

Comparative analysis of two-dimensional and three-dimensional simulation in rotor and stator cascades

Qice Song¹, Rennian Li^{1,2,*}, Wei Han^{1,2}, Rong Guo¹, Hao Chen¹

¹School of Energy and Power Engineering, Lanzhou University of Technology, Lanzhou 730050, China

²Key Laboratory of Fluid machinery and System, Gansu Province, Lanzhou 730050, China

*Corresponding author e-mail: lirn@lut.cn

Abstract. The eddy current problem in centrifugal pump has been a difficult problem, it is not only difficult to capture the vortex but also cause the calculation problem because of too many numbers of the grid. However, using two dimensional model which can greatly simplify the calculation and save computing resources. In order to study the differences between two dimensional model and three dimensional model simulation results, the vane type centrifugal pump is used as the research object and the numerical simulation method, computational fluid dynamics (CFD), is adopted in this paper. Calculated separately two dimensional (2D) and three dimensional (3D) model for calculation, two dimensional LES turbulence model is chosen and the three-dimensional LES model is adopted. In different working conditions, the two-dimensional and three-dimensional calculation of the vortices, pressure and velocity are compared and analysed, and the similarities and differences between the two dimensional model and three-dimensional model calculation results are obtained. By the above results, it is proved that the two-dimensional model can replace the three-dimensional model in a certain sense.

1. Introduction

The tandem cascade is the basic diffusion, diversion and working unit of multistage pump and other high-energy fluid machinery. The internal vortex structure is very complex that affected both by the impeller outlet "jet-wake" flow, dynamic and static blade rotor stator interaction. Numerical simulation and experimental study are the main methods for the analysis of the inner flow field of the centrifugal pump. The numerical simulation of the internal flow of the centrifugal pump in the present stage is mainly carried out from two-dimensional and three-dimensional simulation. Simulation model selection large eddy simulation (LES). Stephen B Pope [1] raised some fundamental questions concerning the conceptual foundations of LES and about the methodologies and protocols used in its application. J.P. Boris et al. [2] considers an approach to large eddy simulation (LES) using built-in sub grid turbulence models which appear naturally from the monotone computational fluid dynamics (CFO) algorithms used to simulate the resolved components of the flow. D. Boorish et al. [3] found that the two-dimensional large eddy simulation using a fine grid resolution, especially in the near wall region, gives a good representation of the quasi-two-dimensional mechanisms of the flow.



M.A. Lengthen et al. [4] concerned with the simulation of the flow in a flat, ‘two-dimensional’ laboratory centrifugal pump and concern of the study is the calculation of the flow-induced noise. QI Xue-yi et al. [5] used Gauss filter function to filter the N-S equation and the sub grid-scale Reynolds stress model is introduced to deduce the practical form of LES equation for 2-D flow calculation of hydraulic machine. WANG Le-Qin et al. [6] establish numerical method of solving the unsteady fluid flow around the starting or accelerating blade, numerical simulation on transient flow of a two dimensional centrifugal pump during its starting period was performed. Kudu A et al. [7] Compressible Navies–Stokes equations are solved using a fifth-order upwind scheme in the AUSM+ framework to visualize a compressible vortex ring generated from a shock tube, during vortex–wall interaction, some key experimentally observed features are identified on all the grids, but the details of the vertical structure look significantly different on different grids. In experiment, Feng J et al. [8-9], GAO Bo et al. [10], Yang J et al. [11] Using LDV, PIV and other equipment to process the internal flow field of centrifugal pump. In this paper, the two and three dimensional model of the vane type centrifugal pump is carried out, and the numerical calculation of the guide vane type centrifugal pump is carried out by using two dimensional large eddy simulation and three dimensional large eddy simulation.

2. Methodology

In order to study the internal flow of the guide vane type centrifugal pump, the large eddy simulation (LES) with Smagorinsky-Lilly sub grid model was used to simulate in two-dimensional and three-dimensional conditions. The method of Wall functions was used near wall, no slip condition is adopted in the impeller and guide vane.

In the Smagorinsky-Lilly model, the eddy-viscosity is modeled by

$$\mu_t = \rho L_s^2 |\bar{s}| \quad (1)$$

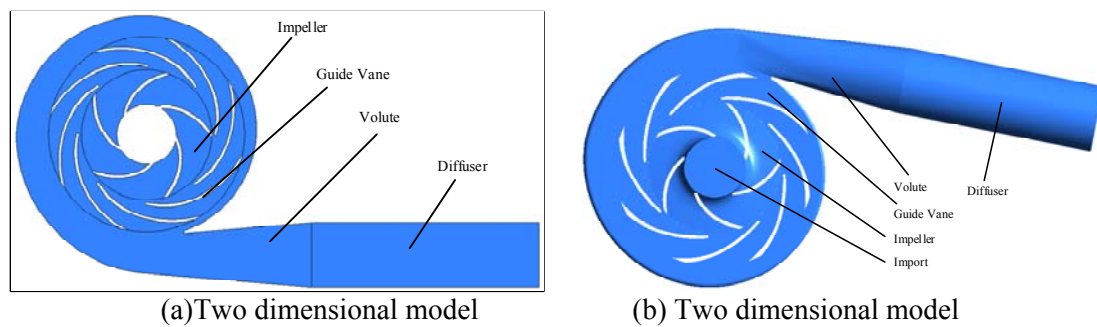
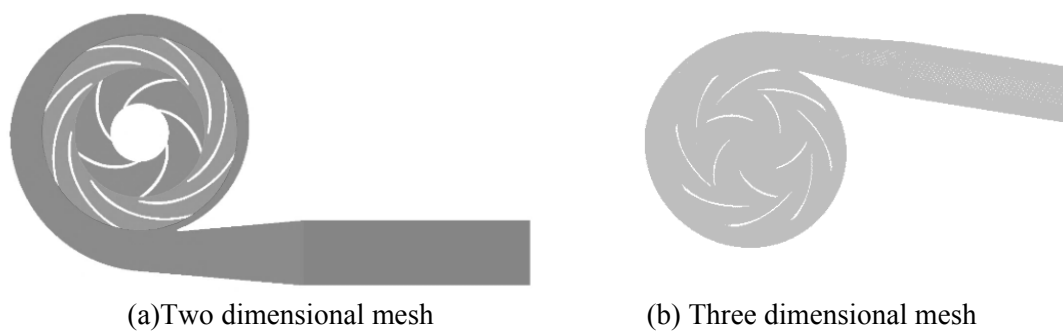
Where, L_s is the mixing length for sub grid scales and $|\bar{s}| \equiv \sqrt{2\bar{S}_{ij}\bar{S}_{ij}}$. In ANSYS Fluent, L_s is computed using $L_s = \min(\kappa d, C_s V^{\frac{1}{3}})$

Where, κ is the von Karman constant, d is the distance to the closest wall, C_s is the Smagorinsky constant, V is the local grid scale, and derived a value of 0.1 for C_s .

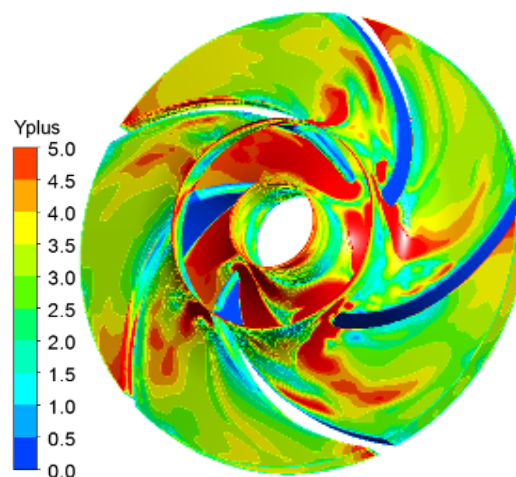
3. Calculation model

3.1. Model and Grid

Figure 1 displays the guide vane centrifugal pump of two dimensional and three dimensional model. Two dimensional model includes four parts, impeller, guide vane, spiral case and diffuser. Three dimensional model includes five parts, suction type, impeller, guide vane, spiral case and outlet section. The internal flow field was discrete with structured grid as Figure 2.

**Figure 1.** Vane type centrifugal pump model**Figure 2.** Simulation mesh

In order to make the research of the flow characteristics and the effect of particles on the flow in the pump to be more accurately, large eddy simulation calculation is selected. The structural mesh is generated in ICEM CFD. The mesh of 2D used in the simulation was finally divided into 3066352 nodes, and the mesh of 3D used in the simulation was finally divided into 12563053 nodes. Y+ distribution in 3D impeller of vane type centrifugal pump as Figure 3.

**Figure 3.** Y+ distribution

Some geometrical parameters are listed in Table 1, and the details mesh node number is in Table 2.

Table 1. Model Parameter

Item	Value
Number of impeller blade	5
Number of guide vane blade	8
Impeller outlet diameter/mm	161
Guide vane outlet diameter/mm	242
Pump outlet diameter/mm	160

Table 2. Mesh Parameter

Item	Value
Nodes of inlet	207152(3D)
Nodes of impeller	2446305(2D) 5793185(3D)
Nodes of guide vane	166281(2D) 4806271(3D)
Nodes of volute	362560(2D) 1549293(3D)
Nodes of diffuser	91206(2D) 207152(3D)
Y plus of impeller boundary	1(2D) 5(3D)
Number of boundary layers	20

3.2. Boundary Condition

In this paper, the numerical simulation of 2D and 3D vane type centrifugal pump are carried out under the design conditions. The simulation was conducted with ANSYS CFD, using the large eddy simulation (LES) and Smagorinsky-Lilly sub grid model, calculation of solid-liquid two phase calculation using Mixture model. Under the design condition, the flow rate of vane type centrifugal pump is 22.85m³/h. Import selection Velocity-inlet, export choice outflow.

4. Result

4.1. Performance curve of pump

Performance curves of the 2D and 3D numerical calculation of the vane type centrifugal pump as Figure 4, the pump flow head and efficiency curve in the state of water and solid- liquid two phase flow are compared in the figure. It can be seen the performance vary curves of 3D model have the same changing tendency with the experimental, the head decreased with the increase of flow, the efficiency increases firstly and then decreases with the increase of the flow rate and it reached the maximum under the design condition. However, head of 2D is much higher than the experimental. The above phenomenon is caused by the difference of the structure between the 2D and the 3D model.

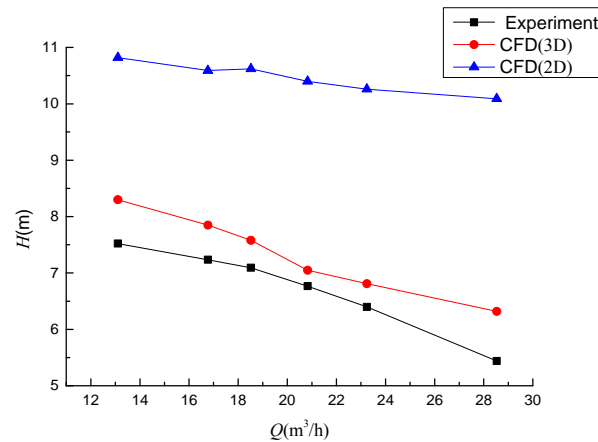


Figure 4. Performance curve of pump

4.2. Vortex Analysis

The vortices distribution in the impellers and the guide vanes of 2D and 3D under the condition of the water in Figure 5. It can be seen that the vortices concentration region is the back of the blade and the excessive part of the impellers and the guide vanes. The large eddy area of 2D is significantly higher than that of the 3D, and more trailing vortex appear in the transition part of impellers and guide vanes. The reason for these different results is that the 2D model is a simplification of the 3D model, which cannot fully reflect the flow of the real pump and some results of 2D calculation are ignored by the 3D calculation. In general, the vortices concentration distribution area of 2D model and 3D model is similar to each other, 2D simulation can be used as a reference for 3D simulation under pure water, and the flow disturbance area can be figured out.

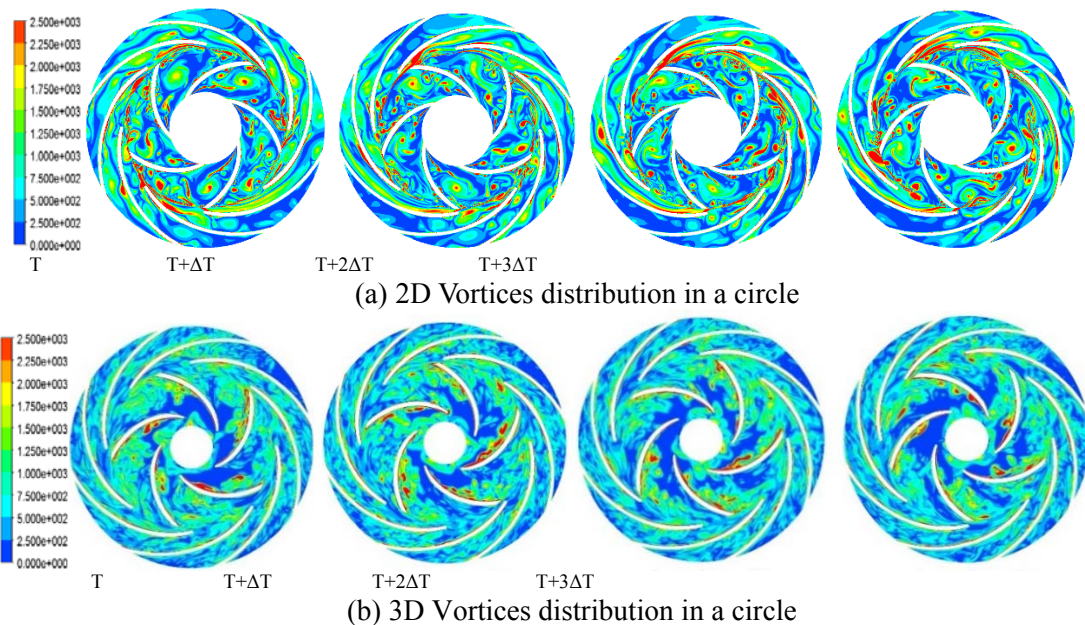


Figure 5. Vortices distribution

The vortices distribution in the impellers and the guide vanes of 2D and 3D under the condition of solid-liquid two phase in Figure 6. It can be found from the two-dimensional calculation that the addition of particles significantly changed the vortex distribution of the impeller and guide vane. By

analysis of the particle volume fraction and vortices distribution, it is found that the large vortices is larger in the area with smaller particle size fraction, and the same results can be obtained with the three-dimensional calculation.

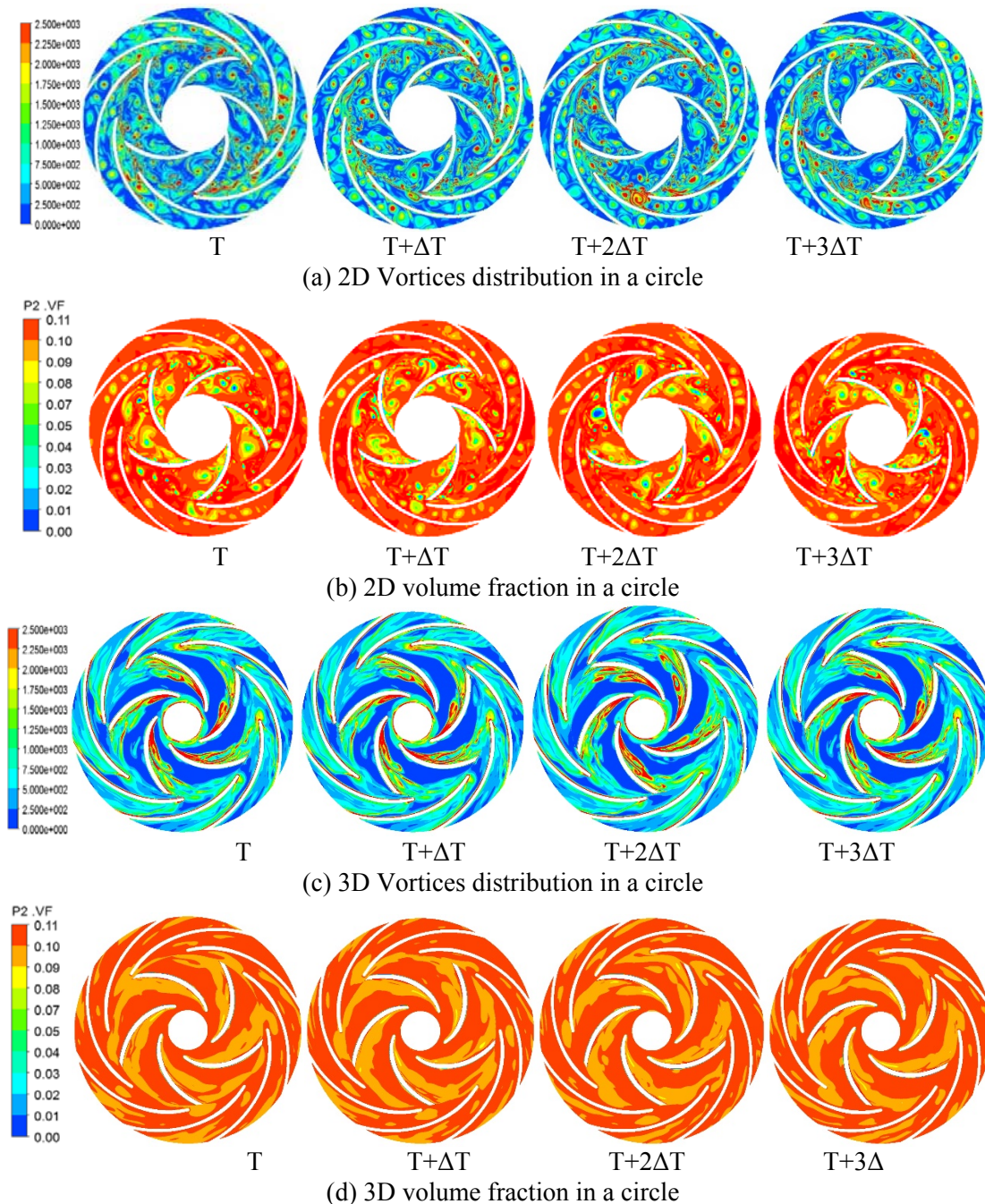


Figure 6. Vortices distribution and volume fraction

5. Conclusion

In this paper, by comparing the two-dimensional and three-dimensional numerical calculation, we get the vortex distribution of the vane type centrifugal pump under the conditions of water and solid-liquid two phases and sum up the relationship between the two-dimensional and three-dimensional numerical

simulation. Some conclusions were obtained as follow according to above calculation and analysis results:

(1) The vortices concentration region is the back of the blade and the excessive part of the impellers and the guide vanes. The large eddy area of 2D is significantly higher than that of the 3D, and more trailing vortex appear in the transition part of impellers and guide vanes.

(2) The addition of particles significantly changed the vortex distribution of the impeller and guide vane, the large vortices is larger in the area with smaller particle size fraction, and the same results can be obtained with the three-dimensional calculation.

(3) For the vane type centrifugal pump, 2D numerical calculation can be to a certain extent the flow of the centrifugal pump. The optimization of the centrifugal pump design and flow analysis has played a guiding role, it can also predict the results of three-dimensional numerical calculations. But for the twisted blades and other complex structure of the centrifugal pump, still need to be further analyzed.

Acknowledgments

This work was financially supported by the National Natural Science Foundation of China (51579125, 51669012).

References

- [1] Pope S B. Ten questions concerning the large-eddy simulation of turbulent flows [J]. New journal of Physics, 2004, 6(1): 35-58.
- [2] Boris J P, Grinstein F F, Oran E S, et al. New insights into large eddy simulation [J]. Nasa Sti/recon Technical Report N, 1992, 10(4-6):199-228.
- [3] Bouris D, Bergeles G. 2D LES of vortex shedding from a square cylinder [J]. Journal of Wind Engineering & Industrial Aerodynamics, 1999, 80(1):31-46.
- [4] Langthjem M A, Olhoff N. A numerical study of flow-induced noise in a two-dimensional centrifugal pump. Part I. Hydrodynamics[J]. Journal of Fluids & Structures, 2004, 19(3):349-368.
- [5] QI Xueyi, LIU Zailun, QI Chong. Application of 2-D large eddy simulation to flows in impeller of double-flow-conduits-sewage pump [J]. Chinese Journal of Hydrodynamics, 2003, 18(3):289-293.
- [6] WANG Leqin, LI Zhifeng, DAI Weiping. 2-D NUMERICAL SIMULATION ON TRANSIENT FLOW IN CENTRIFUGAL PUMP DURING STARTING PERIOD[J]. Journal of Engineering Thermophysics, 2008, 29(8):1319-1322.
- [7] Kundu A, De S, Thangadurai M, et al. Numerical visualization of shock tube-generated vortex-wall interaction using a fifth-order upwind scheme[J]. Journal of Visualization, 2016: 1-12.
- [8] Feng J, Benra F K, Dohmen H J. Application of Different Turbulence Models in Unsteady Flow Simulations of a Radial Diffuser Pump[J]. Forschung Auf Dem Gebiete Des Ingenieurwesens, 2010, 74(3):123-133.
- [9] Feng J, Benra F K, Dohmen H J. Unsteady flow visualization at part-load conditions of a radial diffuser pump: By PIV and CFD [J]. Journal of Visualization, 2009, 12(1):65-72.
- [10] GAO Bo, YANG, et al. Experimental Research on Salt-out Particle Motion and Concentration Distribution in a Vortex Pump Volute[J]. Chinese Journal of Mechanical Engineering, 2010, 23(1):53-59.
- [11] Yang J, Yuan S, Yuan J, et al. Numerical and Experimental Study on Flow-induced Noise at Blade-passing Frequency in Centrifugal Pumps[J]. Chinese Journal of Mechanical Engineering, 2014, 27(3):606-614.