International Conference on Advances in Computational Modeling and Simulation

Prediction and experimental verification of vortex flow in draft tube of Francis turbine based on CFD

Yongzhong Zenga, Xiaobing Liua*, Huiyan Wanga

*aSchool of Energy and Environment, Xihua University, Chengdu 610039, China

Abstract

The three-dimensional unsteady turbulent flow in the draft tube of a Francis turbine was calculated by using the SIMPLE algorithm with the body-fitted coordinate and tetrahedral grid system with the help of FLUENT software. This calculation was based on the Navier-Stokes equations and LES (large-eddy simulation) model. The flowing situations in the draft tube under three operating conditions of this turbine were successfully calculated. The PIV (particle image velocimetry) was used to measure the distribution of flow field at the draft tube cone section at 4 horizontal sections from the outlet of turbine runner. Specialized image analysis software was used to process the experimental results. Through comparison, the results of numerical simulations were in good coincidence with the experimental results.

© 2011 Published by Elsevier Ltd. Selection and/or peer-review under responsibility of Kunming University of Science and Technology. Open access under CC BY-NC-ND license.

Keywords: Turbine; Vortex flow; LES model; PIV

1. Introduction

With the rapid development of computer technology and the continuous improvement of turbulent numerical simulation theory in recent years, the obtaining of precise flow field calculation results through numerical simulation in replacement of physical experiments has become one of the main researching means at present. Based on this premise, some researchers have been started to study this problem through numerical simulation. The development of numerical simulation technologies of flow field provides the possibility to study the flow in the turbine. Meanwhile, it can help us to fully understand the relationship

* Corresponding author: Tel: +86-28-87720083; fax: +86-28-87720521.
E-mail address: liuxb@mail.xhu.edu.cn
between the shape of vortex strip in the draft tube and the operating conditions and geometric parameters. This is very effective for the hydraulic design and the predication of overall performance of a turbine [1].

Shyy and Braaen were the first in using the turbulent calculation method of k-ε model to study steady state flow in the draft tube of a turbine. They verified the feasibility for the calculation of flow in the draft tube by k-ε model. Later, Shyy cooperated with Vu [2] and YANG Jian-ming [3] et al. They improved the calculation conditions and some processing methods, and calculated the steady state flow field in the draft tube. LES (Large Eddy Simulation) is a new kind of numerical predication method established on the basis of turbulent statistics theory and the understanding of coherent structure. This method overcomes the shortcomings of turbulent model in time average and universal applicability. The LES method was firstly used to study the atmospheric motion law by atmospheric scientists Smogrins, Deardoff, et al. in 1970. Since then, this method has been gradually used in various fields. LES is especially suitable for the internal flow with complicated boundary shapes and anisotropic large size eddies, and it can be used to calculate the unsteady flow. Its application not only includes the fixed components in the flow system, but also includes the rotating or moving components [4].

The LES method which can be used to calculate the unsteady flow in the turbine was mainly analyzed in this paper. The geometric physical model of the whole flow passage of a type of Francis turbine was established. The vortex flow in the turbine was numerically simulated by using CFD software and FLUENT software. The flowing situations in the draft tube under three operating conditions of this turbine were successfully calculated. The PIV was used to measure the distribution of flow field at the draft tube cone section at 4 horizontal sections from the outlet of turbine runner. Specialized image analysis software was used to process the experimental results. The results of numerical simulation were in good coincidence with the experimental results. This proves that this researching method plays a decisive role in the optimal design of turbine, the predication of stable operating zone of the unit, the guaranteeing of stable operation of the unit, and the improvement of power quality.

### Nomenclature

- \( V \) flow velocity, m/s
- \( \bar{V} \) average velocity of \( V \), m/s
- \( \bar{P} \) average pressure of flow, Pa
- \( \nu \) kinematic viscosity coefficient of flow, m\(^2\)/s
- \( \tau \) sub-grid tension, N
- \( \nu_t \) sub-grid turbulent viscous force, N
- \( L_s \) mixing length of grid, m
- \( \kappa \) Von Karman constant
- \( C_s \) Samagorin constant
- \( d \) distance to the nearest wall, m
- \( \Omega \) volume of the calculating unit, m
- \( a_o \) opening of guide vane, mm
- \( n' \) unit speed of turbine, rpm
- \( Q' \) unit flow rate of turbine, L/s
2. Mathematical Model

Vortexes with large sizes play a dominant role in turbulent flow, while vortexes with small sizes are the main cause of turbulent momentum diffusion. In theory, the turbulent model from direct numerical simulation (DNS) can be used. However, due to its extra large calculation cost, it is not feasible in practical projects. For the traditional Reynolds average numerical simulation turbulent model (RANS, or k-ε model), its Reynolds average N-S equation is only the average number of transmission. Finding a feasible average flow variable will greatly reduce the workload of the computer. If the average flow is in steady state, its solution method will be more effective. The LES method is in between DNS and RANS, and this method can be used for vortexes with large sizes, and RANS equations can be used to solve for vortexes with small sizes. Due to the rapid development of computer technologies in recent years, the LES method has been widely used in engineering. The LES equations are given in the following, as well as the tension model on the grid and its boundary conditions.

2.1. LES equations

The LES equations are obtained through Fourier or space or N-S equations by eliminating the time. During calculation, vortexes smaller than the grid can be effectively eliminated, thus the momentum equation of large vortex can be obtained. This theory is mainly used for incompressible fluid. The following equations (or as LES equations) [4] can be obtained by eliminating the incompressible N-S equations:

\[
\frac{\partial}{\partial x_j} (\bar{V}_j) = 0 \tag{1}
\]

\[
\frac{\partial}{\partial t} (\bar{V}_j) + \frac{\partial}{\partial x_j} (\bar{V}_i \bar{V}_j) = - \frac{1}{\rho} \frac{\partial \bar{P}}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \nu \frac{\bar{V}_j}{\bar{x}_j} \right) - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}
\]

Where, \( \bar{V} \) represents the average velocity of \( V \), \( \nu \) represents the kinematic viscosity coefficient of flow, \( \tau_{ij} \) represents the sub-grid tension, which is defined as:

\[
\tau_{ij} = \bar{V}_i \bar{V}_j - \bar{V} \bar{V} \tag{3}
\]

The obtained sub-grid tension after elimination is unknown, so modeling is required. The most widely used vortex viscosity model equation at present is:

\[
\tau_{ij} = \frac{1}{3} \tau_{ik} \delta_{ik} = -2 \nu_t \bar{S}_y \tag{4}
\]

Where, \( \nu_t \) represents the sub-grid turbulent viscous force, \( \bar{S}_y \) represents the tensor vortex frequency, which is defined as:

\[
\bar{S}_y = \frac{1}{2} \left( \frac{\partial \bar{V}_j}{\partial x_i} + \frac{\partial \bar{V}_i}{\partial x_j} \right) \tag{5}
\]

The model about \( \nu_t \) was proposed by Samagorin and further improved by Lilly, and the equation of this model is:
\[ v_s = L_s^2 \left[ S \right] \]  

Where, \( L_s \) represents the mixing length of the grid, and \( \left[ S \right] = \sqrt{2S_x S_y} \), and the calculation equation of \( L_s \) is:

\[ L_s = \min \left( \kappa d, C_s \Omega^{1/3} \right) \]

Where, \( \kappa \) represents the Von Karman constant, \( C_s \) represents the Samagorin constant, \( d \) represents the distance to the nearest wall, and \( \Omega \) represents the volume of the calculating unit.

The value of \( C_s \) calculated by Lilly for similar turbulent flow in the inertia area is 0.23. However, this value will generate large damping vibration during the appearance of average shear force or in the flow field transition area. \( C_s = 0.1 \) is an ideal value for most flows.

2.2. Boundary conditions of LES model

According to the random perturbation theory, the velocity composition of flow at the boundary of specified velocity inlet can be expressed as:

\[ \vec{V}_i = \langle \vec{V}_i \rangle + \varPsi \| \vec{V} \| \]

Where, \( I \) represents the fluctuation strength, \( \varPsi \) represents the Gaussian random number, and is defined as \( \varPsi = 0 \) and \( \sqrt{\varPsi} = 1 \).

If the grid is well partitioned, the following wall shear force equation can be obtained from the relation between the thin wall stress and tension:

\[ \frac{\vec{V}}{V_r} = \frac{V_r y}{\nu} \]

If the grid is roughly partitioned, the flow situation on the thin wall cannot be solved. We can assume that the centroid of grid unit adjacent to the wall is in the convective region of the boundary layer, and its equation can be expressed as:

\[ \frac{\vec{V}}{V_r} = \frac{1}{\kappa} \ln \left( \frac{V_r y}{\nu} \right) \]

Constant \( E = 9.793 \).

3. Numerical Simulation of Vortex Flow in Draft Tube of Turbine

3.1. Establishment of geometrical physical model

A general geometric physical model of the flow passage of a Francis turbine (taken HL220-WJ-50 as example) was established in this paper, including the guide vane, runner and draft tube. Fig.1 shows the three main components and the grid map of the flow passage of the turbine.
3.2. Partition of grids and connection

The supporting GAMBIT drawing software of flow calculation software FLUENT was used to partition the grid for the flow passage. In view of the complicated geometric shape of the model, unstructured grids were used for meshing. The function of unstructured grid of FLUENT software made it possible to correctly simulate the complicated large scale geometric flow-around problem. The outlet edges of the guide vane and the runner blade were very thin, so the sizes of grids for this model were non-uniform. Therefore, the local densified grid technology was adopted to obtain thick grid distribution near the outlet edge of the blade, and grids of uniform specification in other areas. As shown in the grid geometric model in Fig.1, TGRID software was used to connect several flow passage components into a whole. In view of the rotation of the runner relative to the guide vane, the glide grid technology in FLUENT software was used to solve this problem.

During the grid partition process, it should be taken into consideration that static grids shall be adopted to the guide vane and draft tube, while rotational (gliding) grids shall be adopted to the runner. There should be independent grid nodes on the interface of these two kinds of grids, and the shape on the interface should be maintained consistent to guarantee the gliding. Therefore, the inlet plane of the runner and the outlet plane of the runner band were taken as the glide planes.

Due to the complicated structure of the flow passage components of the turbine, the completely unstructured tetrahedral grids were adopted. The outlet edge of the blade was densified before generating the entire grid automatically by the software. There were 169,508 solving units, including 27,695 for guide vane, 83,844 for runner, and 57,969 for draft tube.

3.3. Boundary and initial conditions

This research was mainly used to calculate the unsteady flow in the turbine. The model turbine of HL220-WJ-50 was taken as the example, which rated speed is 1000rpm and working head is 44m (the optimum operating head of this turbine). The effluent from the draft tube of this turbine was assumed to be submerged flow in the calculation process, and the water depth was 2m. Firstly, k-ε model was used to
calculate a steady initial value, and then the LES model was used to calculate the unsteady flow state. In this way, the calculation time by LES model can be greatly reduced, and the time step of this calculation was 0.0001s.

3.4. Numerical predication results

There were two indicators in the setting of calculation conditions for this research, which are unit flow and unit speed. The control of unit flow was realized by adjusting the opening of the guide vane, and different guide vane openings would generate different flows; and the control of unit speed was realized by changing the magnitude of working heads, and the magnitude of unit speed could be changed by setting different inlet pressures.

Under the small load, partial load and overload conditions, the velocity distribution, pressure distribution and flow streamline (the vortex of flow can be displayed, and the channel vortex and vortex strip in the draft tube can be analyzed), as well as the vortex strip pressure fluctuations in the draft tube were numerically simulated in this research.

![Velocity distribution of shaft section as predicted by CFD](image1)

![Vortex flow (vortex strip) in draft tube as predicted by CFD](image2)

![Vorticity and streamline diagram of section (2) as measured by PIV](image3)

![Vorticity and streamline diagram of section (4) as measured by PIV](image4)

Fig. 2. $a_0=18$mm, $n_i=95$rpm, $Q'_i=595$L/s, about 38% full load

The velocity distribution of flow field in the turbine and flow streamline in the draft tube under the
small load conditions \((a_0=18\text{mm}, n_1=95\text{rpm} \text{ and } Q_1'=595\text{L/s}, 38\% \text{ full load})\), partial load conditions \((a_0=18\text{mm}, n_1=55\text{rpm} \text{ and } Q_1'=695\text{L/s}, 55\% \text{ full load})\), and over load conditions \((a_0=42\text{mm}, n_1'=70\text{rpm}, Q_1'=1315\text{L/s}, 105\% \text{ full load})\) were given in the following, as shown in (a) and (b) in Fig.2, and 4. In this way, the flow vortexes in the draft tube under various conditions, namely draft tube vortex strip, could be seen.

The design unit flow under the optimum unit speed as recommended in the design manual is 1,140L/s, and the unit flow under the optimum condition is 1000L/s, and the full load refereed in this research was the design load.

(a) velocity distribution of shaft section as predicated by CFD; (b) vortex flow (vortex strip) in draft tube as predicated by CFD

(c) vorticity and streamline diagram of section (2) as measured by PIV; (d) vorticity and streamline diagram of section (4) as measured by PIV

\(\alpha_0=18\text{mm}, n_1'=55\text{rpm}, Q_1'=695\text{ L/s}, \text{ about 55\% full load}\)
Fig. 4. $\alpha_0=42\text{mm}, n_1=70\text{rpm}, Q_1=1315\text{L/s}, \text{about 105\% full load}$

3.5. Numerical predication results

3.5.1. Section layout and procedures of PIV test

According to the actual requirements of the research, we selected 4 representative horizontal sections at the taper pipe of the draft tube for PIV two-dimensional flow field tests under 3 different conditions.

Fig. 5. Layout diagram of PIV observation plane of draft tube cone
The specific distributions of 4 testing planes on the 4 horizontal sections at the taper pipe were shown in Fig.5.

1. The numerical image of flow field was obtained by the PIV system;
2. The image processing software Insight and Tecplot were used to calculate, analyze and process the experimental results and data;
3. The output velocity vector distribution diagram, streamline diagram, vorticity field diagram were used to show the actual flow situations of the flow field.

Due to the restrictions by the objective conditions of instrument layout, the pictures captured by PIV were taken from the bottom of the draft tube to the top. Therefore, the flow direction we saw was opposite to the direction shown in the CFD pictures. Due to the space constraints, only the images of the horizontal sections (2) and (4) as shown in Fig.5 were quoted.

4. Numerical predication results

Through the above CFD calculation results, we can summarize the situations of vortex strips in the taper pipe of the draft tube under several conditions:

In Fig.2: during 30% to 40% full load, the vortex strip was slightly eccentric in spiral shape with large helical angle. Most water flowed spirally downward along the wall of the draft tube in clockwise (looking from top to down). Compared with the numerical image of the flow field measured by the PIV system, we can see that the same result (the PIV footage was from down to top, and the direction was opposite to the results of CFD). The diameter of the vortex strip was large, and the flow in the center of the vortex strip was not regular. Analyzing from the section diagrams of (c) and (d) in Fig.2, we can see that the vortex strip was eccentric, and this was in good coincidence with the results of numerical simulation.

In Fig.3: during 55% full load, the vortex strip was seriously eccentric in spiral shape, and the central diameter of the vortex strip was obviously reduced. The water flowed spirally downward in clockwise (looking from top to down). Compared with the results of PIV test, the diameters of vortex strip in the section diagrams of (c) and (d) in Fig.4 were also very small, and the water flowed in clockwise and gradually expanded. The numerical simulation results were in good coincidence with the experimental results.

It was noteworthy that the distribution of vortex strip under overload conditions was different from all the above conditions, as shown in Fig.5. The calculated diameter of vortex strip was small in spiral shape, and flowed counterclockwise from top to down. This can be verified by comparing the section diagrams of (c) and (d) in Fig.4.

Through the calculation of velocity distribution diagram in the draft tube by CFD, we can find out that there were obvious vortex distributions in the draft tube under the small load conditions (38% to 55% full load). This was consistent with the research results in reference [5].

According to the above analysis and comparison, this numerical calculation method can be used to quantitatively study the vortex strip in the draft tube of turbine. It can be seen from this research that the calculated flow situations under many conditions could clearly show the velocity distribution and pressure distribution in the flow passage of the turbine, the vortex strip distribution in the draft tube, the eccentric of vortex strip, the magnitude of helical angle, and the vortex flow under small load. It also shows that vortex flow existed in the draft tube of turbine under all partial load conditions of the turbine, including the overload conditions. The strength of vortex flow was increasing with the distance of the conditions from the design point, especially the small load conditions. The rotation direction of flow was consistent with the rotation direction of the turbine. There were no vortex flows near the design conditions. However, the vortex flows would appear in the overload operating conditions, and the rotation direction of flow was opposite to the rotation direction of the turbine. These calculation results were in good coincidence with
the results of PIV test.

5. Conclusions

Starting from computation fluid dynamics, the LES method which can be used to calculate the unsteady flow in the turbine was analyzed in this paper. The geometric physical model of the whole flow passage of a Francis turbine was established. And FLUENT software was used to numerically simulate the flow (vortex flow) in the turbine. The calculation results were in good coincidence with the results of PIV test. Therefore, the prediction of unsteady flow in the turbine by the LES method was confirmed. These researches play a decisive role in the optimal design of turbine, the prediction of stable operating zone of the unit, the guaranteeing of stable operation of the unit, and the improvement of power quality.

5. Acknowledgements

This work was supported by the Open Research Funded Project of Provincial University Key Laboratory of Fluid Machinery in Xihua University (Grant No.: SZJJ2009-012), and partially supported by Scientific Research Fund of SiChuan Provincial Education Department (Grant No.: 11204026), meanwhile, this work was finished in the Provincial University Key Laboratory of Fluid Machinery in Xihua University.

References

[1] LIU Xiao-bing. Solid-liquid two-phase flow and numerical simulation in turbine machinery. Beijing: China WaterPower Press; 1995.12
[2] T. C. Vu and W. Shyy. Navier-Stokes flow analysis for hydraulic turbine draft tubes. ASME J. of Fluids Engineering; 1990, 112: 199-204
[3] YANG Jian-ming. Turbulent flow calculation through draft tube and runner of turbine, PhD thesis. Tsinghua University; 1999
[4] T. Bui, A parallel finite-volume algorithm for large-eddy simulation of turbulent flows, Computer & Fluids; 2000, 29: 877-915.
[5] LIU Xiao-bing. Theoretical study and numerical prediction of pressure fluctuation and hydraulic vibration of turbine. Research report of postdoctoral work, Sichuan University; 2004