Applications of Different Turbulence Models in CFD Simulations of Mixed Flow Hydraulic Pump

Muhammad Tahir Jamil
College of Agricultural Science and Engineering, Hohai University

Yong Hai Yu
College of Agricultural Science and Engineering, Hohai University

Ahmad, Fiaz
Department of Agricultural Engineering, Bahauddin Zakariya University

https://doi.org/10.5109/4102466

出版情報：Proceedings of International Exchange and Innovation Conference on Engineering & Sciences (IEICES). 6, pp.73-78, 2020-10-22. Interdisciplinary Graduate School of Engineering Sciences, Kyushu University

Applications of Different Turbulence Models in CFD Simulations of Mixed Flow Hydraulic Pump

Muhammad Tahir Jamil[1], Yong Hai Yu [1], Fiaz Ahmad[2]
[1] College of Agricultural Science and Engineering, Hohai University, Nanjing 210098, China, [2] Department of Agricultural Engineering, Bahauddin Zakariya University, Multan 60800, Pakistan.
Corresponding author email: tahir@hhu.edu.cn; engr.tahirjamil@yahoo.com

ABSTRACT: *The computational fluid dynamics (CFD) is a cost-effective tool for derivation of performance characteristics of pumps. The turbulent flow comprises the long range of turbulence scales and this needs very fine grid. Turbulence models are used to capture the effect of turbulence and hence play vital role on accuracy of flow simulation. The selection of turbulence model requires understanding of physics of flow domain and validation with experimental results. This study discusses the applications of various turbulence models in CFD simulations. In the field of interest, two mainstreams exist. The first is the so-called Reynolds Average Navier Stokes (RANS) models. These models are most extensively utilized nowadays. The second mainstream presents large eddy simulations (LES). Second approach is important because they have the potential to model turbulence in a much more general and better way than the RANS models.*

Keywords: Mixed flow pump; Turbulence models; Computational fluid dynamics; *RANS models.*

1. INTRODUCTION

Pumps are used for a wide range of industrial and residential applications. The need for higher-efficiency compact pumps with a wider spectrum of operating conditions requires a better understanding of the local and instantaneous characteristics of the flow region inside the pump. A variety of experiments have been carried out for this reason. Any laboratory experiments have only addressed the movement of the impeller, which discharges through the asymmetric collector. Acosta and Bowerman [1], Howard and Kittmer [2] have observed that the movement between blades is compatible with the nature movement. At flow less than 25 to 40 percent of the plan flow however, a separation or recirculation flow emerged.

Due to the pressure gradient, secondary flux, boundary layer inertia, rotary stall, and an unequal peripheral pressure division, the flow in the pump is observed to remain segregated. In addition, the pump has a complex geometry that makes it impossible to calculate several variables. It is therefore very important to simulate the flow inside the pump. Using a graphical approach, the dynamic mechanics of fluid flow can be easily observed, and it is simpler to research the influence of various vector parameters on steady and unsteady physical phenomena. Therefore, the production process is speeding up, reducing the time and saving money for the finished product.

In the field of fluid flow phenomena, computational fluid dynamics represents a wide range of numerical research. Progression in the field of CFD simulations is highly reliant on the advancement of computer related technologies and on our understanding and solution of ordinary and partial differential equations (ODEs and PDEs). But CFD is much more than just informatics and numerical analysis. Because direct numerical solution of complex flows involves a huge amount of computational power to solve such problems in real-like conditions, it is considerably dependent on the applied physical models. This can only be achieved from understanding the physical processes, which persist under certain circumstances [3,4].

Turbulences tend to dominate all other flow phenomena when they are present in a particular flow. That's why effective turbulence modelling greatly improves the efficiency of numerical simulations.

For several different purposes it is possible to use CFD simulation such as calculating hydraulic losses in a pump, calculating performance characteristics, calculating hydrodynamic forces, investigating cavitation in a pump, optimizing a geometry, etc. [5]. Works of [6], [7] and [8] dealt with the CFD analysis of the centrifugal pump. Output characteristics are described in these works as outcomes of CFD simulation. Accurate simulation of CFD depends on various factors. The most important factors are computational mesh density and quality, the turbulence model chosen etc. The effect of mesh density on CFD simulation accuracy was investigated in work. In the CFD analysis of a centrifugal pump, Gullich [5] pays attention to the precision of turbulence models. The turbulence model k-ε was not recommended as appropriate for the CFD analysis of the centrifugal pump in this work. Otherwise SST model was assumed as a good choice to achieve accurate results. The objective of this contribution is to review the applications of turbulence models for CFD simulations of mixed flow hydraulic pumps in a pumping station.

1.1 Ideal turbulence model

The resolution of the CFD problem usually involves four main elements: grid and geometry generation, configuration of the physical model, resolution, and post processing of the calculated results. The technique by which geometry and grid are created, the set problem is calculated and the mode the obtained data is displayed is well recognized. Precision theory is presented. Unfortunately, this is not applicable when creation of a physical turbulence flow model. The concern is that people want to simulate very complex processes with the simplest model possible [7]. Therefore, in the simulation equations the optimal model should have the least complexity representing the substance of the corresponding physics.

1.2 Complexity of the turbulence models

The complexity of the various turbulence models will differ considerably depending on the data one want to evaluate and analyze using such numerical simulations. The complexity is owing to the Navier-Stokes equation (N-S equation). In essence, the N-S equation is non-linear, time dependent, 3D PDE. The uncertainty of laminar flow at high Reynolds (Re) numbers can be considered to be turbulence. Such a source of instability forms an interaction in the N-S equation between nonlinear inertial conditions and viscous terms. These interactions are rotational, time-dependent and entirely 3D. Rotational and 3D interactions are interconnected by vortex stretching. Two-dimensional space cannot stretch the Vortex. That's why 2-dimensional estimates are not adequate for turbulent phenomena.

Turbulence is commonly referred to as a random phenomenon. Therefore, there is no practicable deterministic solution. Any of the characteristics of turbulence can be analysed using statistical techniques. These correlation functions between flux variables are introduced. However, it is difficult to pre-estimate these associations.

Another significant aspect of the turbulent flow is that the vortex structures are moving through the flow. It's typically a very long life. Consequently, such turbulent quantities are not locally defined. Basically, this means that there is obviously a significant role for the upstream flow history.

2. CLASSIFICATION OF TURBULENT MODELS

Turbulent flows can now be calculated with various methods. Either by solving or directly computing the equations of Reynolds-averaged Navier-Stokes with appropriate models for turbulent measures. The following are the key techniques.

- Reynolds-Averaged Navier-Stokes (RANS) Models
- Computation of fluctuating quantities

3. APPLICATIONS OF TURBULENCE MODELS

3.1 Reynolds-Averaged Navier-Stokes (RANS) models

With the help of the recent development of computational resources, numerical analysis based on 3-dimensional RANS equations has been extensively used for turbomachine flow analysis. By minimising the number of experimental evaluations, this analysis lowered the costs and the time of the pump design.

Kim et al., [8] conducted a 3D RANS analysis, which established that the efficiencies of the mixed-flow pump had an effect on two geometric variables related to the discharge area and vane length in the diffuser. For evaluation, finite-volume approaches were discretized in the RANS equations for shear stress transport (SST) turbulence model. Compared to the experimental findings shown in Figure 1, numerical results were validated for flow analysis. The efficiency and head estimates were well matched with the performance test results.

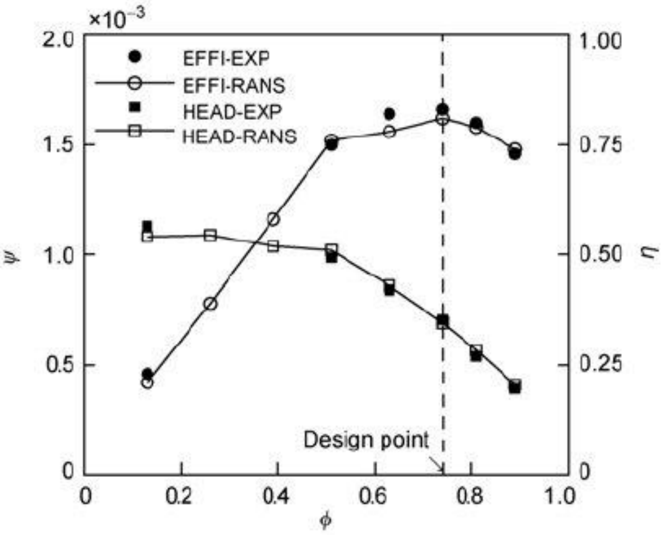

Fig. 1. Validation of the flow analysis [8].

Mao et al. [9] selected four common turbulence models (SST, *k-ω*, *k-ε and RNG k-ε)* to simulate the flow in mixed flow pump. The pump's efficiency is confirmed by experimentations in an open test rig. Figure 2 demonstrates the comparative analysis of the delivery head curves determined in computational measurements and tests for all nominal speed environments. The delivery head agrees very well on the design and overload conditions of the four models, although they significantly differ on the part-load conditions. The SST and *k-ω* models forecast more precisely than the *k-ε* and RNG *k-ε* models.

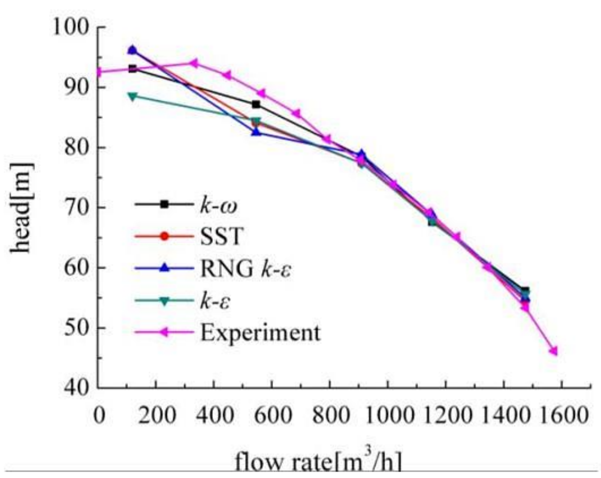

Fig. 2. Comparison of head curves for four models [9].

Hlbočan [10] carried out a comparison of two turbulence models (i.e., k-ε and SST) in CFD simulation of a mixed flow pump. The efficiency features of the pump were determined by CFD analysis. They were then compared with the same characteristics evaluated on the basis of the data collected from the experimental measurements. Figure 3 and 4 show comparison of the efficiency of the pump obtained from experiments and CFD simulation with SST and k – ε turbulence models. $Q - \eta$ curve obtained from results of CFD simulation with SST model is clearly instable. The profile of the $Q - \eta$ curve obtained from CFD simulation is very similar to the profile of the $Q - \eta$ curve, which was obtained from experiments. CFD simulation with k - ε turbulence model exhibited that $Q - \eta$ curve of the pump is stable. It doesn't respond to the reality. In this case SST model seems to be more suitable for CFD simulation of the flow in mixed-flow hydraulic pump.

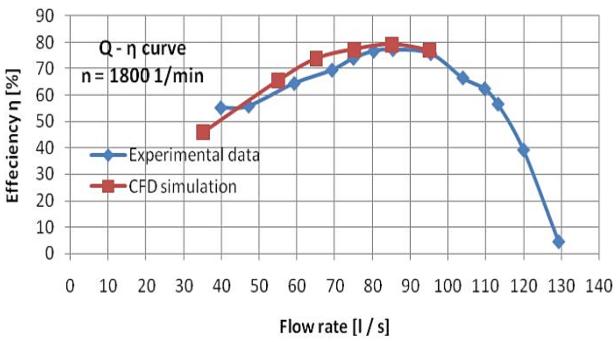

Fig. 3. Comparison of the Q – η for SST model [10].

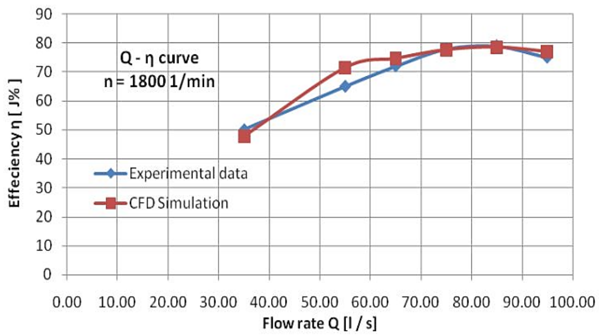

Fig. 4. Comparison of the Q – η for k-ε model [10].

Yang et al. [11] carried out a 3D study of the low head mixing flow pumping system. For analysis, finite volume approximations were discretized between the RANS equations and SST turbulence model. The results have shown that the flow on the double-helix volute line is a spiral movement that combines axial and rotating flows; a symmetric distribution of the static volute pressure; axial velocity distribution and the average speed-weighted swirl angle of the outlet segment are very high. Figure 5 illustrates the curve of the flow lift and flow efficiency. A good agreement for the lifting head was noticed between the predicted and the measured results, compared with the numerical simulation results and the experimental findings. Although the estimate and the calculated results at non-design phases differ in performance, the numerical results cannot accurately determine the pumping system output beyond the highly efficient region that requires more developments.

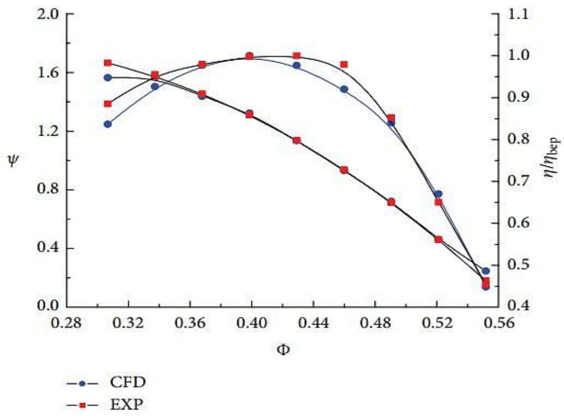

Fig. 5. Performance curves of pumping system [11].

Huang et al. [12] used the *k-ω* SST turbulence model and the mass transfer cavitation model based on the Rayleigh-Plesset equation to analyze the cavitation due to turbulent flow in a mixed-flow water jet pump. The estimated hydraulic productivity, besides the cavitation output, are fairly consistent with experimental outcomes. On the basis of the illustration, formation of cavitation increases the intensity of vorticity and the inconsistency of flow in a mixed-flow water jet device. The vortices are generally located during cavity interface, especially during cavitation on the downstream interface. Figure 6 illustrates the decline in output when the cavitation is being developed in the design operating conditions. The efficiency and head coefficient declined at low cavitation, as shown in figure 6. In this simulation the effects of cavity on pump effectiveness, while critical cavitation values which have unpredictably collapsed pump performance, are accurately predicted between measurement and the experiment. Such a differentiation could be due to the weakness of the cavitation model on the grounds that water and vapour distributions are homogeneous.

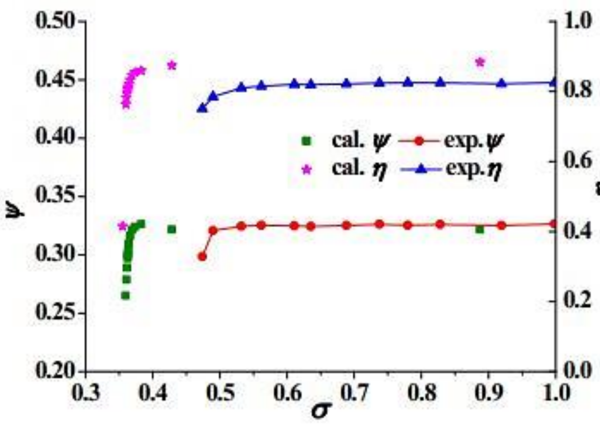

Fig. 6. Cavitation performance curves at the design operation point [12].

The ability of the numerical approach to capture the dynamical and highly unstable flow effect within the pump was studied by Gonzalez et al. [13]. The sliding mesh technique provided by the CFD code, FLUENT, was used with Standard k-ω model. Their findings are well supported by experimental data. Barrio et al. [14] estimated the efficiency of the centrifugal pump and predicted a unsteady distribution of the volute pressure. Comparison was made with many turbulence models including Allmaras, standard k-ω, Reynold Stress Model (RSM) and k- ω. The standard k- ω can be more reliable than other models in their predictions of flow.

3.2 Computation of fluctuating quantities

Predicting a physical structural response requires the concurrent measurement and fluctuation of pressure activity, i.e. turbulent fluctuations. High frequency (usually stochastic). In the computationally intensive approach known as direct numerical simulation, the flow behaviour of the Navier-Stokes equations can be calculated directly without modelling. Unfortunately, the quantity of floating point DNS scale calculations like Re^3 (Reynolds cubed number), Hoffman and Johnson (2006) [15] indicates that even on today's computers, realistic calculations are impossible for engineering simulation. Large Eddy Simulation (LES), offers an alternate but needs extensive modelling specifically on the scale of interest [16]. Modeling based attempts to envisage turbulent pressure and velocity variations, are therefore of prodigious importance.

So as to evaluate the characteristics of turbulent flow in a mixed-flow pump guide vane, various indicator points are placed in the flow guide vane, the numerical value of the pump in the whole flow field will be measured using LES, a sub grid model and a sliding-mesh technology [17]. Because of the unsteady simulation of the mixed-flow pump on the model LES and sub-grid, LES can precisely forecast the positive slope of the head capacity curve, comparing its prediction with its experimental performance. And the instability feature of the mixed flux pump is studied on the basis of prior studies. The experiment outcomes shows that the LES will estimate the positive inclination characteristic of the head and the capacity curve. (Figure 7).

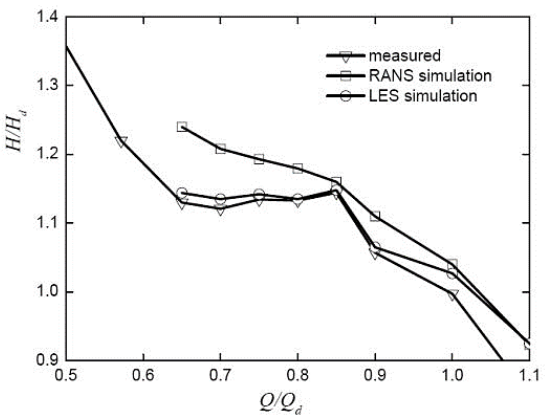

Fig. 7. Performance prediction and experimental verification [17].

LES was performed by Kobayashi (2009) [18] using a cavitation model to measure unsteady flow with a closed-type impeller for a mixed-flow pump. To assess a computational accuracy, a comparison of numerical and experimental results has been made. In numerical and experimental results, positive impulsive peaks were present in the absolute pressure on the front of the suction side of the blade surface. From simulation, it was indicated that these peaks were instigated by a cavitation that had contracted and disappeared round the leading edge. Simulation determined the absolute pressure with a 10% error.

The FSI approach based on the LES model is presented by Pittard (2003) [19]. This test the turbulent flow changing instantaneously. The LES model based results indicate that these variations are related to tube vibration. The relation between the standard deviation in pressure on the wall of the pipe and the rate of flow is almost quadratic. A close correlation between the vibration of the tube and the rate of flow is also shown. The flow field for turbulence in pipe has been modelled with both approaches to demonstrate the variances amongst RANS and LES (Figure 8). It was also inferred from this research that the pressure fluctuations on the pipe wall have a quadratic relation to flow rate.

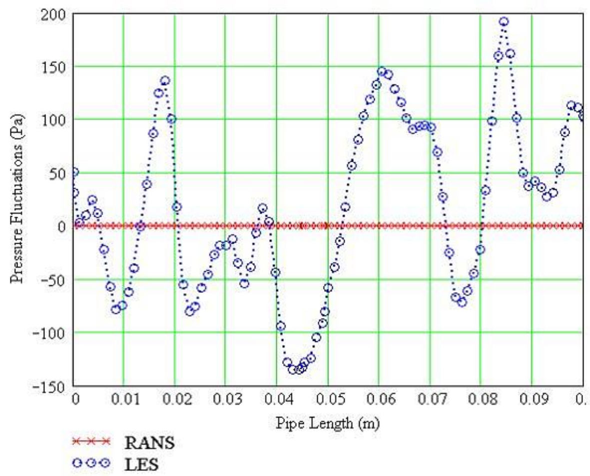

Fig. 8. Pressure fluctuations computed by RANS and LES based models [19].

4. RANS versus LES

The turbulent flow can be made up of several different characteristics. It is also very critical that the CFD model will predict as many of them as possible. Turbulent models are typically evaluated by simulating a flow through the bluff body [20]. In particular example flow past a square block is analyzed. An example of such a flow is shown in Figure 9.

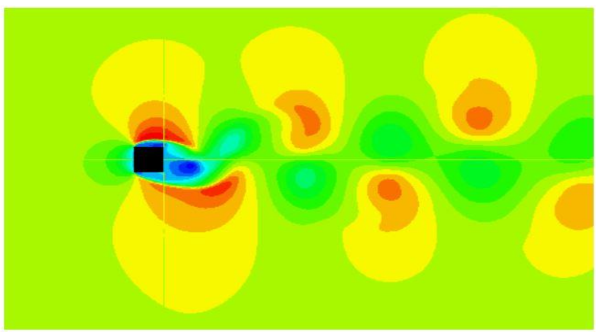

Fig. 9. Flow past a square block in 2D [20].

The streamlines projected by LES, EASM (a sophisticated RANS model) and RANS are shown in Figure 10. The strong influence of the model used in the RANS calculation can clearly be seen. The EASM shows similar topology as the LES, however, the extent of the recirculation zones is different. RANS expects a slightly greater recirculation zone and a much greater wake region.

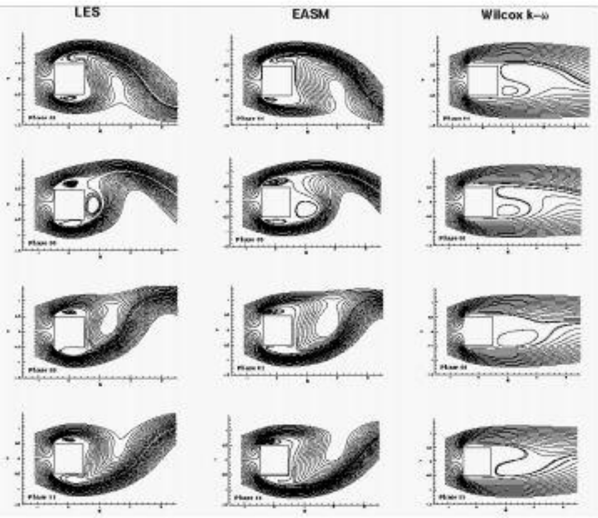

Fig. 10. Streamlines predicted by LES, EASM and RANS [20].

5. CONCLUSION

In recent decades, CFD has been a major engineering tool. In the light of the advances in computing technology, CFD now seems to be capable of dealing with manufacturing applications at an acceptable cost and processing time. The tangible advantage of CFD therefore depends on its flow accuracy and complexity. Since many engineering interest flows are turbulent, careful turbulence control is vital to CFD 's performance. The Newtonian fluid field is defined by the Navier-Stokes equation in its entirety. However, the turbulent flows are slowly changing. Fine grids and time scales are needed to overcome these minor motions, rendering direct simulation difficult for the high Reynolds numbers. The aim of LES is to reduce the dependency on the turbulence model. As a consequence, the main portion of the flow is represented without any templates and must be determined by a grid. A model is only needed for scales that are smaller than the grid resolution. As a result, the LES method is more complex to calculate than the RANS method. The RANS models have a running period of just around 5 per cent of the LES. Sophisticated RANS models such as EASM are capable of capturing critical flow features correctly. They are already a valuable tool for industrial design at low computational costs.

6. REFERENCES

[1] Acosta, A.J.; Bowerman, R. An experimental study of centrifugal pump impellers. **1955**.

[2] Howard, J.; Kittmer, C. Measured passage velocities in a radial impeller with shrouded and unshrouded configurations. **1975**.

[3] Gülich, J.F. *Centrifugal pumps*; Springer: 2008; Vol. 2.

[4] Cheah, K.; Lee, T.; Winoto, S.; Zhao, Z.J.I.J.o.R.M. Numerical flow simulation in a centrifugal pump at design and off-design conditions. **2007**, *2007*.

[5] Patel, K.; Ramakrishnan, N. CFD analysis of mixed flow pump. In Proceedings of International ANSYS Conference Proceedings.

[6] Perez, J.; Chiva, S.; Segala, W.; Morales, R.; Negrao, C.; Julia, E.; Hernandez, L. Performance Analysis of Flow in a Impeller-Diffuser Centrifugal Pumps using CFD: Simulation and Experimental Data Comparisons. In Proceedings of European Conference on Computational Fluid Dynamics ECCOMAS CFD 2010; pp. 1-18.

[7] Davidson, L. An introduction to turbulence models. **2015**.

[8] Kim, J.-H.; Ahn, H.-J.; Kim, K.-Y.J.S.i.C.S.E.T.S. High-efficiency design of a mixed-flow pump. **2010**, *53*, 24-27.

[9] Mao, J.; Yuan, S.; Pei, J.; Zhang, J.; Wang, W. Applications of different turbulence models in simulations of a large annular volute-type pump with the diffuser. In Proceedings of IOP Conference Series: Earth and Environmental Science; p. 022019.

[10] Hlbočan, P. COMPARISON OF K-ε AND SST TURBULENCE MODELS IN CFD SIMULATION OF THE FLOW IN A MIXED-FLOW PUMP.

[11] Yang, F.; Liu, C.J.I.S.R.N. The flow simulation and experimental study of a large low-head mixed-flow pumping system. **2013**.

[12] Huang, R.; Ji, B.; Luo, X.; Zhai, Z.; Zhou, J.J.J.o.M.S.; Technology. Numerical investigation of cavitation-vortex interaction in a mixed-flow waterjet pump. **2015**, *29*, 3707-3716.

[13] Gonza´ lez, J.; Ferna´ ndez, J.n.; Blanco, E.; Santolaria, C.J.J.F.E. Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump. **2002**, *124*, 348-355.

[14] Barrio, R.; Parrondo, J.; Blanco, E.J.C.; Fluids. Numerical analysis of the unsteady flow in the near-tongue region in a volute-type centrifugal pump for different operating points. **2010**, *39*, 859-870.

[15] Hoffman, J.; Johnson, C.J.C.M.i.A.M.; Engineering. A new approach to computational turbulence modeling. **2006**, *195*, 2865-2880.

[16] You, D.; Moin, P.J.P.o.F. A dynamic global-coefficient subgrid-scale eddy-viscosity model for large-eddy simulation in complex geometries. **2007**, *19*, 065110.

[17] Yi-bin, L.; Ren-nian, L.; Xiu-yong, W.J.I.J.o.F.M.; Systems. The numerical simulation of unsteady flow in a mixed flow pump guide vane. **2013**, *6*, 200-205.

[18] Kobayashi, K.; Chiba, Y. Numerical simulation of cavitating flow in mixed flow pump with closed type impeller. In Proceedings of Fluids

Engineering Division Summer Meeting; pp. 339-347.
[19] Pittard, M.T.; Blotter, J.D. Numerical modeling of LES based turbulent flow induced vibration. In Proceedings of ASME International Mechanical Engineering Congress and Exposition; pp. 141-148.
[20] Celic, A.J.A.S.V. Performance of Modern Eddy-Viscosity Turbulence Models. **2004.**