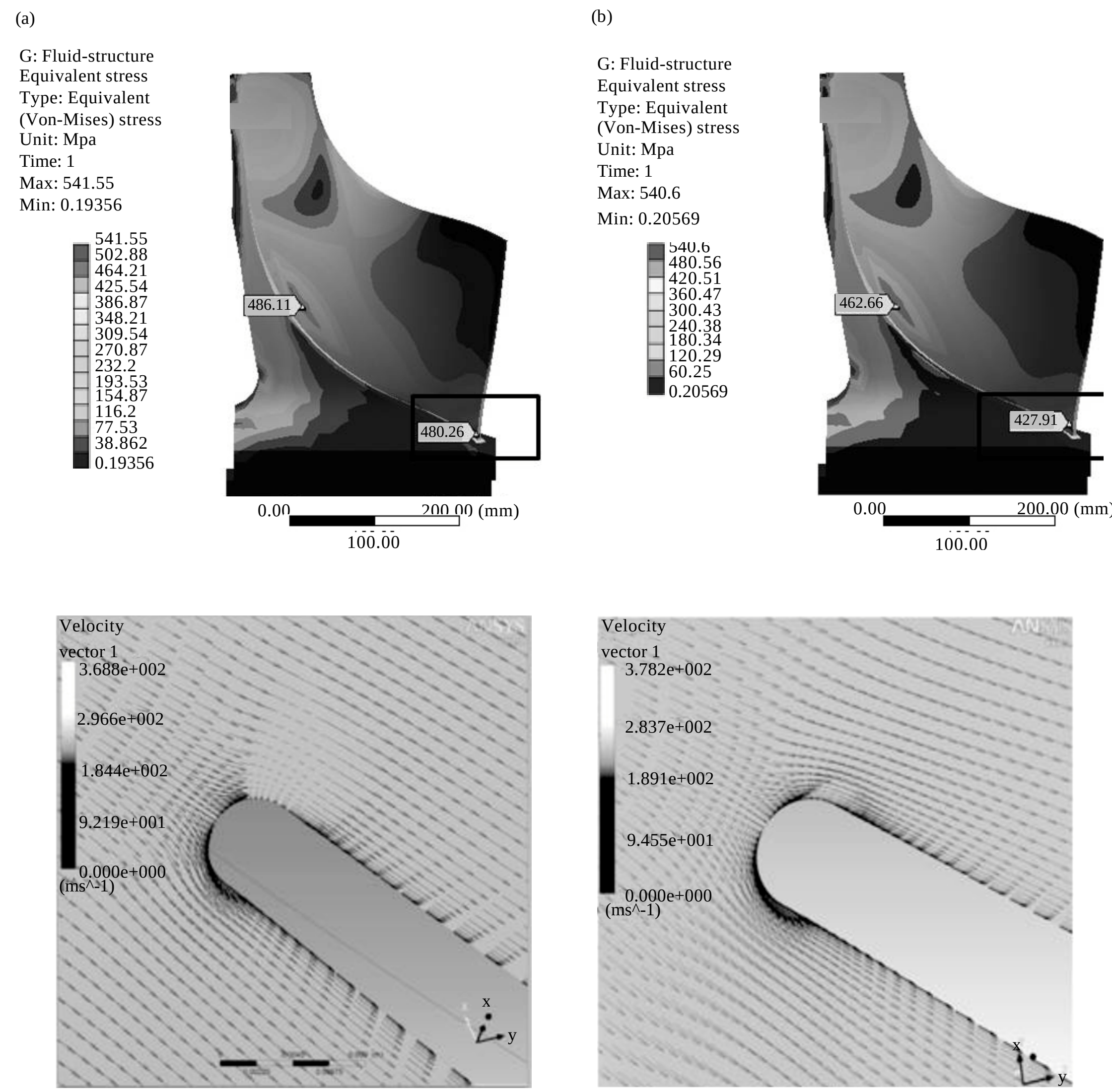

Fig. 11(a-b): Stress cloud of the impeller blade under the fluid-structure interaction of the design condition on the (a) Blade pressure side and (b) Blade suction side

CONCLUSION

The fluid-structure technique is adopted to solve the stress distribution on the semi-open impeller blade. The result of numerical simulation indicated that the distribution characteristics of the Von-Mises stress and deformation under variable conditions coincide with each other well, but the loading of impeller blade is larger under the winter condition. There is an error in the situation of dangerous work face in single mechanical analysis. There is also an error when taking the stress check if consider the aerodynamic load only. Moreover, this computation can be improved with further explaining the cyclical fatigue failure of the impeller blades and taking the impact of the structural deformation of the fluid flow field into account, which has not been discussed in this study. The two-way coupling between the fluid-structure calculation is not achieved, which will be the method of further calculation and the simulation.

ACKNOWLEDGMENTS

The study is supported by "the Natural Science Funds of Liaoning Province" (No. 201102031) and National Ministries Project (Grant No. A0920110001) of P.R. China.

REFERENCES

Chen, X., D. Yuan and M. Yang, 2010. Blade stress of the reactor coolant pump of 300 MWe nuclear power. Chinese J. Mech. Eng., Vol. 4

Ding, M.Y., C. Groth, S. Kacker and D. Roberts, 2006. CFD Analysis of off-design centrifugal compressor operation and performance. http://www.dmut.net/enshowpaper.aspx?id=11044

Elhadi, E.E. and W. Keqi, 2002. Simulation of vortex flows in axial flow fan using computational fluid dynamics. Inform. Technol. J., 1: 242-249.

Gao, L., G. Xi,L. Zhou and S. Wang, 2005. Experimental and computational investigation of a centrifugal compressor stage. J. Power Energy, 219: 25-33.

Li, X., 2010. The flow simulation in the compressor with the effect of fluid-solid interaction. Harbin Institute of Technology, Harbin, China

Mao, Y., D. Qi and Q. Xu, 2008. Numerical study on high cycle fatigue failure of a centrifugal compressor impeller blades. J. Xi'an Jiaotong Univ., 11: 1336-1340.

Peng, B., H. Zhang, L. Zhang and W. Jin, 2011. Stress and deformation analysis of twin-spirals scroll plates. Int. J. Digital Content Technol. Appl., Vol. 5

Schafer, M. and I. Teschauer, 2001. Numerical simulation of coupled fluid-solid problems. Comput. Methods Applied Mech. Eng., 190: 3645-3667.

Versteeg, H.K., 1995. An Introduction to Computational Fluid Dynamics the Finite Volume Method. Pearson Education, New Delhi, India, Pages: 257.

Xi, G., L. Liu and J. Jiang, 2008. Numerinal investigation of stator clocking in a centrifugal compressor. J. Eng. Thermophys., 9: 1495-1504.

Xing, J., S. Zhou and E. Cui, 1997. A survey on the fluid-solid interaction mechanics. Adv. Mech., 27: 19-38.

Zhang, Y., R. Yang, H. Zheng and F. Pan, 2012. A three-hybrid treatment method of the compressor's characteristic line in performance prediction of power systems. J. Converg. Inf. Technol., Vol. 7.

Zhou, L., G. Xi and Y. Cai, 2008. Numerical and experimental investigation to unsteady IGV-impeller-diffuser interaction. Chinese J. Applied Mech., 2: 202-205.