IOP Conference Series: Earth and Environmental Science

**OPEN ACCESS**

# Effects of computational grids and turbulence models on numerical simulation of centrifugal pump with CFD

To cite this article: H L Liu *et al* 2012 *IOP Conf. Ser.: Earth Environ. Sci.* **15** 062005

IOP Conf. Series: Earth and Environmental Science **15** (2012) 062005 doi:10.1088/1755-1315/15/6/062005

# Effects of computational grids and turbulence models on numerical simulation of centrifugal pump with CFD

**H L Liu, M M Liu, L Dong, Y Ren and H Du**
Research Center of Fluid Machinery Engineering and Technology, Jiangsu University, 301 xuefu Road, Zhenjiang, 212013, Jiangsu, China

E-mail: liuhoulin@ujs.edu.cn

**Abstract.** In order to verify accurately the effects of computational grids and turbulence models on CFD numerical simulation of centrifugal pump, the calculation results of the different mesh numbers coupling with five kinds of turbulence models are compared. These parts of models (e.g. wear ring and tongue) are meshed with the local refinement technology. At design condition, with CFX software on the Dawning TC3600 parallel computer cluster, the calculation results of phase coupling six different mesh numbers from 1 million to 25 million with five different turbulence models (K-Epsilon，SSG Reynolds Stress, K epsilon EARSM, RNG K-Epsilon, K-Omega)are used to make performance prediction and to analyze the internal flow field. And the comparison between performance prediction results and experimental results are analyzed. It is confirmed that the calculation results are quite different with different turbulence models, and the result under K epsilon EARSM model is better than the others. And it shows that the internal flow in centrifugal pump is depicted more perfect with the increase of mesh numbers.

## 1. Introduction

With the improvement of Computational Fluid Dynamics (CFD) and the rapid development of hardware in computer, some commercial CFD software has been used widely [1], such as Fluent, CFX, etc. As an important part of CFD technique, turbulence model has a directly influence on stability of numerical simulation and accuracy of calculation results [2].For the complicate structure of centrifugal pump, the flow in centrifugal pump is extremely complicate three dimensional turbulence flow, and an omnipotent turbulence model has not been discovered at present. Therefore, researching about how turbulence models influence CFD numerical simulation is still an important issue and it is worth paying attention to [3-5].

Recently, when turbulence model is considered to have a certain impact on calculation results of numerical simulation in centrifugal pump, turbulence models influence on numerical simulation under different conditions and with different specific speed are studied [6, 7] .Meanwhile, mesh numbers influence on performance of centrifugal pump is rarely considered, and it has yet to see relevant research about effects of phase coupling different mesh numbers with different turbulence models in numerical simulation.

The paper couples six different mesh numbers from 1 million to 25 million with five different turbulence models by CFX Solver for having numerical simulation on parallel cluster. And through comparative analysis both in energy performance and in internal flow field, a suitable turbulence model collocation is expected to be proposed. On that basis, the mesh numbers influence on performance of centrifugal pump can be obtained.

## 2. Numerical model

### 2.1. Computation model

A centrifugal pump with specific speed at 44.8 is studied. its characteristics parameters and geometry parameters are as follows: $Q$=20.31m$^3$/h, $H_e$=46.35m, $n$=2900r/min, $\eta_e$=65.44%, $D_2$=0.196m, $b_2$=0.018m, $\beta_2$=32$^O$,$D_3$=0.202m, $b_3$=0.018m. The 3D modeling of impeller, volute, suction, outlet extending segment, cavity (including shroud, hub and ring) are implemented separately by Pro/ENGNEER software. Then they are assembled to establish the whole flow field. Figure 1 is computation domain.

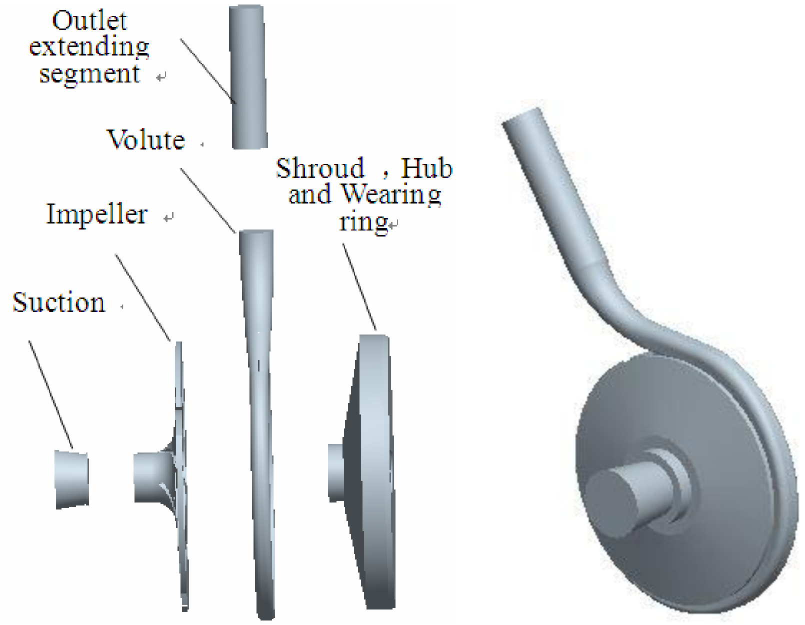



**Figure 1.** Computation domain

**Table 1.** Computation domain mesh for model

| | 1 | 2 | 3 | 4 | 5 | 6 |
|---|---|---|---|---|---|---|
| Suction | 55326 | 55326 | 55326 | 932212 | 9322112 | 932212 |
| Impeller | 254326 | 632398 | 1305652 | 2532664 | 5052634 | 7243156 |
| Cavity | 443215 | 502314 | 1205487 | 2065413 | 5965243 | 7852364 |
| Volute | 352678 | 678642 | 1354623 | 2132546 | 4142352 | 6982513 |
| Outlet extending segment | 65142 | 65142 | 65142 | 872614 | 872614 | 872614 |
| Total mesh number | 1170687 | 1933822 | 3986230 | 8535449 | 16965055 | 23882859 |

### 2.2. Meshing

Impeller, volute, suction, outlet extending segment, cavity are meshed by ICEM separately. For some thin domain (ring, tongue, etc.), it’s hard to generate mesh successfully as a whole at only one time. Therefore, local refinement technique needs to be implemented at such positions. In order to verify the effects of computational grids and turbulence models on CFD numerical simulation of centrifugal pump, six groups of mesh numbers range from 1million to 24 million is meshed. The results of mesh generation are shown in table 1.

### 2.3. Turbulence model, boundary conditions and operating environment

Five turbulence models used in this paper are K-Epsilon, SSG Reynolds Stress, K epsilon EARSM, RNG K-Epsilon, K-Omega respectively. The inlet boundary condition is velocity inlet. It is assumed that the inlet velocity $u_{in}$ is uniform at the axial direction and the radial and tangential components are zero. The value of $k$ and $\varepsilon$ at the inlet boundary can be estimated with the following approximate formula [8]:

$$k_{in} = 0.005u_{in}^{2}, \varepsilon_{in} = C_{\mu}^{3/4} k_{in}^{3/2} / l \tag{1}$$

where, $l$ denotes characteristic length of the inlet.

The volute outlet was extended properly to reduce the effect of boundary conditions on inner flow. Outlet boundary condition is "outflow" and flow rate weighting is set to be 1.

As for wall boundary condition, no slip condition is used on wall surface and standard wall functions are applied to adjacent region.

For the larger mesh number, the simulation is implemented on the Dawning TC3600 parallel computer cluster, whose parameters are as follows: the server is blade server, the CPU is Intel Xeon, the hard disk is SAS/SATA, and the maximum number of CPU is 12.

Performance formulas of centrifugal pump based on calculation result from numerical simulation are given by [9, 10]:

$$H = \frac{P_{out} - P_{in}}{\rho g} \tag{2}$$

where, $H$ denotes head of centrifugal pump, $P_{in}$ denotes total pressure at inlet of impeller. $P_{out}$ denotes total pressure at outlet of volute.

$$\eta_h = \frac{\rho g Q H}{M\omega} \tag{3}$$

where, $M$ denotes torques including press side and suction side of impeller blade, internal and external surface of black shroud and front shroud.

$$\eta = \eta_h \times (1-3\%) = \frac{\rho g Q H}{M\omega} \times (1-3\%) \tag{4}$$

where, $\eta_h$ denotes the total efficiency including volume loss and disk friction. Loss of bear and seal is set to be 3%, which is the same with reference [10].

### 2.4. Turbulence models influence on energy performance

The relative discrepancy of head is calculated as follow:

$$\Delta H = \frac{(H - H_e)}{H_e} \times 100\% \tag{5}$$

The relative discrepancy of efficiency is calculated as follow：

$$\Delta\eta = \frac{\eta - \eta_e}{\eta_e} \times 100\% \tag{6}$$

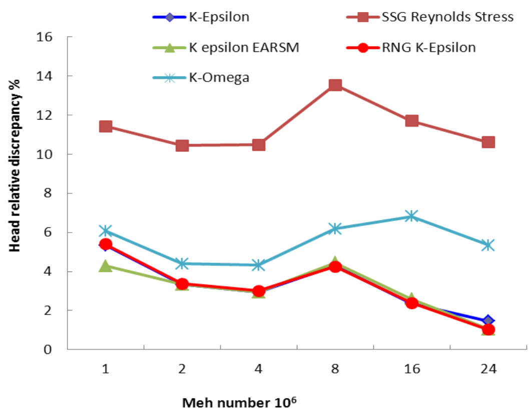



**Figure 2.** Head relative discrepancy at design condition.

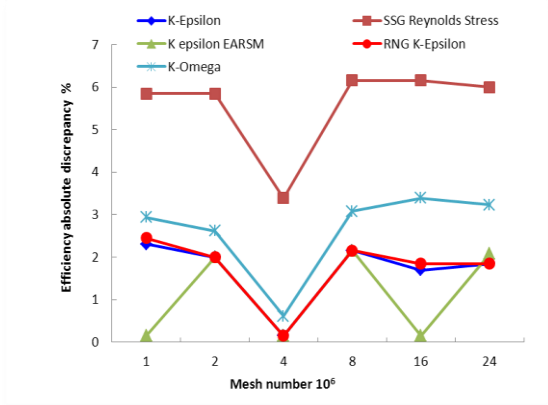



**Figure 3.** Efficiency absolute discrepancy at design condition

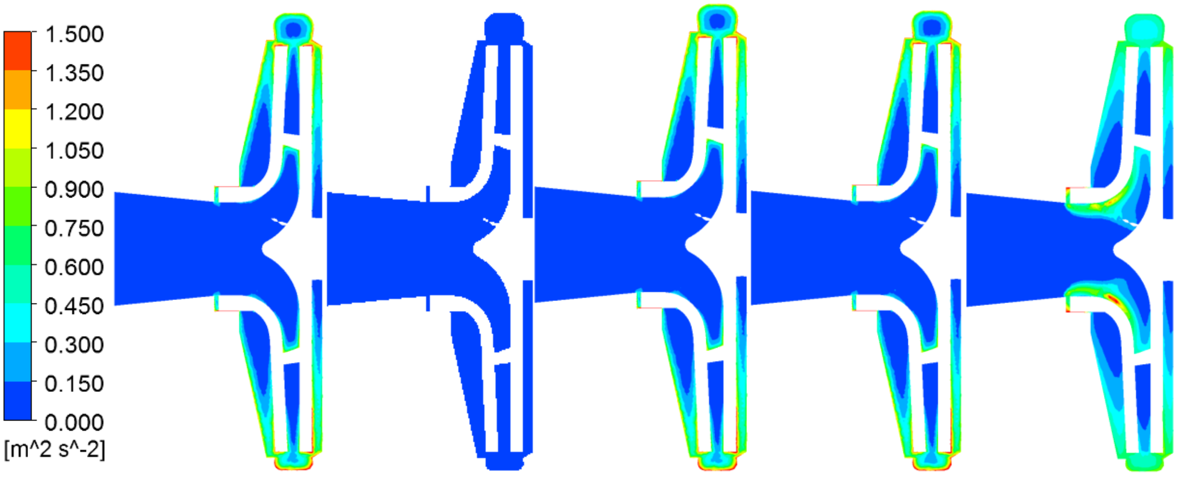
From figure 2, it indicated that, at design condition, the calculation heads with different turbulence models have the same variation trend, but still have a certain error compared with experimental value. Among the five calculation results, SSG Reynolds Stress model and K-Omega model have larger deviation, and the relative error under SSG Reynolds Stress model is beyond 10%. The relative errors under RNG K-Epsilon model, K epsilon EARSM model and K-Epsilon model are less and are all around 2%. Then, in the figure 2, the five turbulence models almost get a minimum value at the same time when the mesh number is 3986230.

As is showed in figure 3, under design condition, the calculation efficiencies with the five turbulence models have errors to different degree compared with experimental values. Among the five calculation results, the discrepancy of calculation efficiency under SSG Reynolds Stress model and K-Omega model is larger. And compared with the calculation results under RNG K-Epsilon model and K-Epsilon model, the absolute discrepancy of efficiency under K epsilon EARSM is always the smallest. Then, in the figure 3, we can see that the minimum value for each turbulence model also appear at the time when the mesh number is 3986230.

### 2.5. Turbulence models influence on flow field

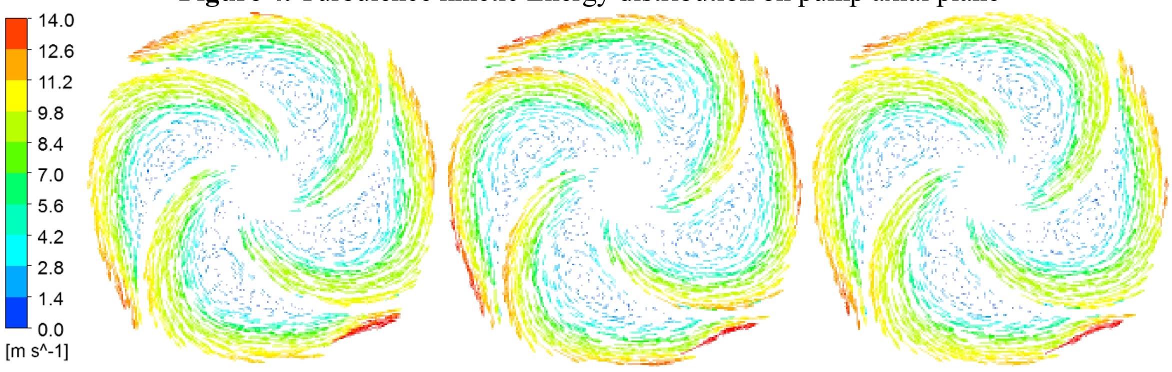
Combination of the calculation results above, 3986230, which is the number of mesh, is chosen to have the numerical simulation and analysis so that the result of turbulence influence on flow field is concluded.

Figure 4 is turbulence kinetic Energy distribution on pump axial plane (in vertical with pump outlet), from which we can see that there are differences among the five turbulence kinetic energy distribution. Turbulence kinetic energy under SSG Reynolds Stress model is minor, while the other four turbulence models have larger turbulence kinetic energy at positions (such as volute, cavity and ring). Different from K-Epsilon, K epsilon EARSM and RNG K-Epsilon, larger turbulence kinetic energy of K-Omega model distributes near the impeller inlet. Besides, section near tongue have different turbulence kinetic energy distribution, the larger turbulence kinetic energy of K-Omega distributes in all the section, while K-Epsilon, K epsilon EARSM and RNG K-Epsilon only in volute wall surface.

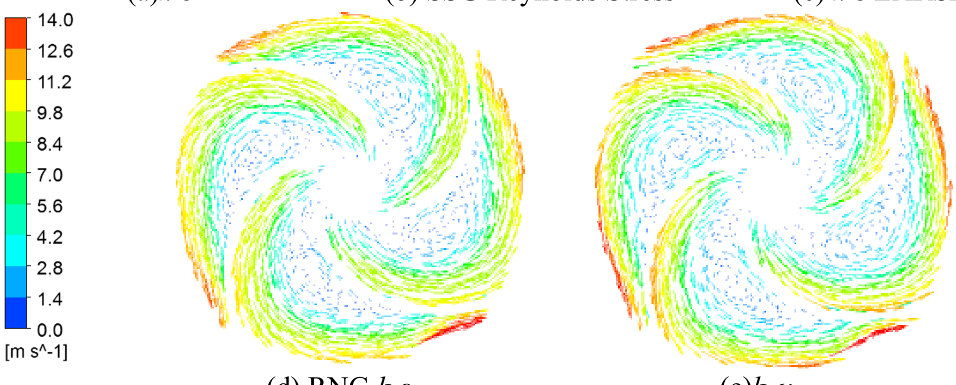



(a)*k-ε* (b)SSG Reynolds Stress (c) *k-ε* EARSM (d) RNG *k-ε* (e)*k-ω*

**Figure 4.** Turbulence kinetic Energy distribution on pump axial plane




(a)*k-ε* (b) SSG Reynolds Stress (c) *k-ε* EARSM



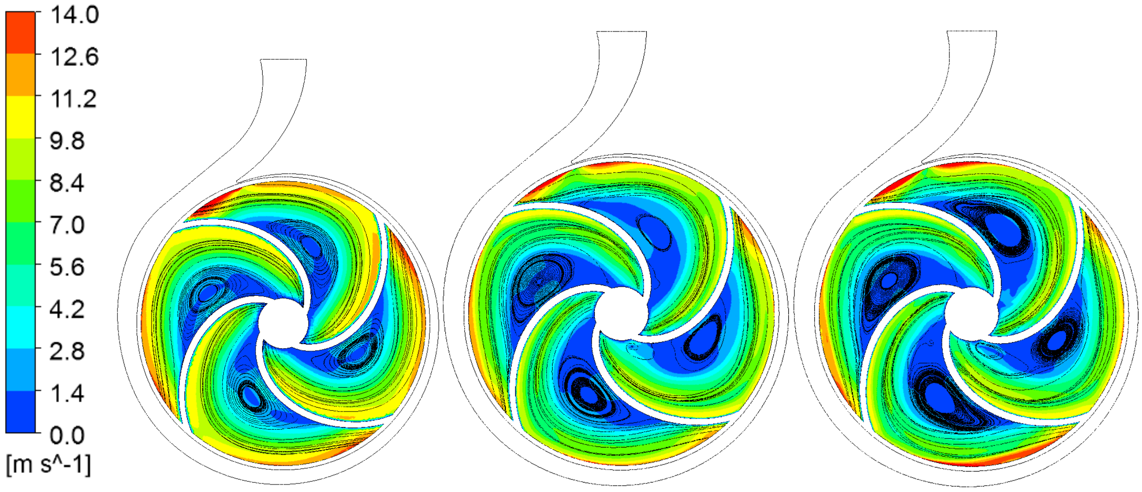

(d) RNG *k-ε* (e)*k-ω*

**Figure 5.** Relative velocity distribution on impeller

It can be seen from figure 5 that the five computational impeller relative velocities have a certain similarity and error of different degree. The relative velocity increases gradually from blade pressure side to blade suction side, there are axial vortex near impeller inlet. Though the five kind of relative velocity distributions are even, compared with K-Epsilon, K epsilon EARSM and RNG K-Epsilon, SSG Reynolds Stress and K-Omega have larger vortex field. Meanwhile, SSG Reynolds Stress exist wide high-speed area.

### 2.6. Mesh numbers influence on internal flow field

According to the analysis both in energy performance and in internal flow field above, K epsilon EARSM model is chosen to study mesh numbers effect on internal flow field.




(a) 1933822 (b) 8535449 (c) 23882859

**Figure 6.** Relative velocity distribution on impeller

Three groups of mesh numbers are used to study how mesh numbers influence on internal flow field. Figure 6 is relative distribution on impeller, it can be seen that a large number of vortex field exist in impeller flow passage. And smaller vortex field is showed when mesh number is less (figure 6a). However, the vortex field enlarges gradually and is more obvious with the increase of mesh number (figure 6b and figure 6c).

## 3. Conclusions

(1) According to the comparison of calculation efficiency and calculation head with five different turbulence models, the calculation results under SSG Reynolds Stress and K-Omega have a larger deviation, and the discrepancy range is over 5%. While the calculation results in K epsilon EARSM, RNG K-Epsilon, and K-Epsilon are closer to experimental value. But compared to RNG K-Epsilon and K-Epsilon, K epsilon has a higher precision in calculation result of energy performance.

(2) According to the comparison and analysis of turbulence kinetic energy distribution on pump axial plane and relative velocity on impeller in five different turbulence models, compared to SSG Reynolds Stress and K-Omega, the descriptions of internal flow in K epsilon EARSM, RNG K-Epsilon and K-Epsilon are better.

(3) By comprehensive consideration of the analysis result of energy performance and internal flow field, K epsilon EARSM model is chosen to study with different mesh numbers effect on internal flow field. According to compare absolute velocity distribution on volute with different mesh numbers, the study shows the internal flow in centrifugal pump is depicted more perfect with the increase of mesh numbers.

## Acknowledgments

This work was supported by National Natural Science Foundation of China (No. 50825902、51079062、51109095、51179075), A Project Funded by the Priority Academic Program Development of Jiangsu Higher Education Institutions, Natural science fund in Jiangsu Province (BK2010346, BK2009006) and Postgraduate Innovation Foundation of Jiangsu Province (CXLX11_0576).

## Appendices

$b_2$ Blade outlet width [m]
$b_3$ Volute inlet width [m]
$D_2$ Impeller outlet diameter [m]
$D_3$ Volute inlet diameter [m]
$g$ Gravitational acceleration [m/s$^2$]
$H_e$ Experimental Head [m]
$H$ Computational Head [m]
$\Delta H$ Relative error of head [m]
$M$ Torque [Nm]
$n$ Rotation speed [r/min]
$Q$ Flow rate [m$^3$/h]
$Z$ Blade numbers
$\beta_2$ Blade outlet angle [$^O$]
$\rho$ Fluid density [m$^3$/kg]
$\eta$ Total efficiency
$\eta_e$ Experimental efficiency
$\Delta\eta$ Relative error of efficiency
$\omega$ Angular velocity of impeller [rad/s]

**References**

[1] Yao Z and Chen K M 2002 *J. University of Shanghai for Science and Technology* **24**(2) 138-43
[2] Shu M H and Liu H L 2011 *China rural water and hydropower* (2) 142-145
[3] Ma Y and Wang Z B 2009 *Chemical Engineering and Machinery* (6) 596-600.
[4] Sun R and Li Z Q 2002 *Power Engineering* **22**(3) 1750-1758
[5] You C F and Qi H Y 2001 *Power Engineering* **21**(2)1128-1131
[6] Li G W, Sun Q and Guo R Y 2011 *Heilongjiang Agricultural Sciences* **3** 129-132
[7] Liu M Q, Li Q W, Bai Y H and Liu Z Y 2010 *Transactions of the Chinese Society for Agricultural Machinery* **41** 6-9
[8] Zhao B J, Yuan S Q, Liu H L and Tan M G 2006 *Chinese Journal of Mechanical Engineering* **19**(1) 59-62
[9] Guan X F 1995 *Modern Pumps Theory and Design* (Beijing: China Astronautic Publishing House)
[10] Tan M G 2008 Prediction Research on Energy Characteristics for Centrifugal Pumps *Ph. D. Thesis* Technical Research Center of Fluid Machinery Engineering, Jiangsu University, Zhenjiang