Development of new “multivolute casing” geometries for radial force reduction in centrifugal pumps

Hamed Alemi∗, Seyyed Ahmad Nourbakhsh, Mehrdad Raisee and Amir Farhad Najafi

Hydraulic Machinery Research Institute, School of Mechanical Engineering, College of Engineering, University of Tehran, Tehran, 143955961, Iran

(Received 21 May 2014; final version received 4 December 2014)

Large radial force causes several issues in pumps, such as noise, vibration, and extra load on the bearings. To reduce the radial force, the effects of concentric volute and multivolute geometry on the head, efficiency, and radial force of a low speed centrifugal pump at off-design conditions were investigated. Commercial software with the $k-\omega$ turbulence model and automatic near wall treatment was employed for the prediction of fluid flow inside the pump. Flow simulations for three casings concentric at 180°, 270°, and 360° from the tongue showed that the 270° concentric volute generates the lowest radial force at throughout the entire range of flow rate. The triple-volute and tetravolute casings are also proposed as new volute geometries. The flow analysis of a double-volute, triple-volute, and tetravolute show that the triple-volute is the most appropriate volute geometry at off-design conditions.

Keywords: pump; volute; CFD; radial force; multivolute; concentric volute

1. Introduction

The circumferential pressure distribution around the impeller periphery becomes non-uniform at off-design conditions. This leads to radial force on the pump shaft. A large radial force generates noise, vibration, and extra load on the bearings, thus deteriorating the natural pump performance. The magnitude of the radial forces can be directly measured using the bearing or shaft forces. Moreover, the radial forces can be estimated by static pressure measurement or computation around the impeller periphery. From a computational point of view, the prediction of the internal flow developing in a pump is extremely challenging at off-design conditions due to the presence of rotation, the curvature of the impeller, impeller-volute interaction, and flow separation. Nevertheless, numerous investigations have recently been carried out to predict turbulent flow in centrifugal pumps. Kelder, Dijkers, Vas Esch, and Kruyt (2001) used both experimental and numerical methods to study turbulent flow at a Reynolds number of $1.7 \times 10^6$ in the volute of a low specific-speed pump close to the design point. They showed that in such conditions, the core flow characteristics are similar to the potential flow. Thus, they employed an inviscid solver to obtain the flow field. Gonzalez, Fernandez, Blanco, and Santolaria (2002) investigated gap variation for a pump with a specific speed of $N_S = 26$. They found that an increase by about 50% of the maximum pressure amplitude occurs where the gap distance is reduced from 15.8% to 10% of the impeller radius.

Spence and Amaral-Texeira (2008) attempted to estimate pressure pulsation in a double-entry, double-volute centrifugal pump with a specific speed of $N_S = 39$ by means of using CFX-TASC, a commercial flow code. Their numerical results were in good agreement with the measurements. Yedidiah (2008) tried to synthesize knowledge between pump experts and CFD specialists with a test case study. Pumps with higher $N_s$ were found to need a larger cutwater gap for pump performance improvement. Feng, Benra, and Dohmen (2010) applied various turbulence models to simulate non-stationary turbulent flow in a radial diffuser pump. The numerical results obtained for the pressure, velocity, and turbulence fields were analyzed and compared with LDV and PIV data. They reported that the choice of turbulence model does not have a large influence on the pressure field predictions. Among the turbulence models examined, the $k-\omega$ turbulence model had superior performance in the prediction of the velocity vector directions and the turbulent kinetic energy levels. Barrio, Fernandez, Blanco, and Parrondo (2011) computed steady and unsteady pressure distributions around an impeller by solving full 3D-URANS equations with the commercial code FLUENT as a function of flow rate between 10% and 130% of the rated conditions. Zhou, Wang, and Yang (2012) numerically analyzed a double-suction pump using SST turbulence models and compared the results with experiment results. They reported that an SST turbulence model can accurately predict pump performance.
at different flowrates. Liu, Liu, Dong, Ren, and Du (2012) used commercial CFX software to investigate the effects of mesh size and turbulence models on the numerical simulation of a centrifugal pump at the design point. Their CFD calculation was carried out with five different turbulence models ($k-\varepsilon$, SSG Reynolds Stress, $k-\varepsilon$ EARSM, RNG $k-\varepsilon$, and $k-\omega$) and six different mesh numbers ranging from 1 million to 25 million. It was confirmed that the $k-\varepsilon$ EARSM turbulence model is more robust than the others, and the internal flow was depicted more perfectly with increased mesh numbers. Zhongyong, Junjie, Shuai, and Shouqi (2011) presented a numerical simulation of the steady and unsteady radial force of centrifugal pump impellers equipped with various volutes, including a single volute, double-volute, and a vaned volute. Their results showed that both the double-volute and the vaned volute decreased the radial forces. They concluded that the double-volute is superior, since it significantly decreases the radial force in comparison with the single volute, and it has a more compact radial size than the vaned volute. 

Asuaje, Bakir, Koidri, Kenery, and Rey (2005) analyzed a centrifugal pump with a specific speed of $N_S = 32$ using CFX code. In their study, the three well-known $k-\varepsilon$, $k-\omega$, and SST turbulence models were employed. They computed the velocity and pressure fields in the flow domain and the radial force on the pump shaft. Torabi and Nourbakhsh (2011) studied the effects of variation in some of the geometrical factors of volutes on the radial force. They found that the gap between the impeller and tongue severely affects the radial force. More specifically, they found that a 6% reduction in the gap increases the maximum value of the static pressure by 50% round the impeller exit.

As discussed, a number of experimental and computational investigations have been carried out to study the effects of various geometrical parameters on the hydraulic performance and radial force of centrifugal pumps. In this study, multivolute casings are proposed for application in a centrifugal pump for the first time. To the best of our knowledge, no numerical simulation has been performed to investigate various concentric volutes for different angles from the tongue. One of the difficulties of this study was the grid generation of multivolute casings due to the intricate geometry. We overcame this challenge using special tools in ANSIS-Meshing 13.0 software, such as local refinement and inflation.

2. Main pump geometry

In the present study, we used experimental measurements of Kelder et al. (2001) for CFD validation. The details of pump geometry used by Kelder et al. are shown in Figure 1. The investigated centrifugal pumps have a low-$N_S$ of 21. Design parameters of the studied pump are summarized in Table 1.

| Parameter                          | Value |
|------------------------------------|-------|
| Metric specific speed, $N_s$       | 21    |
| Nominal flow coefficient, $\phi_n$| 0.15  |
| Nominal head coefficient, $\psi_n$ | 0.124 |
| Impeller exit diameter, $d_2$ (mm) | 640   |
| Impeller exit width, $b_2$ (mm)    | 25    |
| Number of Blades, $Z$              | 7     |

Figure 1. (a) Kelder volute geometry and locations of velocity and static pressure measurement (b) Volute cross section dimensions in millimetres.
Experiments were performed at flow rates of 82.5\%, 100\%, and 112\% of the design flow rate (28.8 m$^3$/h). An LDV measurement system was employed for velocity measurements. Velocity and static pressure data were collected at the locations shown in Figure 1(a). As shown in Figure 1(a), with the pressure tabs are located at the shroud side just outside the impeller, yet in volute at mid-height of the outer wall. More details of the pump configuration and experimental methods can be found in (Kelder et al., 2001).

3. Numerical method and model description

To solve the full 3D Reynolds Averaged Navier-Stokes (3D-RANS) equations for different working conditions the commercial software CFX-13.0 was used in this study. The solver was a 3D CFD code in which the governing equations are discretized using the finite-element based finite-volume method. Two main advantages of such a hybrid CFD method are geometric flexibility due to the finite-element method and retaining the conservation properties of the finite-volume method which results in low numerical error in the case of non-smooth grids. For space discretization, the second-order upwind scheme was applied to achieve high accuracy and reasonable solution stability.

For grid generation, the pump was divided into four components. As shown in Figure 2(a) these four components are: (i) inlet duct, (ii) impeller, (iii) volute, and (iv) outlet duct. Uniform flow normal to boundary and constant static pressure were implemented for inlet and outlet boundary conditions, respectively. Straight ducts are enough long to eliminate effects of inlet and outlet boundary conditions on flow regime in pump. Spence and Amaral-Texeira (2009) reported that these boundary conditions cause rapid convergence. Figure 2(b) shows the generated grids for the pump impeller and volute. As shown, due to complexity of geometry, hexahedral cells near important regions (with curvature and strong gradient) including blades and volute outer wall, and tetrahedral cells in other regions were used. Mesh clustering was applied close to some effective regions, such as leading and trailing edge of the blades and the volute tongue where flow separation may occur (see Figure 2). During grid generation process, orthogonal quality, aspect ratio, and skewness were assured to be in desirable range. For quasi-steady simulation the grids between the impeller and volute on one hand, and inlet duct and impeller on the other hand, were conjoined by means of a frozen rotor interface. In other words, impeller rotation was considered and the frame of reference was changed but relative orientation of the components across the interface was fixed. Both two frames of reference were connected in such a way that each of them had a fixed relative position throughout the calculation. This was equivalent to taking a snapshot from the flow field in an instance of time. Moreover, general grid interface (GGI) was employed between the volute and outlet duct. Boundary conditions in solution domain are summarized in Table 2.

Grid independency studies were performed using different mesh sizes. As shown in Figure 3, it was found that by increasing the number of cells from $7 \times 10^5$ to

| Location                        | Boundary condition                |
|---------------------------------|-----------------------------------|
| Inlet duct entrance             | Uniform mass flow, normal to      |
| Inlet duct/impeller interface   | Frozen rotor interface            |
| Impeller/volute interface       | Frozen rotor interface            |
| Volute/outlet duct interface    | General grid interface (GGI)      |
| Outlet duct exit                | Constant static pressure          |
| Walls                           | Smooth and No-slip                |
Further mesh refinement from $1 \times 10^6$ cells to $2.4 \times 10^6$ cells, decreases the head by about 0.6%. Since the predicted head did not show a strong sensitivity to the grid refinement, the numerical results can be considered as grid independent. All the numerical results were obtained using the $1 \times 10^6$ cells mesh.

It was found that by increasing the turbulent intensity at inlet from 5% to 10% and viscosity ratio from 10 to 100, the predicted head changes less than 0.01%. Thus, turbulent intensity and viscosity ratio were set to 5% and 10%, respectively.

4. Validation of CFD results

Numerical simulations of the present study were validated by comparing the predicted static pressure and velocity components with the experimental data of Kelder et al. at selected locations (see Figure 1). In this paper, three well-known turbulence models were examined; namely, the standard $k-\varepsilon$, the low-Re $k-\omega$, and the SST. The standard $k-\varepsilon$, developed by Jones and Launder (1972) is a high-Re turbulence model that is able to predict inviscid flow regime properly. So, this model needs wall function approximation to solve viscous region near walls. The low-Re $k-\omega$ turbulence model, developed by Wilcox (1986), could be used for entire flow region even near walls. The low-Re $k-\omega$ model would typically require a near wall resolution of $y^+ < 2$. In turbomachinery flows, even $y^+ < 2$ can not be guaranteed and for this reason, a new near wall treatment was developed for the $k-\omega$ models. It allows for smooth shift from $k-\omega$ model to a wall function formulation regarding local $y^+$ value. The SST turbulence model, developed by Menter (1993, 1994), combines the $k-\omega$ and the $k-\varepsilon$ turbulence models. It consists of a transformation of the $k-\varepsilon$ model in the outer region to a $k-\omega$ formulation near the walls.

In Figure 4, pressure distributions obtained from three turbulence models are compared with the experimental data of Kelder et al.

It is observed that all three turbulence models produce comparable results in both design and off-design conditions. Despite the presence of local discrepancies between CFD predictions and measurements, both the level and the trend of CFD results are in overall agreement with the measurements. The wavy shape of pressure distribution seen in CFD predictions is due to impeller blades effects. Experimental and numerical results showed that depending on the value of flow rate, the static pressure distribution has different behavior. For the design flow rate, the static pressure remains almost uniform, which leads to minimum radial force. At the off-design conditions corresponding to the low capacity $\phi/\phi_n = 0.825$, the static pressure increases with $\theta$ while an opposite trend is observed for $\phi/\phi_n = 1.12$. At overload, the effective cross-sections are small for the flow. As a result, volute acts like a nozzle and the pressure decreases. In contrast, at low capacity, the effective cross-sections are large for the flow; accordingly, the volute acts like a diffuser and the pressure raises.

In Figure 5 the predicted radial and tangential velocity components were compared with the experimental data of Kelder et al. Comparisons were made for three working conditions ($\phi/\phi_n = 0.825, 1.0, and 1.12$) along four traverses (A, D, F and H) in the volute (see Figure 1). More or less, all turbulence models returned similar results consistent with the experimental data of Kelder et al. There are some discrepancies between the simulation results and experimental data in the $V_\theta$ on the Traverse A. The Traverse A is close to tongue, thus flow regime induced by blade-tongue interaction is strongly unsteady. Since, using quasi-steady CFD simulation may lead to numerical error increment.

![Figure 4](image-url) The pressure distributions around the impeller periphery.
As mentioned in the introduction, Kelder et al. employed Euler equations in their numerical predictions and found reasonable agreement between computational and experimental data. However, level of agreement with the measured data was not as close as those shown in Figure 5.

The predicted coefficients using three turbulent models with steady simulation and just $k-\omega$ model with unsteady simulation were compared with experimental data for a wide range of flow coefficients; namely, the design point and off-design conditions (see Figure 6). Firstly, in steady simulations, although the $k-\varepsilon$ turbulence model returns more accurate results for the non-dimensional head near the design point, the head predictions of the $k-\omega$ model at off-design conditions are closer to the experimental data, especially at high flow rate (Figure 6(a)). Radial forces were computed by means of a full integration of the pressure and shear stress distribution on the impeller surfaces (blades, Hub, and shroud). The $k-\omega$ turbulence model returns radial force predictions in better agreement with experiments than other models (Figure 6(b)). Secondly, as shown in Figure 6, although unsteady simulation using $k-\omega$ turbulence model yields the closest head coefficients to the measurement data, the estimated radial forces by unsteady simulation are more or less similar to steady simulation. Since, the main focus of the current study is radial...
Figure 6. Comparison of predicted head and radial force using various turbulence models with the experimental data.

Figure 7. Different volute shapes: (a) standard volute, (b) 270° concentric volute, and (c) double-volute.

Figure 8. Comparison of effect of three volute shapes; namely, standard, concentric, and double-volute casings on pump characteristics.
force prediction at off-design conditions; the $k-\omega$ turbulence model with automatic near wall treatments was employed for calculations.

5. Results and discussion

To study the effects of volute geometry on the head, efficiency and radial forces in this investigation, seven various volutes were designed. The results obtained for different volutes are discussed in the following sections.

5.1. Volutes with various shapes

One of the main objectives was to minimize the radial force while having reasonable head and efficiency. The high radial force created by single volutes at low capacity is due to the interaction between volute cutwater and impeller outflow, which leads to flow separation and a pressure minimum downstream of the cutwater. This can be clearly observed from the radial force distribution in Figure 8(c). The radial force drops in a slightly downward curve with increasing flow rate until the point of best efficiency. At overload, the cross section is small for the flow. As a result, a low pressure zone is formed in the domain upstream of the discharge nozzle, as shown in Figure 9. Thus, the radial force increases (Figure 8(c)). Double-volute and concentric casings are employed to decrease the radial force, especially at off-design conditions. Generally, in double-volute, a second cutwater is located at an angle of 180° from the

![Figure 9. Pressure contours in volutes with various shapes.](image-url)

Figure 9. Pressure contours in volutes with various shapes.
first cutwater, as shown in Figure 7 (c). Consequently, as in Figure 9, similar pressure distributions are created in both partial volutes around the impeller periphery. This results in a reasonable reduction of the radial force at off-design conditions.

As shown in Figures 7(a), (b), and (c), three distinct volute shapes have been numerically investigated: a standard volute, concentric volute, and double-volute casing. The predicted output curves for three volutes are displayed in Figure 8. As shown in Figures 8(a) and (b), the concentric volute yields higher head and efficiency at low capacity. As expected, the standard volute has the best performance and the lowest radial force at the design point. The double-volute casing generates the minimum radial force at off-design conditions. Furthermore, the force at the design point is not very different from the standard one.

The 270° volute gives the highest radial force and the lowest efficiency at the design point. This volute generates lower radial force than the standard one at off-design conditions. The observations for the radial force are consistent
with Figure 9, which shows that the pressure values at off-design conditions are reasonable equal at opposite positions (with respect to center of rotation) around the impeller in the double-volute. It can be concluded that the radial force in the double-volute depends on the equality of pressure in opposite positions rather than pressure uniformity around the impeller, which is important in a standard volute. This means that the minimum radial force does not necessarily occur at the design point in the double-volute, as shown in Figure 8(c). For a pump which is likely to work at off-design conditions, double-volute could be chosen as the most promising.

5.2. Various concentric volutes

As shown in Figure 8, the radial force can be reduced at the low flow rate range by using a concentric casing. Also, these volutes generate considerably more head and efficiency when the pump operates mainly below the best efficiency point, which frequently occurs in process pumps. The large cutwater gap in these volutes reduces interaction effects between the impeller blades and volute tongue. Consequently, the pressure distribution around the impeller periphery becomes more uniform at low flow rate. At the design point, the pressure distribution is not uniform around the impeller due to the constant cross-sectional area

![Figure 12. Designs of three multivolute; namely, (a) Double-volute, (b) Triple-volute, and (c) Tetravolute.](image)

![Figure 13. Comparison of effect of three multivolute casing designs on pump performance.](image)
of the volute. It causes the minimum radial force point to be at a low flow rate instead of the design flow rate. The schematics of the three casings concentric at 180°, 270°, and 360° from the tongue are shown in Figure 10. The numerical results in Figure 11 show that the pump with the 180° volute yields higher head and efficiency at the design condition. However, the 360° volute displays more adequate head and efficiency at low flow rate. Interestingly, in the obtained curves, the best efficiency point has been shifted to 70% \( Q_d \) in the 360° concentric volutes. In a similar way, the minimum radial force occurred at 70% \( Q_d \) in the 180° and 360° concentric volutes. The 270° volute dominantly creates lower radial force at all flow rates and has moderately good head and efficiency. Considering all these aspects, the 270° volute could be selected as the best choice among these three concentric volutes.

5.3. Multivolute casings

For the first time, the application of multivolute casings is proposed to improve pump performance, especially at off-design conditions. Thus, as shown in Figure 12, three designs were generated, including a double-volute, triple-volute, and tetravolute casings. CFD simulations of the flow field within the volutes are illustrated in Figure 13.

As shown in Figure 13(a), the triple-volute provides the highest head at low capacity and the lowest radial force at the design and overload points, whereas the double-volute yields the lowest radial force at low capacity. As shown in Figure 13(c), the trend of radial force in the tetravolute is the most uniform at all flow rates. As a result, the triple-volute could be considered as the most appropriate alternative.

6. Conclusion

Quasi-steady flow through a low-\( N_a \) pump was numerically simulated by CFX 13.0 commercial code using a \( k-\omega \) turbulence model and automatic near wall treatments. The dependence of computational prediction was checked with the grid. Subsequently, numerical results including the pressure distribution and velocity profile were compared with available experimental measurements for output validation. After that, various concentric and multivolute casings were developed to study their hydraulic effects on head, efficiency, and radial reaction of a low-\( N_a \) centrifugal pump. As expected, the standard volute provides the most convenient head, efficiency, and radial force at the design point. In comparison with the standard and concentric volutes, the double-volute showed superiority in the radial force at off-design conditions, whereas the concentric volute created the highest head and efficiency at low capacity.

The 180° and 360° concentric volutes produced the highest head and efficiency at low capacity and the design capacity, respectively. The 270° concentric volute gives the lowest radial force at entire flow rate. Interestingly, in the 360° concentric volute, the best efficiency point was shifted to 70% \( Q_d \). The analysis of the various multivolute casings showed that the triple-volute is the most appropriate volute configuration due to its superior overhead, efficient performance at all capacities, and low radial reaction curve at the design flow rate and high flow rate. The absolute minimum radial force was generated by the double-volute at 65% \( Q_d \).

In this study, we did not carry out experimental measurements or unsteady simulation of the new geometry designs due to running time reduction and cost limitations. In future work, preparing the experimental data for new designs of volute geometries and comparing the results with quasi-steady and pure unsteady CFD results could complete this research.

Acknowledgements

The authors gratefully acknowledge the cooperation of Hydraulic Machinery Research Institute of Tehran University.

NOTATION

- \( b_2 \): Impeller outlet width, \( \text{mm} \)
- \( d_2 \): Impeller outlet diameter, \( \text{mm} \)
- \( r_0 \): Radius of volute wall midline, \( \text{mm} \)
- \( r^* \): \( r-r_4/r_0-r_4 \) Volute radial distance, non-dimensional
- \( u_2 \): Blade tip velocity, \( \text{m/s} \)
- \( V_r \): Radial velocity, \( \text{m/s} \)
- \( V_\theta \): Circumferential velocity, \( \text{m/s} \)
- \( F_r \): Radial force, \( \text{N} \)
- \( F_\theta \): Fr/(0.5\( \rho u_2^2 \pi d_2 b_2 \)) Radial force coefficient, non-dimensional
- \( g \): Gravity, \( \text{m/s}^2 \)
- \( H, H_n \): Head, Normal or design head, \( \text{m} \)
- \( k \): Turbulent kinetic energy, \( \text{m}^2/\text{s}^2 \)
- \( N \): Rotation speed, \( \text{r/min} \)
- \( N_d \): \( \text{NQ}^0.5 \text{H}^{-0.75} \) Metric specific speed, [rpm, \( \text{m}^3/\text{s}, \text{m} \)]
- \( P \): Static pressure, \( \text{Pa} \)
- \( P_{in} \): Inlet static pressure, \( \text{Pa} \)
- \( P^* \): \( (P-P_{in})/\rho u_2^2 \) Pressure coefficient, non-dimensional
- \( Q, Q_n \): Flow rate, Normal or design flow rate, \( \text{m}^3/\text{s} \)
- \( T_{ave} \): Average pressure and viscous moment on impeller walls, \( \text{N} \cdot \text{m} \)
- \( y^+ \): Distance from the wall, non-dimensional
- \( Z \): Number of blades

Greek Symbols

- \( \varepsilon \): Dissipation rate, \( \text{m}^2/\text{s}^3 \)
- \( \eta_{hyd} \): \( \mu gH / (T_{ave} \Omega) \) Hydraulic efficiency, %
- \( \rho \): Fluid density, \( \text{kg/m}^3 \)
- \( \theta, \theta v \): Angular coordinate, Angular position of tongue-tip, °
- \( \omega \): Specific dissipation rate, \( \text{1/s} \)
- \( \phi, \phi_n \): \( Q/(Q_d^2b_2) \) Flow coefficient, Flow coefficient at design flow rate \( Q_n \), non-dimensional
- \( \psi, \psi_n \): \( gH/(\Omega^2d_2^3) \) Head coefficient, Head coefficient at design head \( H_n \), non-dimensional
References

Asuaje, M., Bakir, F., Kouidri, S., Kenyery, F., & Rey, R. (2005). Numerical modelization of the flow in centrifugal pump: volute influence in velocity and pressure fields. *International Journal of Rotating Machinery*, 3, 244–255.

Barrio, R., Fernandez, J., Blanco, E., & Parrondo, J. (2011). Estimation of radial load in centrifugal pumps using computational fluid dynamics. *European Journal of Machines B/Fluids*, 30, 316–324.

Feng, J., Benra, F. K., & Dohmen, H. J. (2010). Application of different turbulence models in unsteady flow simulations of radial diffuser pump. *Forsch Ingenieurwes*, 74, 123–133.

Gonzalez, J. J., Fernandez, J., Blanco, E., & Santolaria, C. (2002). Numerical simulation of the dynamic effects due to impeller volute interaction in a centrifugal pump. *ASME Journal of Fluids Engineering*, 124, 348–355.

Jones, W. P., & Launder, B. E. (1972). The prediction of laminarization with a two Equation Model of Turbulence. *International Journal of Heat & Mass Transfer*, 15, 301–314.

Kelder, J. D. H., Dijkers, R. J. H., Vas Esch, B. P. M., & Kruyt, N. P. (2001). Experimental and theoretical study of the flow in the volute of a low specific-speed pump. *Journal of Fluid Dynamic Research*, 28, 267–280.

Liu, H. L., Liu, M. M., Dong, L., Ren, Y., & Du, H. (2012, August 19–23). Effects of computational grids and turbulence models on numerical simulation of centrifugal pump with CFD. Proceeding of 26th IAHR Symposium on Hydraulic Machinery and Systems, Beijing, China.

Menter, F. R. (1993, July 6–8). Multiscale model for turbulent flows. In *24th Fluid Dynamics Conference, Orlando, Florida*. Reston, VA: American Institute of Aeronautics and Astronautics.

Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32, 1598–1606.

Spence, R., & Amaral-Texeira, J. (2008). Investigation into pressure pulsations in a centrifugal pump using numerical methods supported by industrial tests. *Computers & Fluids*, 37, 690–704.

Spence, R., & Amaral-Texeira, J. (2009). A CFD parametric study of geometrical variations on the pressure pulsations and performance characteristics of a centrifugal pump. *Computers & Fluids*, 38, 1243–1257.

Torabi, R., & Nourbakhsh, S. A. (2011, July 24–29). Hydrodynamic design of the volute of a centrifugal pump using CFD. Proceeding of ASME-JSME-KSME Joint Fluids Eng. Conference, Hamamatsu, Shizuoka, Japan.

Yedidiah, S. (2008). A study in the use of CFD in the design of centrifugal pumps. *Engineering Applications of Computational Fluid Mechanics*, 2, 331–343.

Wilcox, D. C. (1986, January 6–8). Multiscale model for turbulent flows. In *AIAA 24th Aerospace Sciences Meeting, Reno, NV*. Reston, VA: American Institute of Aeronautics and Astronautics.

Zongying, P., Junjie, L., Shuai, L., & Shouqi, Y. (2011, July 24–29). Study on radial forces of centrifugal pumps with various collectors. Proceeding of ASME-JSME-KSME Joint Fluids Eng. Conference, Hamamatsu, Shizuoka, Japan.

Zhou, P. J., Wang, F. J., & Yang, M. (2012, August 19–23). Internal flow numerical simulation of double-suction centrifugal pump using DES model. Proceeding of 26th IAHR Symposium on Hydraulic Machinery and Systems, Beijing, China.