Numerical simulation and uncertainty estimation in a deep-well centrifugal pump

L Bai¹, W D Shi¹, L Zhou¹,² and W G Lu¹
¹ Research Center of Fluid Machinery Engineering & Technology, Jiangsu University, China
² Department of Mechanical Engineering & Materials Science, Washington University in St. Louis, USA
E-Mail: wdshi@ujs.edu.cn

Abstract. One stage deep-well centrifugal pump section is investigated numerically in this study. Three-dimensional steady Reynolds-Averaged Navier–Stokes (RANS) equations are solved on high-quality fine structured grids in conjunction with the shear stress transport (SST) k-ω turbulence model by employing the CFD (computational fluid dynamics) software ANSYS-Fluent 13.0. A detailed grid sensitivity analysis was carried out to analyse the influence of grid elements number. The discretization errors were calculated specifically through the Richardson extrapolation theory. The uncertainty of fine grid for the simulation in this paper was estimated as 1.23%. The prototype experimental test results were acquired and compared with the data predicted from the numerical simulation, which showed that there is a good agreement between two methods.

1. Introduction
Pump is one of the most energy consuming devices widely used with general machinery in industrial applications. The information about the pump efficiency, manufacturing cost and reliability is required by the end-users before installation decisions are made. Pump manufacturing industry is currently focused on improving the energy efficiency of pumps while addressing stricter and stricter manufacturing constraints [1, 2]. This is particularly of interest in multistage deep-well centrifugal pump (DCP) design since it has high head requirement and severe constraints on operating dimensions. In our previous work [3, 4] we proposed a new type of multistage deep-well centrifugal pump with a three-dimensional surface return diffuser, which has a compact size and high pressure conversion capability. However, additional research is needed to expand its application range and to further optimize its hydraulic performance.

Many studies have been conducted to investigate and improve flow field of in centrifugal pumps by computational fluid dynamics (CFD) and experiments. Goto et al. [5] presented a new design methodology based on the application of a three dimensional inverse design and its validation by CFD, and the effectiveness of this new approach were proved with the redesign of a pump diffuser. Boncinelli et al. [6] redesigned a bowl-type diffuser by optimizing the blade turning angle distribution and meridional channel shape. Goel et al. [7] studied the influence of vane shape on the hydrodynamic performance of a diffuser in a pump stage, and concluded that thin vanes helped to improve the hydrodynamic performance significantly.
In this study, a pump section chosen from one stage of a deep-well centrifugal pump was simulated in ANSYS-Fluent, the computational model include the whole flow components. Three-dimensional steady Reynolds-Averaged Navier–Stokes (RANS) equations are solved on high-quality fine structured grids in conjunction with the shear stress transport (SST) k-ω turbulence model. The discretization errors were calculated and analysed, and the simulated pump head was compared with experimental results.

2. Geometric Models

2.1. Design parameters
A single-stage pump section is chosen from a multistage deep-well centrifugal pump (DCP). The detailed geometry parameters of the impeller and the diffuser are given in our previous paper [3]. Some important pump characteristics are summarized in Table 1.

| Description                  | Parameter | Value |
|------------------------------|-----------|-------|
| Design flow rate (m³/h)      | \( Q_{des} \) | 20    |
| Single-stage head (m)        | \( H_s \) | 11    |
| Rotating speed (r/min)       | \( n \)  | 2850  |
| Stage number                 | \( N \)  | 6     |
| Specific speed (m³/h, m, r/min) | \( n_s \) | 35    |
| Non-dimensional value        | \( \Omega_s \) | 0.664 |

2.2. Impeller design
The impeller’s front shroud is extended to the inner casing diameter to maximize the impeller single-stage head. Its cross-section and solid model created using Pro/E are shown in Fig.1, and the main geometric parameters of the impeller are presented in Table 2.

![Impeller Cross-section and Solid Model](image)

### Table 2. Geometric parameters of impeller

| Description                  | Parameter | Value |
|------------------------------|-----------|-------|
| Blade number                 | \( z_i \) | 7     |
| Inlet blade angle (°)        | \( \beta_1 \) | 32    |
| Outlet blade angle (°)       | \( \beta_2 \) | 21    |
| Blade wrap angle (°)         | \( \phi_i \) | 130   |
| Outlet blade width (mm)      | \( b_2 \) | 12    |
| Front shroud diameter (mm)   | \( D_{2\text{max}} \) | 119   |
| Rear shroud diameter (mm)    | \( D_{2\text{min}} \) | 108   |

2.3. Diffuser
Three-dimensional surface return diffuser (3DRD) was used in the investigated pump. The inlet of 3DRD is twisted, and there are three baselines in the blade concave surface. The diffuser inlet blade
angles of three baselines are set at a variable camber angle with respect to the circumferential direction. The cross section and solid model was shown in Fig.3.

![Figure 3. Diffuser](image)

3. Numerical Methodology
The 3D incompressible Reynolds Averaged Navier-Stokes (RANS) computations are performed using ANSYS-Fluent 14.5 code. The finite volume method is used for discretization of the governing equations, and a second-order accurate numerical algorithm is employed to solve the governing equations. The time step in the transient simulation is $5.848 \times 10^{-5}$ s, corresponding to 1° of the impeller rotating speed, so a complete revolutions is performed each 360 time steps. Numerical convergence is set to a maximum of $10^{-5}$ and periodicity solution is fulfilled after seven impeller revolutions. Before the transient simulation, a steady simulation was carried out by using Frozen Rotor, that is, without any relative shifting of the impeller meshes but including the centrifugal effects in the flow equations. When achieving the setting convergence, the steady simulation was used as an initial value to start the transient sliding mesh simulation. Sliding mesh model is a special case of general dynamic mesh motion wherein the nodes move rigidly in a given dynamic mesh zone. Additionally, multiple cells zones are connected with each other through non-conformal interfaces. As the mesh motion is updated in time, the non-conformal interfaces are likewise updated to reflect the new positions each zone.

3.1. Computational Domain
The flow domain of the CFD model consisted of impeller, diffuser, lateral cavity, leakage gap, inlet section and the outlet section. The inlet section and outlet section were extended to the exact position of the pressure measurement points in the experiment. The leakage gap was set as 0.5mm according to the original design. However, it is should be note that the actual gap maybe change due to the unsteady axial force under different flow conditions. After modeling in Pro/E, the geometric model was imported to ANSYS-ICEM for the further processing.

![Figure 4. Computational domain](image)

3.2. Mesh
The mesh of the computational domain (Fig.4) was created by means of the software of ANSYS-ICEM 13.0, and the entire computational domain was meshed with the high quality structured grid based on the Q-type and Y-type block topology. A detailed grid sensitivity analyses with three different mesh numbers have been carried out by steady simulation, as summarized in Table.1.
Computations have been performed using three grids with 1.298, 2.879, and 5.399 millions of cells, and the average global gird size is 1.80mm, 1.35mm and 1.00mm, respectively.

| Parameters       | Case 1 | Case 2 | Case 3 |
|------------------|--------|--------|--------|
| Global Grid size | 1.80mm | 1.35mm | 1.00mm |
| Diffuser         | 314 546| 687 030| 1 236 456|
| Impeller         | 196 371| 488 712| 830 417|
| Lateral cavity   | 85 320 | 177 408| 488 064|
| Inlet section    | 201 068| 381 330| 608 000|
| Leakage Gap      | 26 784 | 47 232 | 115 040|
| Outlet section   | 474 501| 1 097 552| 2 071 104|
| Total            | 1 298 590| 2 879 264| 5 399 367|

In order to evaluate the grid independence, a monitoring line was created on the diffuser vanes, as shown in the Fig.5(a). The detailed static pressure values on the monitoring line of three different mesh cases were compared in Fig.5(b). Obviously the static pressure just experience a few change as the grid size was varied from approximately 1.35mm to 1.00mm. Considering the computational ability of computer workstation used in this work, the grid 1.00mm was chosen as the finest mesh for the following study. The total mesh number of the fine mesh is more over than 5.40 million. Furthermore, the value of y+ was less than 3 in the entire computational domain, which indicates that the near-wall nodes are located in the viscous sublayer. Fig.6 shows a general view of the mesh in the impeller and the diffuser.
3.3. Boundary Conditions
Various boundary conditions used in the computational domain are summarized in Table 4. The inlet of the modelled pump connects to a tank; therefore the pump inlet is set at total pressure value of 1.5 m of water column. Meanwhile, mass flow rate was set at pump inlet and pressure outlet boundary condition was used at the pump exit. No-slip wall conditions are used at all the physical surfaces.

| Position   | Boundary condition    |
|------------|-----------------------|
| Inlet      | Mass flow inlet       |
| Outlet     | Pressure outlet       |
| Walls      | Nonslip wall          |
| Impeller   | Rotating reference frame |
| Diffuser   | Stationary reference frame |

4. Numerical Uncertainty Estimation
With the purpose of estimate the numerical accuracy of the calculations, an established method recommended by the Fluids Engineering Division of ASME was adopted [8]. In principle, this method is based on the Richardson extrapolation theory [9] and has been developed into a generalized formulation applicable for the practical CFD cases.

In this paper, three different grid sizes have been studied for the numerical accuracy analysis, namely grid size of 1.0, 1.35 and 1.8. The average mesh volume and grid size \( h \) was computed, where the \( h \) is defined as:

\[
h = \left( \frac{1}{N} \sum_{i=1}^{N} (\Delta V_i) \right)^{1/3}
\]  

Then two monitor points in the flow channel were selected as the “variable \( \phi \)”. As shown in Fig.1, P1 is in the impeller outlet, and P2 is in the middle of the diffuser. The detailed parameters of two points are summarized in Table 1. It is desirable that the grid refinement factor, \( r = \frac{h_{\text{coarse}}}{h_{\text{fine}}} \), be greater than 1.3. This value of 1.3 is based on experience, and not on formal derivation.

![Figure 7. Monitoring points’ position](image)

| Table 5. Numerical results of monitoring points |
|-----------------------------------------------|
| Global Grid size | 1.00 | 1.35 | 1.80 |
| Grid number      | \( N_1, 5\,399\,367 \) | \( N_2, 2\,879\,264 \) | \( N_3, 1\,298\,590 \) |
| Average grid volume | 0.6423 | 1.4605 | 3.3348 |
| Average grid size \( h_i \) | \( h_1, 0.8628 \) | \( h_2, 1.1346 \) | \( h_3, 1.4940 \) |
| Tangential velocity of P1, m/s | \( \phi_1, 10.9806 \) | \( \phi_2, 11.1086 \) | \( \phi_3, 11.3840 \) |
| Static pressure of P2, Pa | \( \phi_1, 124960 \) | \( \phi_2, 124116 \) | \( \phi_3, 123878 \) |
Let \( h_1 < h_2 < h_3 \) and \( r_{21} = h_2 / h_1, r_{32} = h_3 / h_2 \), and calculate the apparent order, \( p \), of the method using the expression

\[
p = \frac{1}{\ln(r_{21})} \ln\left| \varepsilon_{32} / \varepsilon_{21} \right| + q(p)
\]

(2)

where \( \varepsilon_{32} = \varphi_3 - \varphi_2, \varepsilon_{21} = \varphi_2 - \varphi_1, \varphi_k \) denoting the solution on the \( k^{th} \) grid. Agreement of the observed apparent order with the formal order of the scheme used can be taken as a good indication of the grids being in the asymptotic range; the converse should not necessarily be taken as a sign of unsatisfactory calculations.

Calculate the extrapolated values from:

\[
\varphi_{21}^{\text{ext}} = \frac{r_{21}^p \varphi_1 - \varphi_2}{r_{21}^p - 1}
\]

(5)

Calculate and report the following error estimates, along with the apparent order \( p \):

Approximate relative error:

\[
e_a^{21} = \left| \frac{\varphi_3 - \varphi_1}{\varphi_1} \right|
\]

(6)

Extrapolated relative error:

\[
e_{e^{21}} = \left| \frac{\varphi_{21}^{\text{ext}} - \varphi_1}{\varphi_{21}^{\text{ext}}} \right|
\]

(7)

The fine-grid convergence index:

\[
\text{GCI}_{\text{fine}}^{21} = \frac{1.25 e_a^{21}}{r_{21}^p - 1}
\]

(8)

These errors for \( P1 \) and \( P2 \) have been calculated following above process. The results were indicated in Table 4. In this case, the numerical uncertainty in the fine-grid solution (i.e. global grid size of 1.0) should be reported as 1.23%.

| Parameters | \( \varphi = \) tangential velocity of \( P1 \) | \( \varphi = \) static pressure of \( P2 \) |
|------------|-----------------------------------|-----------------------------------|
| \( N_1, N_2, N_3 \) | \( N_1 = 5399, N_2 = 2879, N_3 = 1298 \) |
| \( r_{21} \) | 1.1210 | 124960 |
| \( r_{32} \) | 1.1018 | 124116 |
| \( \varphi_1 \) | 10.9806 | 123878 |
| \( \varphi_2 \) | 11.1086 | 123878 |
| \( \varphi_3 \) | 11.3840 | 123878 |
| \( p \) | 2.7859 | -4.6547 |
| \( \varphi_{21}^{\text{ext}} \) | 10.8688 | 123788 |
| \( e_a^{21} \) | 1.166% | 0.675% |
| \( e_{e^{21}} \) | 1.029% | 0.946% |
| \( \text{GCI}_{\text{fine}}^{21} \) | 1.227% | 1.172% |

5. Comparison of Computed and Experimental Pump Head

The investigated pump section was manufactured and tested. Pump head for different flow rates is measured by experimental analysis and predicted by the numerical simulations. The pump head is measured by a high accuracy pressure transmitter in the experiment.

Fig. 8 shows the comparison of computed and measured head for different flow rates, and a good agreement is obtained. At the rated flow rate, the computed head of 10.57m is 2.3% higher than the experimental head of 10.33m. The difference may be attributed to the fact that in the numerical
simulation we assumed the face seal in front of the impeller shroud to be completely sealed, neglecting the volume losses caused by the leakage flow in the impeller side chambers. Nevertheless, for all the computed flow rates, the numerical results are consistent and in good agreement with the experimental data. It should also be noted that for smaller ($Q/Q_{des} < 0.6$) flow rate as well as the bigger flow rate ($Q/Q_{des} >1.2$), the error between the computation and the experimental data becomes greater.

**Figure 8.** Comparison of the computed and measured pump head

6. Conclusions
In this study, one stage of a multi-stage deep-well centrifugal pump was simulated in CFD software. The simulations employ the multi-reference frame (MRF) technique, and the turbulence effects are simulated with the SST k-ω turbulence model. The entire computational domain was meshed with the high quality structured grid based on the Q-type and Y-type block topology. The discretization errors were calculated specifically through the Richardson extrapolation theory. The uncertainty of fine grid for the simulation in this paper was estimated as 1.23%. Pump head for different flow rates is measured by experimental analysis and predicted by the numerical simulations, the comparisons presents a good agreement between two methods. At the design flow rate, the computed head is approximately 2.3% higher than the experimental head.

Acknowledgments
The work was supported by the National Natural Science Foundation of China Grant Nos. 51279069 and 51109093.

References
[1] Gopalakrishnan S 1999 *J. Fluids Eng. - Trans ASME* **121** 237
[2] Hergt P H 1999 *J. Fluids Eng. - Trans ASME* **121** 248
[3] Zhou L, Shi W D, Lu W G, Hu B and Wu S Q 2012 *J. Fluids Eng. - Trans ASME* **134** 0711002
[4] Shi W D, Lu W G, Wang H L and Li Q F 2009 *FEDSM* 1 Pts A-C 91
[5] Goto A and Zangeneh M 2002 *J. Fluids Eng. - Trans ASME* **124** 319
[6] Boncinelli P, Biagi R, Focacci A, Corradini U, Arnone A., Bernacca M and Borghetti M 2008 *J. Turbomach. - Trans ASME* **130** 031013
[7] Goel T, Dorney D J, Hafkka R T and Shyy W 2008 *Computer Fluids* **37** 705
[8] Westra R W, Broersma L, van Andel K and Kruyt N P 2010 *J. Fluids Eng. - Trans ASME* **132** 061104
[9] Celik I B, Ghia U, Roache P J, Freitas C J, Coleman H and Raad P 2008 *J. Fluids Eng. - Trans ASME* **130** 1
[10] Richardson L F and Gaunt J A 1927 *Philos. Trans. R. Soc. London, Ser. A* **226** 299