

PAPER • OPEN ACCESS

Effects of Different Turbulence Models on Simulation Accuracy of the Miniature Super-low Specific Speed Centrifugal Pump

To cite this article: Jin Zhou *et al* 2019 *IOP Conf. Ser.: Mater. Sci. Eng.* **490** 052026

View the [article online](#) for updates and enhancements.



IOP | ebooks™

Bringing you innovative digital publishing with leading voices to create your essential collection of books in STEM research.

Start exploring the collection - download the first chapter of every title for free.

Effects of Different Turbulence Models on Simulation Accuracy of the Miniature Super-low Specific Speed Centrifugal Pump

Jin Zhou¹, Gailan Xing¹ and Shaoping Zhou^{1,*}

¹School of Mechanical and Power Engineering, East China University of Science and Technology, Shanghai, China

*Corresponding author e-mail: shpzhou@ecust.edu.cn

Abstract. In order to study the effects of different turbulence models on simulation accuracy of the miniature super-low specific speed centrifugal pump, the RNG $k-\varepsilon$ turbulence model and SST $k-\omega$ turbulence model had been used in numerical simulation by Fluent. The performance curves and the internal flow field distribution of those two turbulence models had been analysed and compared with the experiment results. The results show that at the low flow rate conditions, the simulation accuracy of the SST $k-\omega$ model is lower than that of the RNG $k-\varepsilon$ model, while at the high flow rate conditions, the result is just the opposite. In the whole operating conditions, the average simulation error of the SST $k-\omega$ model is 3.5% and that of the RNG $k-\varepsilon$ model is 4.3%.

1. Introduction

With the development of computational fluid dynamics, performance prediction and flow field analysis based on numerical simulation have gradually become an important mean of pump design and optimization [1-3]. For the mixture super-low specific speed centrifugal pump, due to the boundary layer separation, the secondary flow and the jet-wake structure in the internal flow field, turbulence models have important effects on the simulation accuracy [4,5]. At present, $k-\varepsilon$ model and $k-\omega$ model are widely used, due to their strong applicability in different operating conditions. Shu's research shows that in pump numerical simulation, the applicability of RNG $k-\varepsilon$ model and SST $k-\omega$ model are related to the specific speed [6]. Ren's research shows that the simulation accuracy of SST $k-\omega$ model is higher than that of RNG $k-\varepsilon$ model at off-design conditions [7]. In the study of the applicability of $k-\varepsilon$ model, Zhang's research shows that the realizable $k-\varepsilon$ model is better than RNG $k-\varepsilon$ model, but Hu's research has reached the opposite conclusion [8,9]. In summary, although many scholars have studied the applicability of turbulence models in pump numerical simulation, there are no clear conclusions. In fact, in pump numerical simulation, the suitable turbulence models often depend on the pump type, the complexity of internal flow field, the required precision and many other factors.

In this paper, a miniature super-low specific speed centrifugal pump had been studied. The effects of the RNG $k-\varepsilon$ turbulence model and SST $k-\omega$ turbulence model on the simulation accuracy of this type of pump had been analyzed, from the view of the performance curve and the internal flow field distribution, and the results were compared with the experiment results, which will provide guide for the numerical simulation of this type of pumps.



2. Computational model

2.1. Research object

In this paper, the research object is a miniature super-low specific speed centrifugal pump. The main model parameters are as follows: the design flow rate is 0.36 m³/h, the design head is 9 m, the rotational speed is 2950 r/min, and the specific speed is 21. The main parameters and sketch of pump structure are shown in Table 1 and Figure 1.

Table 1. Main parameters of pump structure

D ₂ /mm	D ₁ /mm	D _j /mm	b ₂ /mm	S ₈ /mm ²	β ₁ (°)	β ₂ (°)	φ/(°)	d _h /mm	z
80	17.1	19	1.5	16	24	35	160	14	5

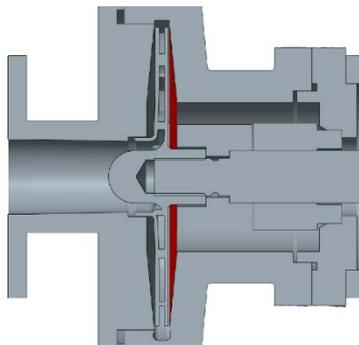


Figure 1. Sketch of pump structure

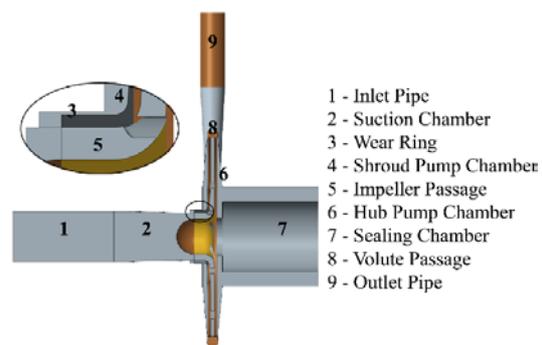


Figure 2. Computational domain

2.2. Computational domain

The computational domain includes impeller passage, volute passage, pump chamber, sealing chamber, wear ring, suction chamber, inlet pipe and outlet pipe. The pump chamber is connected with the volute passage and the shroud chamber is connected with the suction chamber through the wear ring, as shown in Figure 2.

3. Numerical simulation

3.1. Governing equations

Both the RNG k - ε model and SST k - ω model are used to closed Reynolds-Averaged Navier-Stokes equations. The governing equations are as follows:

The continuity equation

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i) = 0 \quad (1)$$

The momentum equation

$$\frac{\partial}{\partial t}(\rho u_i) + \frac{\partial}{\partial x_j}(\rho u_i u_j) = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial u_i}{\partial x_j} - \rho \overline{u_i u_j} \right) + S_i \quad (2)$$

According to the Boussinesq assumption [10], the Reynolds stress can be expressed by an average velocity gradient, which is as follows:

$$-\rho \overline{u_i u_j} = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \left(\rho k + \mu_t \frac{\partial u_i}{\partial x_i} \right) \delta_{ij} \quad (3)$$

In this assumption, the key to solve the above equations is to introduce a suitable turbulence model to solve the turbulent viscosity μ_t .

3.2. Turbulence models

In the Standard k - ε model, the turbulent viscosity μ_t is expressed as a function of the turbulent kinetic energy k and the turbulent dissipation rate ε , which is as follows:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (4)$$

Based on the Standard $k-\varepsilon$ model, the RNG $k-\varepsilon$ model considers the swirl flow effect in the mean flow by the renormalization group statistical technique, and add an average strain rate term into the ε equation. Therefore, the RNG $k-\varepsilon$ model can deal with the complex flow field in the impellers. Similarly, the SST $k-\omega$ model replaces the turbulent dissipation rate ε with the specific dissipation rate ω , which not only improves the simulation accuracy in the near wall region, but also avoids the turbulent transport being overestimated. Therefore, the SST $k-\omega$ model can effectively predict the position of flow separation and the size of the separation zone in the impeller at the condition of inverse pressure gradient.

3.3. Boundary conditions and discrete algorithm

In this paper, the Fluent steady solver was used to solve the internal flow field. The rotating motion of impeller was simulated by using multi-reference frame method, that is to say, the impeller passage was set as a rotating region, and the rest were set as static region. In the near wall region, the scalable wall function was used in the RNG $k-\varepsilon$ model, and the y^+ value was kept below 60 in the SST $k-\omega$ model. All the walls were set as no-slip rough wall, and the roughness is set to $6.3\mu\text{m}$. The pressure inlet and mass-flow outlet boundary condition were used, and the specific parameters depend on the operating condition.

In the impellers of super-low specific speed centrifugal pumps, the strong inverse pressure gradient, boundary layer separation and secondary flow will induce oscillation or divergence. In order to improve the solution stability, the SIMPLE algorithm was adopted. In this paper, the pressure under-relaxation factor was set to 0.2, and the velocity under-relaxation factor was set to 0.1. The convection term was set to second order upwind scheme and the diffusion term was set to central difference scheme.

3.4. Grid generation and independence validation

In order to improve the simulation accuracy and simulation efficiency to capture more turbulent flow characteristics, computational domain adopted structural grid. The grid of the tongue, the pump chamber, the wear ring and the near wall region should be refined to improve the resolution of flow field. The grid of computational domain is shown in Figure 3.

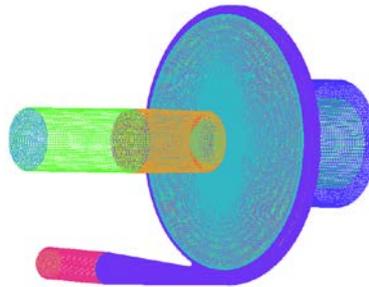


Figure 3. Grid of computational domain

The number of grid affects the simulation accuracy and simulation efficiency. In order to improve simulation efficiency, it is necessary to control the number of grid on the basis of sufficient accuracy. The change of the head simulation values with the number of grid is shown in Table 2. According to Table 2, with the increase of the number of grid, the relative errors of the head simulation values are less than 0.5%, which satisfies the requirement of grid independence, so the number of grid should be kept about 3.0×10^6 .

Table 2. Grid independence validation

No.	Grid numbers $/\times 10^6$	H/m	Relative error / %
1	3.0	8.93	/
2	3.7	8.92	0.11
3	4.4	8.90	0.22

4. Results and discussions

4.1. Analysis of performance curve

Figure 4 shows the comparison between the head simulation curves and head experiment curve. Figure 5 shows the simulation error of those two turbulence models. Figure 6 shows the efficiency simulation curves of those two turbulence models.

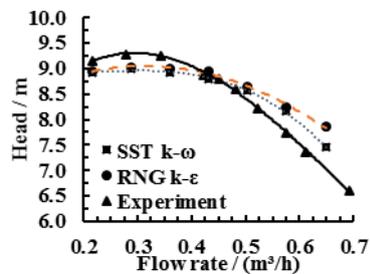


Figure 4. Head curves

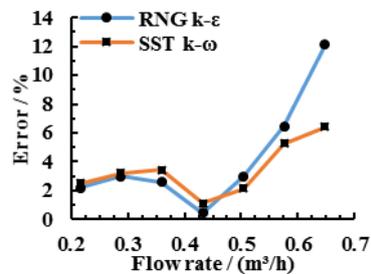


Figure 5. Simulation error

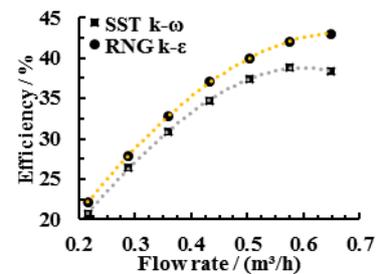


Figure 6. Efficiency curves

As shown in Fig.4, the head simulation curves of those two turbulence models are in good agreement with the experiment curve. At the low flow rate conditions, the simulation values are smaller than the experiment values, while at the high flow rate conditions, the simulation values are larger than the experiment values. At the condition of 0.44 m³/h (1.2 times the design flow rate), the simulation values are basically equal to the experiment values. Both RNG $k-\epsilon$ model and SST $k-\omega$ model can predict the hump phenomenon of the head curve. However, In the whole operating conditions, the head simulation values of the RNG $k-\epsilon$ model are larger than that of the SST $k-\omega$ model.

As shown in Fig.5, for the SST $k-\omega$ model and the RNG $k-\epsilon$ model, with the increase of flow rate, the change trend of the simulation error is coincident. At the low flow rate conditions, the simulation error of the SST $k-\omega$ model is larger than that of the RNG $k-\epsilon$ model, while at the high flow rate conditions, the simulation error of the SST $k-\omega$ model is smaller than that of the RNG $k-\epsilon$ model. Furthermore, with the increase of flow rate, the simulation error of the RNG $k-\epsilon$ model increases rapidly. In the whole operating conditions, the average simulation error of the SST $k-\omega$ model is about 3.5%, while that of the RNG $k-\epsilon$ model is about 4.3%.

As shown in Fig.6, in the whole operating conditions, the efficiency simulation values of the SST $k-\omega$ model are smaller than that of the RNG $k-\epsilon$ model. With the increase of flow rate, the difference between the simulation efficiency values of the SST $k-\omega$ model and the RNG $k-\epsilon$ model is gradually increasing.

4.2. Analysis of internal flow field distribution

In order to further analyze the difference between the RNG $k-\epsilon$ model and the SST $k-\omega$ model, the internal flow field distribution of two turbulence models are analyzed by taking the operating conditions of 0.36 m³/h (the design flow rate) and 0.504 m³/h (1.4 times the design flow rate) as examples.

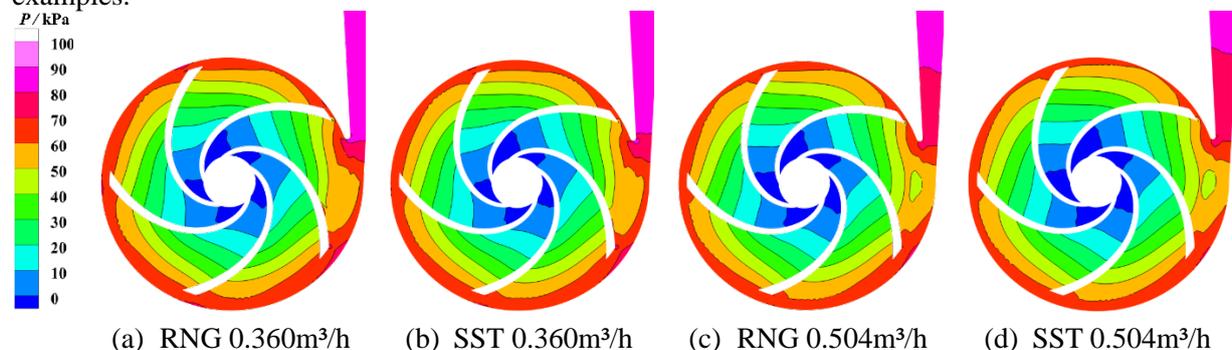


Figure 7. Static pressure distribution on middle section

Fig.7 shows the static pressure distribution on the middle section of the impeller. The static pressure distribution solved by different turbulence models is basically coincident. In the impeller passage, the static pressure increases gradually along the radius direction, and there are negative pressure regions at the impeller inlet. With the increase of flow rate, the static pressure distribution in the impeller passage is unchanged, but the mean static pressure in the volute diffuser increases obviously. At the operating of $0.504 \text{ m}^3/\text{h}$, there is an isolated low pressure region at the eighth section of the volute. The mean static pressure in the volute diffuser solved by the RNG $k-\varepsilon$ model is higher than that solved by the SST $k-\omega$ model, which results in the head simulation values of former being higher than that of the latter.

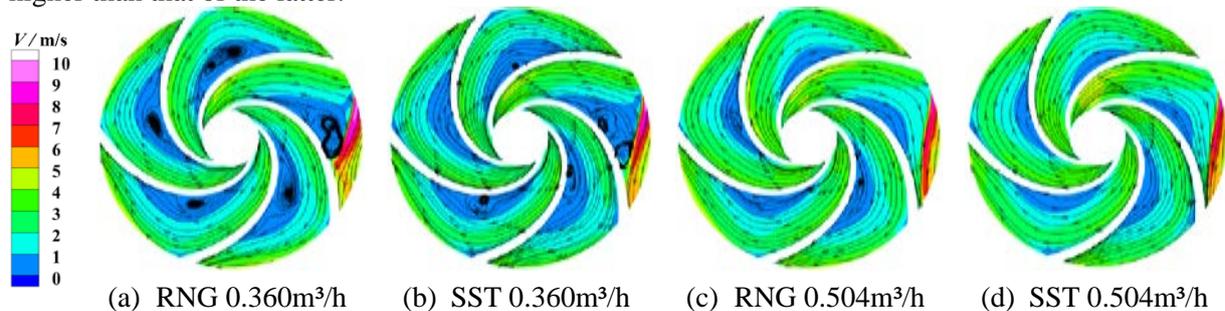


Figure 8. Relative velocity distribution on middle section

Fig.8 shows the relative velocity distribution on the middle section of impeller. There is an obvious low velocity region on the blade pressure surface in each impeller passage, and the area of this region will decrease with the increase of flow rate. At the impeller outlet, along the circumference direction, from the blade pressure surface to the blade suction surface, the relative velocity decreases gradually, which named the jet-wake structure and will cause hydraulic loss. The curvature of streamline changes suddenly at the impeller outlet, due to the blades losing control of the fluid, and it is one of the causes of the jet-wake structure. However, in the low velocity region, the phenomena of boundary layer separation solve by the RNG $k-\varepsilon$ model and the SST $k-\omega$ model are different. The strength and scale of separation vortex solved by the RNG $k-\varepsilon$ model is obviously larger than that of the SST $k-\omega$ model.

5. Conclusion

In this paper, a miniature super-low specific speed centrifugal pump had been studied by Fluent, based on the RNG $k-\varepsilon$ model and the SST $k-\omega$ model. The conclusions are as follows:

(1) For miniature super-low specific speed centrifugal pump, at the low flow rate conditions, the simulation accuracy of the SST $k-\omega$ model is lower than that of the RNG $k-\varepsilon$ model, while at the high flow rate conditions, the result is just the opposite. In the whole operating conditions, the average simulation error of the SST $k-\omega$ model is 3.5% and that of the RNG $k-\varepsilon$ model is 4.3%.

(2) In the volute diffuser, the average static pressure solved by the RNG $k-\varepsilon$ model is higher than that by the SST $k-\omega$ model, which make the head simulation values of the former higher than that of the latter. On the blade pressure surface, the strength and scale of boundary layer separation solved by the RNG $k-\varepsilon$ model are larger than that of the SST $k-\omega$ model.

References

- [1] Hamed Alemi, Seyyed Ahmad Nourbakhsh, Mehrdad Raisee, Effect of the volute tongue profile on the performance of a low specific speed centrifugal pump, Proceeding of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy. 229 (2014) 210-220.
- [2] Gao Bo, Zhang Ning, Li Zhong, Influence of the Blade Trailing Edge Profile on the Performance and Unsteady Pressure Pulsations in a low Specific speed Centrifugal Pump, Journal of fluids Engineering, Transactions of the ASME. 138 (2016).
- [3] Pei Ji, Wang Wenjie, Yuan shouqi, Optimization on the Impeller of a Low-specific-speed Centrifugal Pump for Hydraulic Performance Improvement, Chinese Journal of Mechanical Engineering, 29 (2016) 992-1002.

- [4] Shao Chunlei, Gu Boqin, Huang Xinglu, Experimental study on flow in the impeller of low specific speed pump using particle image velocimetry, *Journal of Aerospace Power*. 25 (2010) 2091-2096.
- [5] Wang Fujun, Research Progress of Computational Model for Rotating Turbulent Flow in Fluid Machinery, *Transactions of the Chinese Society for Agricultural Machinery*. 47 (2016) 1-14.
- [6] Shu Minhua, Liu Houlin, Tan Minggao, Research on the Application of Turbulence Models to the Design of Centrifugal Pumps, *China Rural Water and Hydropower*. 40 (2011) 18-22+74.
- [7] Ren Yun, Liu Houlin, Shu Minhua, Analysis of Applicability of Turbulence Models in Performance Prediction for Centrifugal Pumps at the Off-design Conditions, *Fluid Machinery*. 40 (2012) 18-22+74.
- [8] Zhang Shujia, Li xianhua, Zhu baolin, Application of k- ϵ Eddy viscosity Turbulence Models on Numerical Simulation of Centrifugal Pump, *Journal of Mechanical Engineering*. 45 (2009) 238-242.
- [9] Hu Jingning, Jiang Wei, Liu Sanhua, Influence of Various Turbulent Flow Models on the Precision of Simulating a Seawater Desalination-purpose Lift Pump, *Journal of Engineering for Thermal Energy & Power*. 26 (2011) 483-486.
- [10] J.O. Hinze, *Turbulence*. McGraw-Hill, New York, 1975.