Application of energy gradient theory in flow instability in a centrifugal pump

H S Dou and W Jiang

Faculty of Mechanical Engineering and Automation, Zhejiang Sci-Tech University, Hangzhou 310018, China

Email: huashudou@yahoo.com

Abstract. The flow instability in a centrifugal pump is studied using the energy gradient theory. Since the Re is high, the base flow is assumed to be turbulent. The distribution of the energy gradient function K at various flow rates is obtained from numerical simulations. According to the energy gradient method, the area with larger value of K is the place to cause instability and to be of high turbulence intensity. The results show that instability is easier to be excited in the area of impeller outlet and volute tongue. In order to improve the stability of centrifugal pumps working under low flow rate condition, carefulness must be taken in these two key areas.

1. Introduction

Centrifugal pumps are widely used in various industrial equipments. Since these devices consume a lot of energy, it is important to design energy-saving pumps and to improve the performance of these pumps. Experimental study and numerical calculation have been carried out in the past years and much progress has been obtained [1-2]. Kaupert and Staubli [3] have measured the unsteady pressure field inside an impeller of a centrifugal pump using piezoresistive pressure transducers and a telemetry system. They measured pressure pulsation amplitude and found that they were particularly high at the trailing edge of the blades, on their pressure sides, and reached even 35% of the pump head at off-design conditions. In Guleren and Pinarbasi’s study [4], numerical simulation was used to explore the stall phenomenon in a centrifugal pump, and the result verified the accuracy of the previous experimental results on flow stall. Till now, so many researchers have attributed their work to study the flow instability in pumps, however the physical mechanism of flow instability is still unknown.

In this study, flow characteristic in centrifugal pump is simulated based on the three-dimensional (3D) Reynolds-averaged Navier-Stokes equations. The present numerical results are validated comparing with experimental data. Then, energy gradient theory is applied to analyze the physical mechanism of flow instability.

2. Revisiting the Energy Gradient Method

From the classical theory of the Brownian motion, the fluid particles exchange energy and momentum all the time via collisions. The fluid particle will collide with other particles in transverse directions as it flows along its streamline, and this particle would obtain energy expressed as \( \Delta E \) after many cycles; at the same time, the particle would drop energy due to viscosity along the streamline; with the same periods, the energy loss expressed as \( \Delta H \) would be considerable. Consequently, there exists a critical value of the ratio of \( \Delta E \) and \( \Delta H \), above which the particle would leave its equilibrium by moving to a new streamline with higher energy or lower energy and below which the particle would
not leave its streamline for its oscillation would be balanced by the viscosity along the streamline. Making reference to [5-10], we can express the criteria of instability as follows:

\[ F = \frac{\Delta E}{\Delta H} = \frac{\left( \frac{\partial E}{\partial n} 2\bar{A} \right)}{\left( \frac{\partial H}{\partial S} \pi \frac{u}{\omega_d} \right)^2} = 2 \pi^2 K \frac{v'_m}{u} < \text{Const} \]  

(1)

where

\[ K = \frac{\partial E/\partial n}{\partial H/\partial s} \]  

(2)

Here, \( F \) is a function of coordinates which expresses the ratio of the energy gained in a half period by the particle and the energy loss due to viscosity in the half period. \( K \) is a dimensionless field variable (function) and expresses the ratio of transversal energy gradient and the rate of the energy loss along the streamline. \( E = P + \frac{1}{2} \rho U^2 \) is the kinetic energy per unit volumetric fluid, \( s \) is along the streamline direction, and \( n \) is along the transverse direction. \( H \) is the loss of the total mechanical energy per unit volumetric fluid along the streamline for finite length, which can be calculated from the Navier-Stokes equations. Further, \( \rho \) is the fluid density, \( u \) is the streamline velocity of main flow, \( \bar{A} \) is the amplitude of the disturbance distance, \( \omega_d \) is the frequency of the disturbance, and \( v'_m = \bar{A} \omega_d \) is the amplitude of the disturbance of velocity.

3. Centrifugal pump model and grid

The pump consists of an impeller with four backward curved blades and a volute. The geometric parameters of the pump are shown in table 1. The schematic diagram of flow channels in centrifugal pump is shown in figure 1.

Table1. Main parameters of the centrifugal pump

| Q   | 551 m³/h | Z   | 4  |
|-----|---------|-----|----|
| H   | 25 m    | η   | 78%|
| n   | 980 rpm | P   | 48 kw|
| D₀  | 230 mm  | n₀  | 34.23|
| D₁  | 450 mm  |     |    |

Figure 1. Flow space in centrifugal pump  
Figure 2. Centrifugal pump grid
In the present study, the transport equations are solved using the finite volume method. Structured hexahedral cells are generated to define the inlet zones (28080 cells), while tetrahedral cells are used to define the impeller and volute (460750 cells and 891841 cells respectively). In total, there are 1380671 cells and 304362 nodes in the whole domain. This mesh size could give reasonable and reliable result for the pump performance and allows us to analyze the details of the main phenomena in the pump. The grid of whole domain is shown in figure 2.

4. Numerical algorithm and Results Discussions

Flow characteristic in the centrifugal pump is studied using the finite volume method and the SIMPLEC algorithm. Time-averaged Navier-Stokes equations are chosen as the governing equations and the RNG $k-\varepsilon$ turbulent model is applied in this study. The mesh independence of the computation has been validated using three-sets of meshes. The accuracy of numerical results has been verified by comparing the present results with the experimental ones.

Flow parameters of the whole flow field have been obtained from computations. The magnitudes of static pressure, velocity, total pressure, and energy gradient function $K$ at the design condition are shown in figure 3 for comparison. It can be seen that the static pressure and velocity are well distributed in some impeller channels except the one near the tongue. The total pressure increases through the impeller due to the work done by blades. At the impeller exit, the total pressure has its maximum, but it is not axisymmetric in the circumferential direction. There is a valley zone in the tongue area.

The distribution of $K$ at design condition is shown in figure 3 (d). It is found from the figure that the value of $K$ is larger in the areas of impeller outlet and the vaneless diffuser space as well as the tongue area. This means that the flow instability may be first produced in these areas. The mechanism of this phenomenon is due to the following reasons. Firstly, the Re number at the impeller outlet is very large and there is large energy gradient in the flow. Secondly, there is strong jet-wake at impeller outlet and there is strong energy gradient between streamlines. Thirdly, there is strong interaction between the impeller and volute tongue which leads to flow nonuniformity. The distribution of $K$ in figure 3 explains the experimental results in literature [11-12]. Guo [11] made some numerical simulations in order to analyze the coupling of impeller and volute in a centrifugal pump. He found that the nonuniformity of the circumferential flow at the inlet of volute is very strong due to the interference of the impeller and the volute. In particular, the nonuniformity is most intense in the volute tongue area. Wuibaut et al [12] carried out experiments to study the distribution characteristics of the transient flow in design and off design conditions by using PIV technology. They found that there is strong jet-wake flow pattern at the impeller exit and in the diffuser. They also found that the intense unstable flow occurring in vaneless diffuser develops to the outlet of impeller in small flow rate condition.

The distributions of the energy gradient function $K$ in the pump, which are under different flow rates, are shown in figure 4. As indicated in the figure, the value of $K$ is large in the red areas and is lower in the blue areas. According to the energy gradient theory, it is noted here that the larger the value of $K$, the more unstable the flow and the stronger the turbulent intensity. This means that the flow is much easier to lose its stability with a large value of $K$.

The design condition is shown in figure 4 (e). It can be seen that the value of $K$ is larger in the area of impeller outlet, and the pattern of jet-wake is clearly indicated. Instability can be easily generated in these areas. This is why most unstable flows were observed in these regions in literature. However, the flow within the impeller, especially in the entrance section, the flow is stable. Within the impeller, it is observed that the two regions near the impeller outlet tend to lose its stability because of the boundary layer separation which resulted from the diffusing of the blade channel.

When the flow rate is larger than the design flow rate, e.g., $Q/Q_n=1.2$ as in figure 4 (f), the flow is even more stable than that in the design condition. This is because the total pressure head is lower compared with that in the design condition.
Figure 3. Static pressure, total pressure, velocity and energy gradient function K at design condition. (a) Static pressure; (b) total pressure; (c) velocity; (d) K.

When the flow rate is lower than the design condition, e.g., Q/Qn=0.8 and Q/Qn=0.6 as shown in figure 4(c) and 4(d) respectively, the value of K tends to be much larger comparing with figure 4(e) and 4(f). Moreover, the largest value of K occurs near the vortex tongue where the base flow is much easier to lose its stability. When Q/Qn = 0.48 and Q/Qn=0.28 as shown in figure 4(a) and 4(b), the distribution of K is similar to those in figure 4 (c) and 4(d), and the only difference is that the value of K is much higher. Therefore, the flow in the pump is more unstable when the flow rate is reduced. As it is shown, the flow in figure 4(a) is much deteriorated and the value of K is much higher in the whole pump.
Figure 4. Distribution of energy gradient function $K$ at different flowrate conditions
(a) $Q/Q_n=0.28$;(b) $Q/Q_n=0.48$;(c) $Q/Q_n=0.6$;(d) $Q/Q_n=0.8$;(e) $Q/Q_n=1.0$;(f) $Q/Q_n=1.2$
($Q_n$ is the flow rate at design condition)

5. Conclusions
The flow field in a pump is simulated using the finite volume method and the mechanism of flow instability in the pump under different flow rates is studied. The main conclusions are summarized as follows:

1) At the design condition, the flow in the entrance section within the impeller is stable; however, the flow tends to be unstable near impeller outlet and tongue. The jet-wake pattern from the impeller outlet has important influence on the flow stability in the pump.

2) When the flow rate is larger than the design flow rate, the flow is even more stable than that in the design condition. When the flow rate is lower than the design condition, the flow in the impeller is much deteriorated and the flow becomes much unstable.
(3) As the flow rate is reduced, the value of K in the pump increases. The maximum of K always occurs near the outlet of the impeller and the tongue. These areas are the origin to lead to instability.

(4) All the above analysis demonstrates that the application of energy gradient method in the study of internal flow in the turbomachinery is reliable. The results of this study provide with some support to reveal the physical mechanism of complex flow in centrifugal pump. The results are in agreement well with the experimental results in literature.

References
[1] Guan X F 1995 Handbook of modern pump technology (Beijiang: Aerospace)
[2] Hou S Q, Wang C X and Lin J Z. 2005 Fluid machinery 33(5) 30-34.
[3] Kaupert K A, Staubli T. 1999 Transactions-American society of mechanical engineers journal of fluids engineering 121 621-26.
[4] Guleren K M and Pinarbasi A. 2004 Journal of Mechanical Engineering Science 218(4) 425-35.
[5] Dou H S 2006 International Journal of Non-Linear Mechanics 5(41) 512-17.
[6] Dou H S 2011 International Journal of Physical Science 6(6) 1411-25.
[7] Dou H S, Khoo B C and Yeo K S 2008 Inter. J. of Therm. Sci. 47 1422-35.
[8] Dou H S and Khoo B C 2009 Modern Physics Letters B 23(3) 457-60.
[9] Dou H S and Khoo B C 2010 Modern Physics Letters B 24(13) 1437-40.
[10] Dou H S and Khoo B C 2005 arXiv preprint nlin 0501048
[11] Guo P C 2006 Study on mechanical properties and numerical prediction of complex internal flow in hydraulic machinery (Xi’an: Xi’an University of Technology).
[12] Wuibaut G, Bois G and Dupont P 2002 Journal of Fluids Engineering Transactions of the ASME 124(9) 791-797.