Computation of stress distribution in a mixed flow pump based on fluid-structure interaction analysis

F F Hu¹, T Chen², D Z Wu¹ and L Q Wang¹

¹ Institute of process equipment, Zhejiang University, Hangzhou, China
² Hangzhou special equipment inspection institute, Hangzhou, China

E-mail: hff1002@zju.edu.cn

Abstract. The internal flow evolution of the pump was induced with impeller movement. In various conditions, the peak load on centrifugal blade under the change of rotational speed or flow rate was also changed. It would cause an error when inertia load with a safety coefficient (that was difficult to ascertain) was applied in structure design. In order to accurately analyze the impeller stress under various conditions and improve the reliability of pump, based on a mixed flow pump model, the stress distribution characteristic was analyzed under different flow rates and rotational speeds. Based on a three-dimensional calculation model including impeller, guide blade, inlet and outlet, the three-dimension incompressible turbulence flow in the centrifugal pump was simulated by using the standard k-epsilon turbulence model. Based on the sequentially coupled simulation approach, a three-dimensional finite element model of impeller was established, and the fluid-structure interaction method of the blade load transfer was discussed. The blades pressure from flow simulation, together with inertia force acting on the blade, was used as the blade loading on solid surface. The Finite Element Method (FEM) was used to calculate the stress distribution of the blade respectively under inertia load, or fluid load, or combined load. The results showed that the blade stress changed with flow rate and rotational speed. In all cases, the maximum stress on the blade appeared on the pressure side near the hub, and the maximum static stress increased with the decreasing of the flow rate and the increasing of rotational speed. There was a big difference on the static stress when inertia load, fluid load and combined loads was applied respectively. In order to more accurately calculate the stress distribution, the structure analysis should be conducted due to combined loads. The results could provide basis for the stress analysis and structure optimization of pump.

1. Introduction
Stress analysis of pump impeller can only be performed by numerical methods due to its structure complexity. However, at present, the strength design of impeller is mainly based on empirical formula, which would cause a large allowance of strength and the waste of material. Meanwhile, the blade thickness can also affect the flow rate and the head of pump. Therefore, it is important for the strength analysis of impeller to the design of pump.

And nowadays, many researches have been done on the performance of pump during steady-state period [1]-[4]. The studies showed that the pressure distribution on the blade was different under different conditions, such as various flow rate and rotational speed. The pressure distribution would affect the stress on the blade.
In recent years, many researches have been done in strength analysis of pump blades considering fluid force. R.A. Saeed\textsuperscript{[6]}, R.Nergu\textsuperscript{[7]} and Nava\textsuperscript{[8]} carried out numerical studies on the stress induced in a Francis turbine runner by steady fluid flow, which was based on the one-way coupled simulation. It was found that maximum static stresses occurred at the transition between the blades and the crown. SUN\textsuperscript{[9]} proposed an analysis method to calculate the stress distribution on the blade of a multi-phase pump. This method was proved to be able to show the true pressure status on the surface of propeller. Luo\textsuperscript{[10]} carried out transient dynamic structural analysis of the turbine under fluctuant work conditions, which was based on steady flow simulation. There results showed that the reason of cracks was not the static stress but the high dynamic stress.

The studies presented above are all about the blade stress of the Francis turbine runner. However, there were few researches on blade strength for pump impellers. The purpose of this paper is to study the static stress induced in pump impellers by the steady loading. The following analysis is based on the sequentially coupled simulation approach, which consists of fluid flow analysis and structure finite element analysis. Due to high rigidity of the impeller blades, the influence of the blade deformation was not taken into account in the fluid analysis. The structural analysis was performed using ANSYS workbench software. Firstly, with the method of computational fluid dynamics (CFD), numerical simulation is conducted to study the performance of pump under steady conditions which provides the pressure distribution on the blade. Then the stress on the blades of pump is investigated by use of the FEM approach under different flow rate and rotational speed.

2. Fluid flow simulation

Analysis of fluid flow through the pump impeller is needed to study the hydraulic load on the impeller. CFD simulation of all the flow domains is performed to obtain pressure distributions on the blade.

2.1. Physical model and boundary conditions

The computational model for steady simulation is shown in Figure 1. The main components contain inlet pipe, impeller, guide vane and outlet pipe. For the computational domain, unstructured 3D tetrahedral meshing has been applied.

In the steady-state fluid flow simulation, the flow in the impeller is described in the rotating frame of reference, whereas the flow in the stationary bodies is expressed in the stationary frame of reference. The boundary conditions applied for the calculation are a uniform velocity distribution at the inlet and a constant static pressure at the outlet. In the solution, stand $k - \varepsilon$ model is taken. SIMPLEC algorithm is adopted to solve the pressure-velocity coupled discrete equations. Residuals are set as 10e-5.

2.2. CFD results and discussions

The performance and pressure distribution of mixed flow pump is obtained by the steady-state flow field simulation. In the computation, the rotational speed is set to $n_0$ and the flow rate is set to 100\%$Q_0$, 88\%$Q_0$, 76\%$Q_0$, 65\%$Q_0$, 53\%$Q_0$ separately, where $Q_0$ is the designed flow rate. Figure 2 shows the numerical and experimental characteristics of mixed flow pump, where $H_0$ is a constant which describes the head at designed conditions, and $H$ is a variable which describes the head at variable flow rate $Q$. The numerical result shows good agreement with the experimental results. It indicates the numerical simulation is reasonable. The fluid pressure on the pressure side is higher than that on the suction side of the blade.
3. Stress analysis in pump impeller

Stress analysis of the blade subjected to different loading conditions is important for the structural design of pump. In this study, the loads on the blades include the inertia force and the fluid pressure acting on fluid-solid interface induced by fluid flow. The inertia force includes the blades’ own weight and the rotational inertia force.

3.1. The method of data transferred from CFD to structure

The load transfer method involves two analyses, CFD and structural analysis, each belonging to a different field. The two fields are coupled by applying results from CFD as loads in structural analysis. The fluid-structural interaction (FSI) analysis is conducted using a System Coupling component system in ANSYS workbench. FSI analysis is set up by connecting a System Coupling component system to Mechanical and Fluent systems. The geometry cells for the two systems share a single geometry, and the solution cell in the CFD system provides pressure load data to the setup cell in the static structural system.

The pressure data from the cell or face zones of a CFD simulation is mapped using zeroth-order interpolation onto locations associated with a finite element analysis (FEA) mesh. The CFD results are written to a file for inclusion into a FEA simulation.

3.2. Computational model and boundary conditions

As shown in Figure 3, the structural model contains the blade and the flange of pivot. The blade mesh in CFD simulation and stress analysis is finished respectively, so there is a little difference between them. But the geometrical model for the CFD simulation and structural analysis is perfectly matched on the fluid-solid interface. This can make sure the data completely transferred from CFD to structure. The following boundary conditions are applied on the model: zero displacements are assumed at the flange surface (fix support); loads due to the inertia force are applied on the model; loads due to fluid pressure distribution imported from CFD simulation are assumed on the blades.

Table 1 shows the comparison between the fluid force in CFD simulation and the mechanical force in stress analysis transferred from CFD results, taking one condition (at 100%Q0) for an example. As illustrated in table1, the data is almost completely transferred from CFD results to structure analysis, the difference is all below 10%.
Table 1. Comparison between CFD computed forces and mechanical forces

| Location    | CFD computed forces (N) | Mechanical forces (N) |
|-------------|-------------------------|-----------------------|
|             | X-component | Y-component | Z-component | X-component | Y-component | Z-component |
| Pressure    |            |            |            |            |            |            |
| side        | -551.11     | 328.04     | -845.44    | -578.14    | 358.09     | -901.35    |
| Suction     | 2101.1      | -1237.3    | 3195.1     | 2099.3     | -1242.8    | 3199.4     |

3.3. Analysis results and discussions

3.3.1. The influence of inertia force and fluid pressure on the blade strength. In order to figure out the impact of the inertia force and the fluid pressure on the impeller strength, firstly the structure analysis is carried out when only inertia force or fluid pressure is applied, then the combined forces are considered to perform the stress simulation.

To analyze the influence of the centrifugal force on the blade, the stress analysis is carried out under different rotational speed, such as 100% \( n_0 \), 83% \( n_0 \), 67% \( n_0 \), 50% \( n_0 \), 33% \( n_0 \), where \( n_0 \) is the designed rotational speed. Figure 4 shows the stress field in terms of von Mises equivalent stress under different rotational speed, taking 100% \( n_0 \) and 50% \( n_0 \) as an example. Figure 5 shows the variation of the stress with rotational speed on the blade, where \( n \) is a variable rotational speed. The static stress almost linearly changes with the rotational speed.

For analyzing the influence of flow rate on the blade stress, the stress analysis of pump impeller is performed at different flow rate, as mentioned in 2.4.1. Figure 6 shows the stress distribution due to fluid pressure at designed flow rate. The stress distribution is same as that due to inertia force. Figure 7 shows the variation of the stress with flow rate on the blade. With the increasing of the flow rate, the static hydraulic pressure reduces, and the static stress decreases.

The combined effect of inertia force and fluid pressure is also analyzed under designed rotational speed and designed flow rate, as shown in Figure 8.

Comparing Figure 4 and Figure 6, it can be found that at the separate action of inertia force and fluid pressure, the distribution law of stress is generally consistent and the location of maximum static stress is almost not changed. It all appears at the root of the pressure side. But there is a difference on the value of the maximum static stress, 168.21MPa for the effect of fluid pressure and 119.56MPa for the effect of inertia force.

As shown in Figure 8, the maximum static stress under designed conditions is 56.4MPa due to the combined loads, which is much lower than that due to inertia force or static pressure, as shown in...
Figure 4 and Figure 6 respectively. This is caused by the interaction of fluid load and inertia load, which will be discussed in the next section.

3.3.2. The blade strength due to combine loads. For analyzing the influence of combined loads on the blade stress, the stress analysis of pump impeller is performed. At these cases, the load acting on the blade mainly includes the tensile stress induced by centrifugal force and the bending stress induced by fluid pressure.

Characteristic stress field at different flow rate is obtained in terms of von Mises equivalent stress taken 53%Q₀ as an example, as shown in Figure 9. In all cases, the highest stress values in the blade are found at the root of the pressure side.
Figure 10 shows the variation of the static stress on the blade at various flow rate due to different loads. It can be found that at lower flow rate the static stress due to combined loads is a little higher than that due to the inertia force. But at higher flow rate, the stress due to combined loads is much lower than that due to inertial force. And the maximum stress due to fluid force is much bigger than that due to combined loads at the same flow rate. The reasons for this phenomenon are described below.

The centrifugal force points outward along radial direction and is in the same direction as rotational speed along circumferential direction. However, the hydraulic pressure in pressure side is higher than that on the suction side on the blade. The direction of resultant hydraulic force is inward along radial direction and opposite to the rotational speed in the circumferential direction. As a result, the torque induced by hydraulic pressure is contrary to that due to centrifugal force. So the static stress due to combined loads is lower than that due to fluid load. Under the effect of combined loads, at higher flow rate, the stress due to combined loads is lower than that due to inertial force. But at lower flow rate, because of the big difference between pressure side and suction side, the stress due to combined loads is larger than that due to inertial force. In a word, at all cases, the hydraulic load counteracts with the centrifugal load. And the stress distribution is determined by the combined action of hydraulic load and inertial load.

From the result above, it can be seen that due to single load such as inertia load or fluid load, the maximum static stress at all cases is approximately 27% of the yield strength (450MPa) for the inertial load and 47% of the yield strength for the fluid load; Due to combined load, the maximum static stress at all cases is approximately 36% of the yield strength. So it is required to consider all the forces when the structure design is carried out. Otherwise, the structure design will result in a large margin of strength or insufficient strength.

4. Conclusion

A one-way coupled simulation was carried out to analyze the static stress on the blade of a mixed flow pump. Based on the data from CFD simulation, the stress analysis was performed separately due to inertia load, or fluid load, or combined load. The results show that the maximum static stress is far less than the yield strength at all cases; the contribution of fluid load to the combined stress distribution depends on the flow rate. At lower flow rate, the fluid load enhances the effect of the inertia load. At higher flow rate, the fluid load weakens the action of inertia force. At all cases, the hydraulic load counteracts with the centrifugal load.

References

[1] Jaysree P, Pandey N K, Muduli U K, et al 2013 Chemical engineering communications 200(4) 471-493.
[2] Liu Z S, Liu Q Z and Wang H J 2008 Journal of harbin engineering university 29(12) 1304-08.
[3] Yang M G, Gu H F, Liu D, et al 2004 Transactions of the chinese society for agricultural machinery 35(6) 72-74.
[4] Yang M G, Gu H F, Liu D, et al 2006 Chinese journal of mechanical engineering 42(12) 180-5.
[5] Zhao W Y, Ma P F, Bai S B, et al 2012 Jounal of Lanzhou University of Technology 38(1) 41-45.
[6] Saeed R A, Galybin A N and Popov V 2010 Advances in Engineering Software 41 1245-55.
[7] Negrua R, Munteanb S, Marsuvina L, et al 2012 Computational Materials Science 64 253-259.
[8] Nava J M F, Gómez O D and Hernández JARL 2006 Flow Induced Stresses in a Francis Runner using ANSYS International ANSYS conference proceedings (Pittsburgh, PA).
[9] Sun B Q 2008 Oil field equipment 37(7) 34-38.
[10] Luo Y Y, Wang Z W and Lian Q W 2005 J. of Tsinghua Univ. (Sci & Tech) 45(2) 235-237.