The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM)
ISSN: 2602-3199

The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 2022

Volume 21, Pages 202-213

IConTES 2022: International Conference on Technology, Engineering and Science

CFD Investigation on Three Turbulence Models for Centrifugal Pump Application

Abdessalam OTMANI
Badji Mokhtar University

Slimane NIOU
Badji Mokhtar University

Sahrane SARA
Badji Mokhtar University

Azzeddine DEKHANE
Badji Mokhtar University

Salaheddine AZZOUZ
Badji Mokhtar University

Abstract: This paper highlight the influence of three numerical turbulence models on the convergence and the performance of flow simulation. The computational comparative study was realized using the COMSOL Multiphysics 5.5 code. Turbulence was generated numerically in a centrifugal water pump using the k−ϵ, k−ω and k−ω SST models. However, the geometry was performed on SolidWorks due to its complexity. The flow modelling was mainly based on the resolution of the stationary Navier—Stokes equations. The effects of the tested models on CFD numerical simulation were examined. It was found that the best calculation precision was obtained using the K−ω model, while the lowest was provided by the K−ω SST model. However, a very low calculation cost was obtained by the latter. As well as better pumping performance were recorded.

Keywords: Turbulence models, Fluid flow simulation, Centrifugal pump

Introduction

During the past years, researchers devoted a great deal of effort to formulation and testing of turbulence models (Wilcox, 1991). In fact, computational fluid dynamics tools are becoming standard in many fields of engineering involving flow of gases and liquids; numerical simulations are used both in the design phase to select between different concepts and in the production phase to analyze performance (Samy M & Mofreh H, 2011). Turbulence modeling is one of three key elements in Computational Fluid Dynamics (CFD). Very precise mathematical theories have evolved for the other two key elements, viz., grid generation and algorithm development. By its nature, far less precision has been achieved in turbulence modeling. Since the objective is to approximate an extremely complicated phenomenon (Wilcox, 1993).

The most popular turbulence models are the standard k–e model, low-Re k–e model, RNG k–e model, standard k–w model, and SST k–w model (Samy & Mofreh, 2011). In fact, it has been confirmed that the calculation results are quite different with different turbulence models, and the result under K epsilon EARSM model is better than four other models : K-Epsilon, SSG Reynolds Stress, RNG K-Epsilon, K-Omega) (Liu et al., 2012). While the k-ω model gave more realistic velocity profiles, consistently produced values that were too high for the turbulent shear stress (Menter, 1994). However, testing has shown that the k-e Realizable and transition SST turbulence models give the best results in the calculation of supersonic flows, typical for advanced jet engines (Pavlovich & Victorovich, 2013). Furthermore, comparison shows that the realizable k-ε predicts the flow within centrifugal pump with acceptable accuracy (Selim et al., 2016). Regarding the mean flow field of the pump, the SAS model does not show an advantage over the SST model (Markus et al., 2020).

Pump Characteristics

The centrifugal pump considered in this study contains one suction and a single discharge canal as well as a circular spiral shaped volute/casing (Figure 1). The overall plan of the pump is shown in figure 1 and the main characteristics are resumed in table 1. The geometry was realized on SolidWorks using: (Brozoski, 2018).

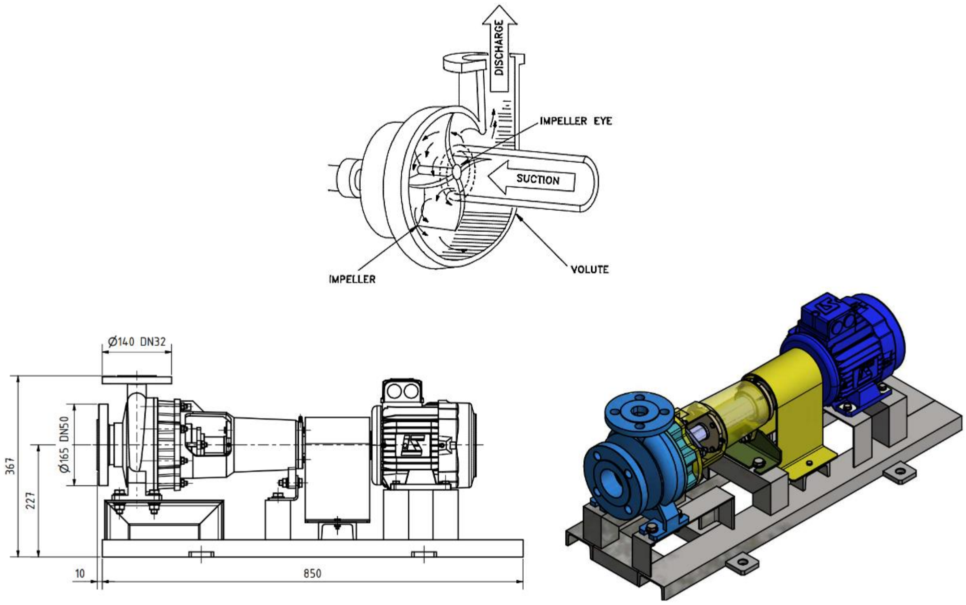

Figure 1. Centrifugal pump general view (Paul et al., n.d) and overall plan of the studied centrifugal pump (Brozoski, 2018)

Table 1. Main characteristics of the studied centrifugal pump

Suction diameter	Discharge diameter	Inlet pressure	Outlet pressure	rpm
60 mm	55 mm	0.5 (bar)	2 (bar)	720 (tr/min)

Numerical Model

Impeller Design

A centrifugal pump converts rotational energy, often from a motor, to energy in a moving fluid. The two main parts that are responsible for the conversion of energy are the impeller and the casing. While passing through the impeller, the fluid is gaining both velocity and pressure (Srivastava, 2020). Consequently, the impeller is one of the essential parts of a centrifugal pump, since it is the source of kinetic energy. The characteristics of the studied impeller are summarized in the Table 2.

Table 2. Impeller main characteristics

Outer diameter	Inner diameter	Number of blades	Tilt angle
154 mm	60 mm	8	60

Impellers can be open, semi-open, or enclosed (Figure 2). Another point is that Impellers can be either single-suction or double-suction. A single-suction impeller allows liquid to enter the center of the blades from only one direction. A double-suction impeller allows liquid to enter the center of the impeller blades from both sides simultaneously. (Paul et al., n.d). In fact, an enclosed impeller with a single-suction is considered in this study. The design (Figure 4) was carried out on SolidWorks using real dimensions shown in Figure 3.

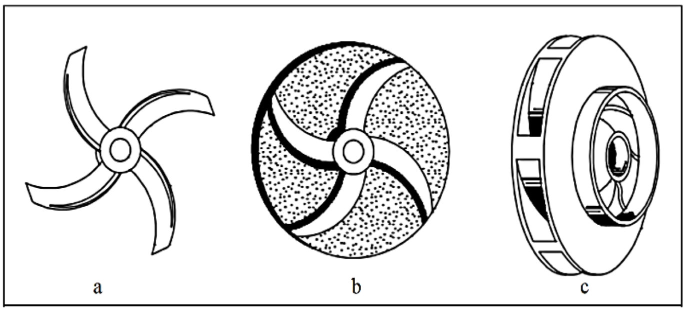

Figure 2. (a) Open, (b) Semi-Open, and (c) Enclosed Impellers

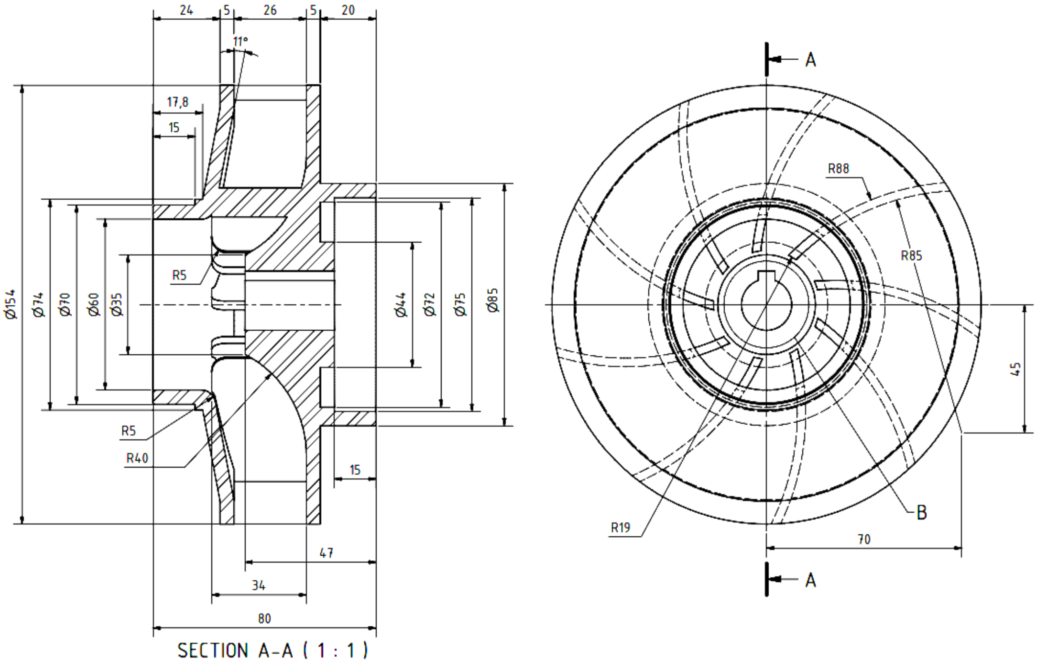

Figure 3. Studied impeller detailed plan (Brozoski, 2018)

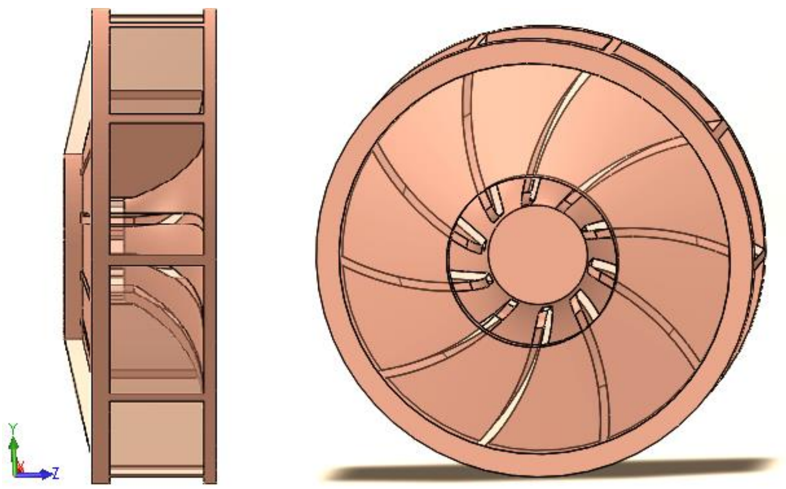

Figure 4. 3d view of the designed Impeller

Casing Design

The pump casing provides a pressure boundary for the pump and contains channels to properly direct the suction and discharge flow (Paul et al., n.d). Furthermore, it slows the flow of the liquid. Therefore, according to Bernoulli's principle, the volute converts kinetic energy into pressure by reducing speed while increasing pressure. The volute/casing considered in this study was designed based on reel dimensions shown in figure 5.

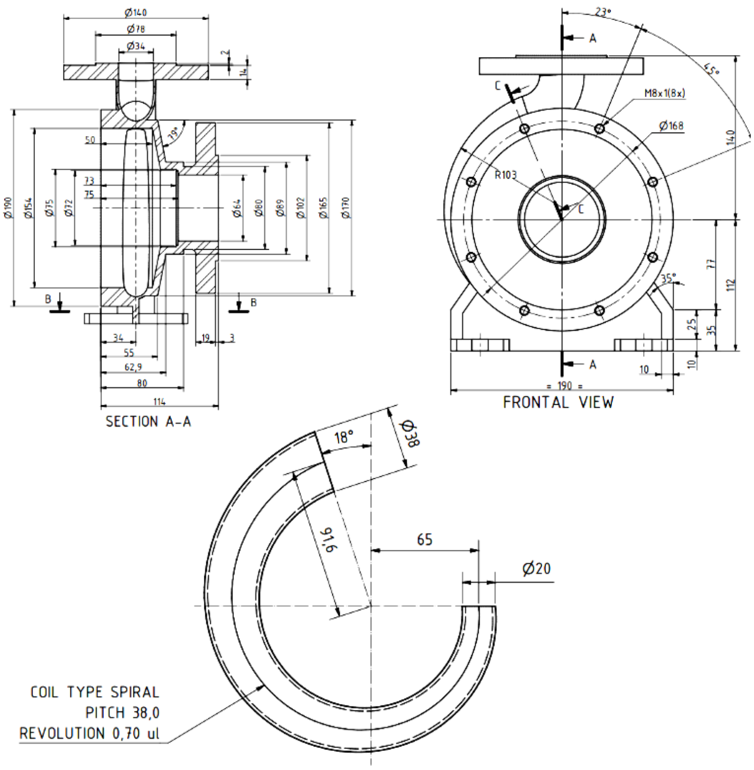

Figure 5. Casing detailed plan (Brozoski, 2018)

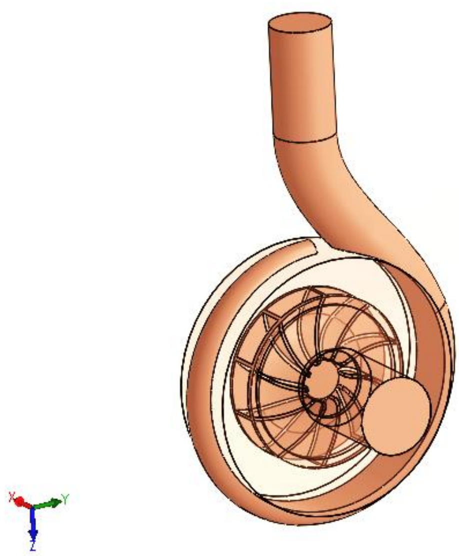

Figure 6. 3D view of the global final geometry

Mesh Structure

A free tetrahedral mesh was applied for the global final geometry. Mesh characteristics are summarized in table 3 and the final mesh structure is shown in Figures 7.

Table 3. Impeller main characteristics

Nodes number	Max element size	Min element size
88528	0.0251	0.00183

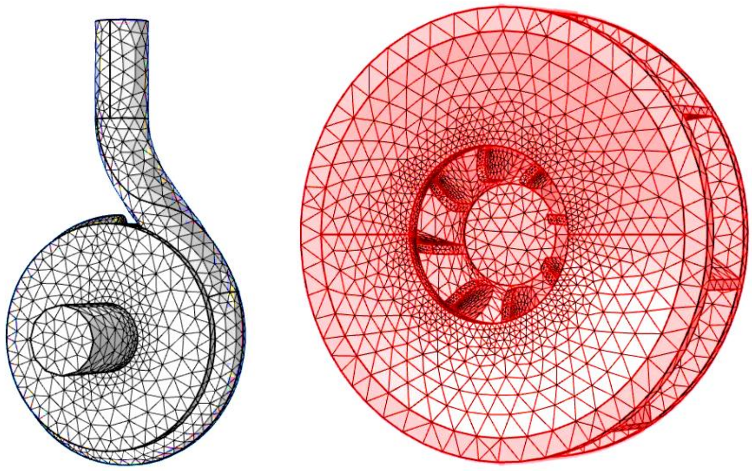

Figure 7. Final geometry as a free tetrahedral Mesh structure

Frozen Rotor Study

The Frozen Rotor study was used to compute the velocity, pressure and turbulence in Comsol-Multiphysics. The frozen rotor approach assumes that the flow in the rotating domain, expressed in the rotating coordinate system, is fully developed. Therefore, it's generally used in rotating machinery and is a special case of a Stationary study. The rotating parts are kept frozen in position, and the rotation is accounted for by the inclusion of centrifugal and Coriolis forces. This study is especially suited for flow in rotating machinery where the topology of the geometry does not change with rotation. It is also used to compute the initial conditions for time-dependent simulations of flow in rotating machinery (multiphysics).

Governing Equations

The k-ε Turbulence Model

The k-ε model is one of the most used turbulence models for industrial applications. This module includes the standard k-ε model. The model introduces two additional transport equations and two dependent variables: the turbulent kinetic energy, k, and the turbulent dissipation rate, ε. The turbulent viscosity is modeled as (multiphysics) :

$$\mu_T = \rho C_\mu \frac{k^2}{\varepsilon} \tag{1}$$

where Cμ is a model constant.

The transport equation for k reads:

$$\rho \frac{\partial k}{\partial t} + \rho u.\nabla k = \nabla.((\mu + \frac{\mu_T}{\sigma_k})\nabla k) + p_k - \rho\varepsilon \tag{2}$$

where the production term is :

$$p_k = \mu_T (\nabla u:(\nabla u + (\nabla u)^T) - \frac{2}{3}(\nabla.u)^2) - \frac{2}{3}\rho k \nabla.u \tag{3}$$

The transport equation for ε reads :

$$\rho \frac{\partial \varepsilon}{\partial t} + \rho u.\nabla\varepsilon = \nabla.((\mu + \frac{\mu_T}{\sigma_\varepsilon})\nabla\varepsilon) + C_{\varepsilon 1}\frac{\varepsilon}{k}p_k - C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \tag{4}$$

The model constants in Equation 1, Equation 2, and Equation 4 are determined from experimental data (Wilcox,1998) and the values are listed in Table 4 (Comsol-multiphysics, 2020).

Table 4. model constants

Constant	Value
c_μ	0.09
$c_{\varepsilon 1}$	1.44
$c_{\varepsilon 2}$	1.92
σ_k	1.0
σ_ε	1.3

The k-ω Turbulence Model

The k-ω model solves for the turbulent kinetic energy, k, and for the dissipation per unit turbulent kinetic energy, ω. The CFD Module has the Wilcox (1998) revised k-ω model (Comsol-multiphysics, 2020).

$$\rho\frac{\partial k}{\partial t} + \rho u.\nabla k = p_k - \rho\beta^* k\omega + \nabla.((\mu+\sigma^*\mu_T)\nabla k) \quad (5)$$

$$\rho\frac{\partial \omega}{\partial t} + \rho u.\nabla \omega = \alpha\frac{\omega}{k}p_k - \rho\beta^*\omega^2 + \nabla.((\mu+\sigma\mu_T)\nabla \omega)$$

Where

$$\mu_T = \rho\frac{k}{\omega} \quad (6)$$

and

$$\alpha = \frac{13}{25}\ ;\ \beta = \beta_0 f_\beta\ ;\ \beta^* = \beta_0^* f_\beta\ ;\ \sigma = \frac{1}{2}\ ;\ \sigma^* = \frac{1}{2}$$

$$\beta_0 = \frac{13}{125}\ ;\ f_\beta = \frac{1+70\chi_\omega}{1+80\chi_\omega}\ ;\ \chi_\omega = \left|\frac{\Omega_{ij}\Omega_{jk}\Omega_{ki}}{(\beta_0^*\omega)^3}\right|;\ \beta_0^* = \frac{9}{100}$$

$$f_\beta = \begin{cases} 1 & \chi_k \le 0 \\ \dfrac{1+680\chi_k^2}{1+400\chi_k^2} & \chi_k > 0 \end{cases} \qquad \chi_k = \frac{1}{\omega^3}(\nabla k.\nabla\omega) \quad (7)$$

where in turn Ω_{ij} is the mean rotation-rate tensor

$$\Omega_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} - \frac{\partial \bar{u}_j}{\partial x_i}\right) \quad (8)$$

and S_{ij} is the mean strain-rate tensor

$$S_{ij} = \frac{1}{2}\left(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i}\right) \quad (9)$$

P_k is given by Equation 3. The following auxiliary relations for the dissipation, ε, and the turbulent mixing length, l_*, are also used:

$$\varepsilon = \beta^*\omega k\ ;\ \ l_{mix} = \frac{\sqrt{k}}{\omega} \quad (10)$$

The SST Turbulence Model

To combine the superior behavior of the k-ω model in the near-wall region with the robustness of the k-ε model, Menter (1994) introduced the SST (Shear Stress Transport) model which interpolates between the two models. The version of the SST model in the CFD Module includes a few well-tested (Menter et al., 2003) modifications, such as production limiters for both k and ω, the use of S instead of Ω in the limiter for μT and a sharper cut-off for the cross-diffusion term. It is also a low Reynolds number model, that is, it does not apply wall functions. “Low Reynolds number” refers to the region close to the wall where viscous effects dominate. The model equations are formulated in terms k and ω, (Comsol-multiphysics, 2020).

$$\rho\frac{\partial k}{\partial t} + \rho\mathbf{u}.\nabla k = P - \rho\beta_0^* k\omega + \nabla.((\mu+\sigma_k\mu_T)\nabla k)$$

$$\rho\frac{\partial \omega}{\partial t} + \rho\mathbf{u}.\nabla \omega = \frac{\rho\gamma}{\mu_T}P - \rho\beta\omega^2 + \nabla.((\mu+\sigma_\omega\mu_T)\nabla \omega) + 2(1 - f_{v1})\frac{\rho\sigma_{\omega2}}{\omega}\nabla\omega.\nabla k \qquad \textbf{(11)}$$

Where,

$$P = min\, P_k, 10\rho\beta_0^* k\omega \qquad (12)$$

And P_k is given in Equation 3. The turbulent viscosity is given by,

$$\mu_T = \frac{\rho\, a_1 k}{\max(a_1\omega, Sf_{v2})}$$

Where S is the characteristic magnitude of the mean velocity gradients,

$$S = \sqrt{2\, S_{ij}S_{ij}}$$

The model constants are defined through interpolation of appropriate inner and outer values,

$$\phi = f_{v1}\phi_1 + (1 - f_{v1})\phi_2 \quad \text{for} \quad \phi = \beta, \gamma, \sigma_k, \sigma_\omega$$

The interpolation functions f_{v1} and f_{v2} are defined as,

$$f_{v1} = \tanh(\theta_1^4)$$

$$\theta_1 = \min\left[\max\left(\frac{\sqrt{k}}{\beta_0^*\omega\, l_w}, \frac{500\mu}{\rho\omega\, l_w^2}\right), \frac{4\rho\mu\sigma_{\omega2}k}{CD_{k\omega}\, l_w^2}\right]$$

$$CD_{k\omega} = \max\left(\frac{2\rho\sigma_{\omega2}}{\omega}\nabla\omega.\nabla k, 10^{-10}\right)$$

where l_w is the distance to the closest wall.

Results and Discussion

Simulation Convergence

Convergence plays a key role in the accuracy of the obtained results using numerical techniques such as finite element analysis. This is the reason why convergence needs to be highlighted. Therefore, the convergence of the studied turbulence models must be analyzed, in order to determine the optimal model in terms of precision and computational cost. On The one hand, the best calculation precision and the lowest error ,5x10^{-7}, was achieved using the k-ω model. While the k-ε model provided an error of 2.6x10^{-6} and the k-ω SST model could only achieve 2x10^{-5} (Figure 8). On the other hand the latter's convergence was accomplished in just 25min8s. While the k-ω model required 43min40s. Even more, the k-ω model exceeded 50 minutes (results was obtained using an intel® core (TM) i5 3470 cpu 3.20ghz processor). A more significant comparison is shown in the table below :

Table 5. Convergence parameters of the studied turbulence models

Model	Calculation time	Iteration number	Error
K-*w*	50 min, 36 s	361	5x10-7
K-*w* SST	25 min 8 s	325	2x10-5
K-*w*	43 min, 40 s	355	2.6x10-6

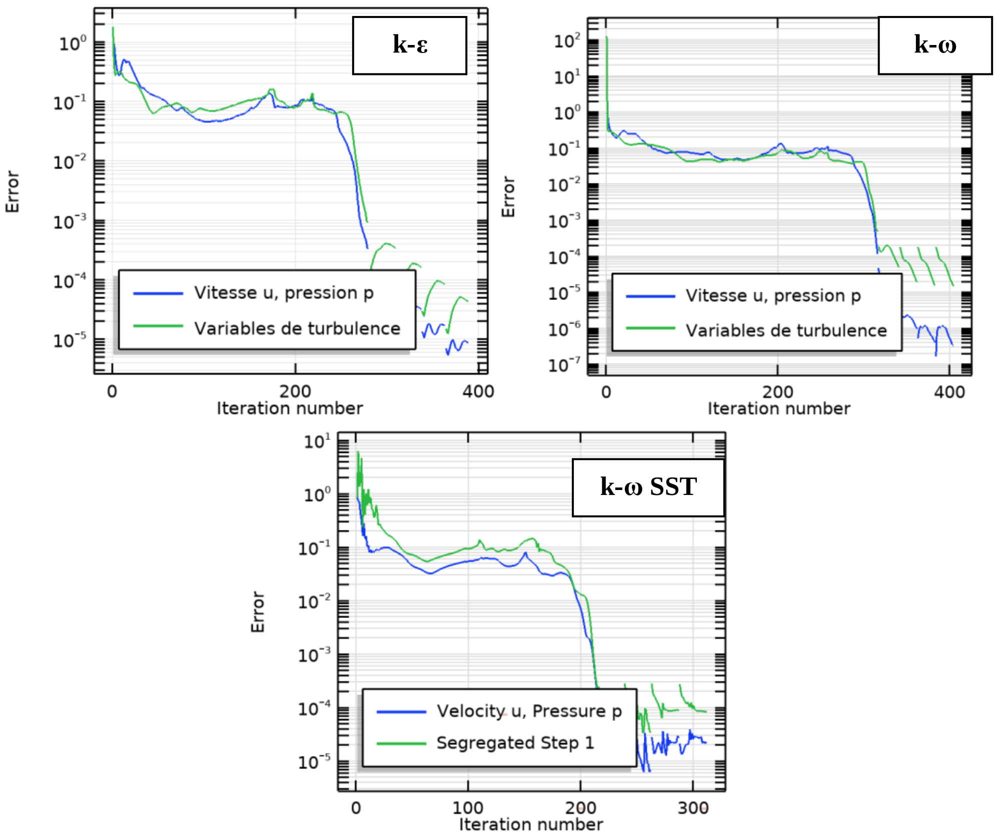

Figure 8. Convergence evolution of the three studied turbulence models.

Velocity Distribution

Water velocity evolution as a function of inlet pressure is shown in figure 9. The effect of increasing inlet pressure on the velocity is not visibly clear due to the large difference between the speed distributions of the three models; also because of the small increase in pressure.

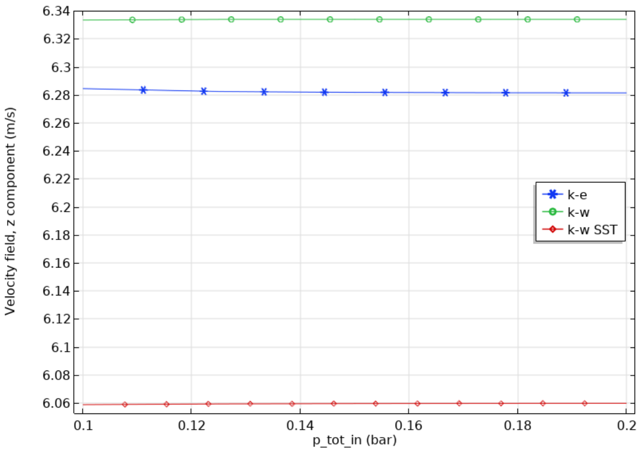

Figure 9. Maximum water velocity evolution as a function of inlet pressure

It can be seen from figure 10 that the three studied models have a certain similarity. However, on the one hand, the maximum velocity was generated by the k-ω sst model, which exceeded 9.5 m/s. On the other hand, this same model obtained the lowest outlet velocity 9. Since, according to Bernoulli's principle, the volute converts kinetic energy into pressure by reducing velocity while increasing pressure. For this reason, the maximum average water velocity was generated by the k-ω SST model. As discussed above by Figure 9, the k-ω model provided the highest discharge speed compared to the other two models. This behavior explains the increase of the Reynolds number at the discharge section for this turbulence model (Figure 11).

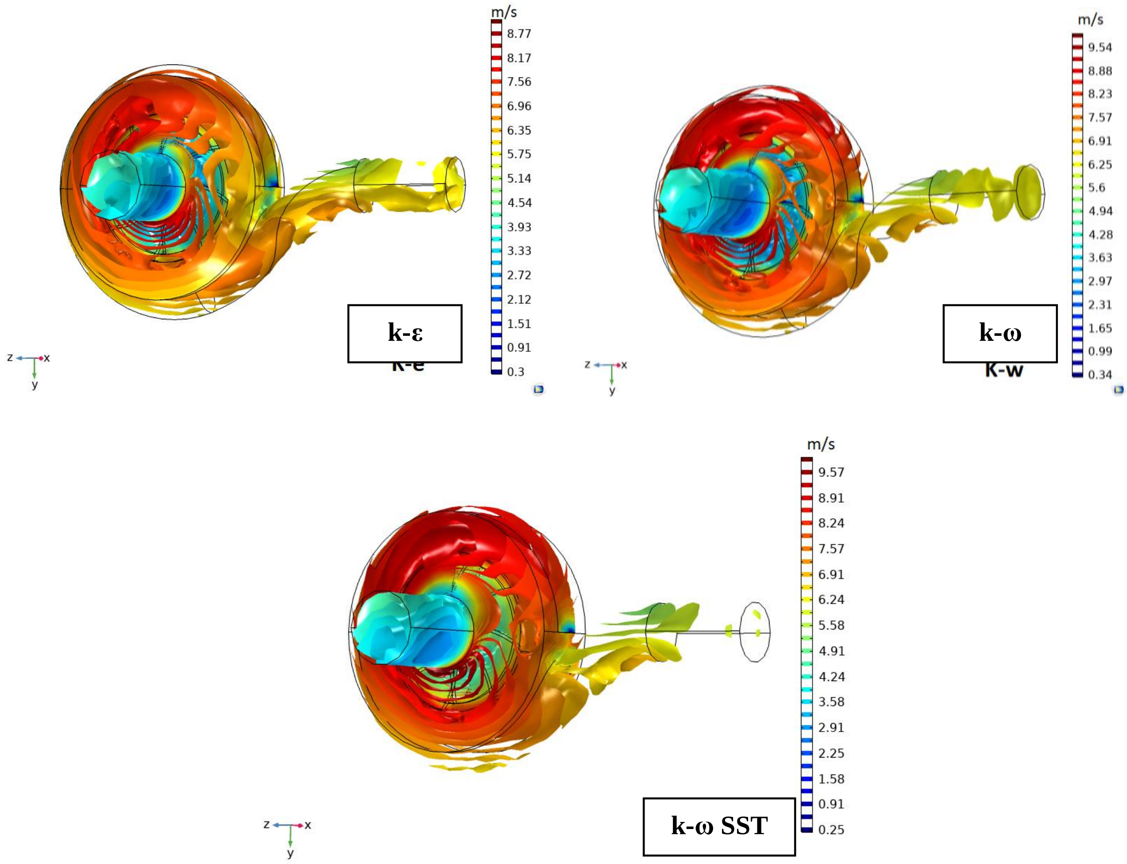

Figure 10. Average water velocity distribution in the volute for the three studied models

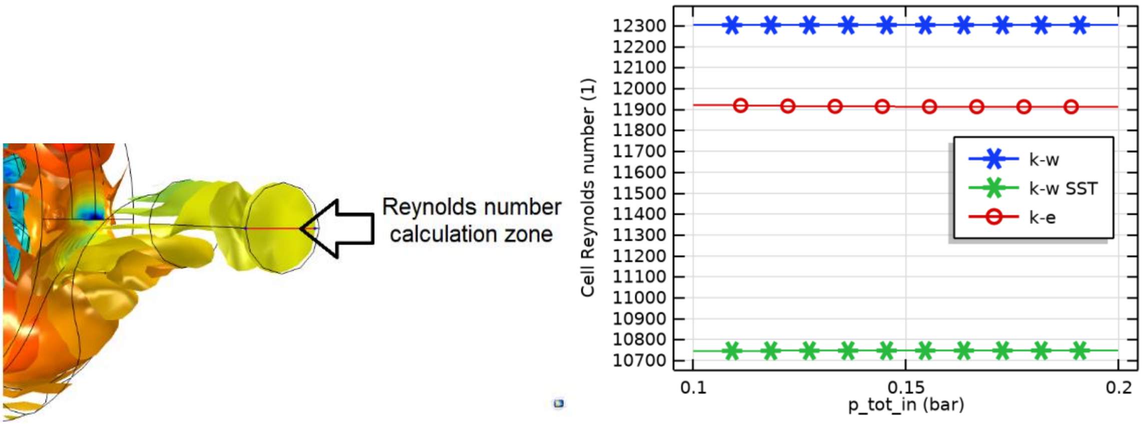

Figure 11. Reynolds number evolution in the discharge section for the three studied models

Pressure Distribution

Figure 12 shows a close-up of the water pressure profiles in the rotational zone, for the three considered models. The highest-pressure contour was obtained by the k−ω SST model. Indeed, a maximum pressure of 1.8 bar was reached with an average pressure of 1.16 bar. Which makes perfect sense, since this model provided the highest water velocity distribution in the volute. As a matter of fact, this speed has been converted into dynamic pressure. Consequently, the highest outlet pressure was recorded for this model (figure 13). Regarding the influence of increasing inlet pressure on the outlet pressure, as mentioned earlier, it is not visibly clear due to the large difference between the pressure distributions of the three models.

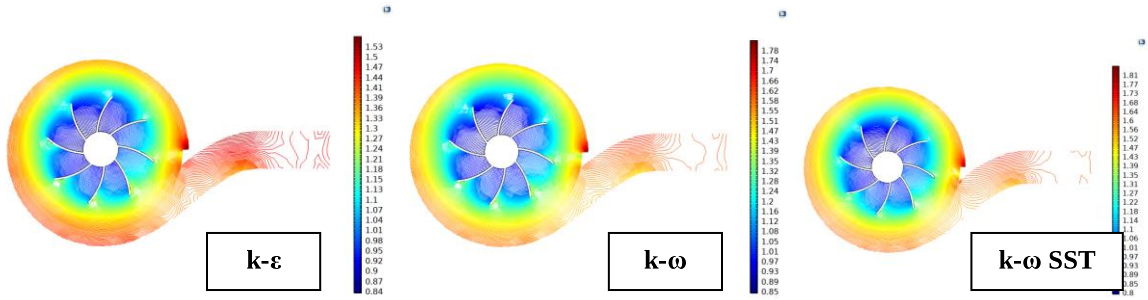

Figure 12. Water pressure [bar] contours in the volute for the three considered models

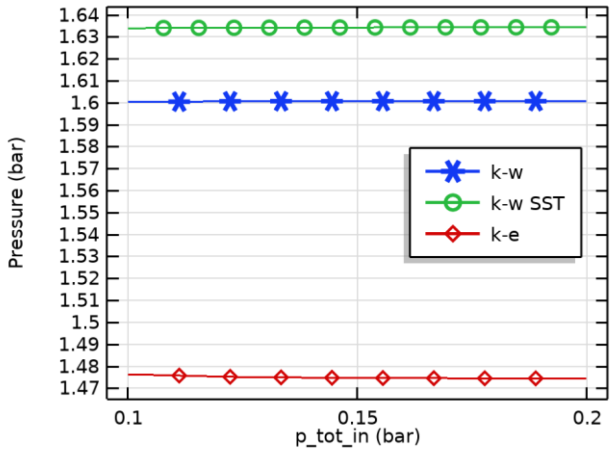

Figure 13. Maximum outlet pressure as a function of inlet pressure

Figure 14 show the static pressure profiles in the impeller domain. Obviously, the highest static pressure was generated by the k-w SST model (1.36 bar). Since he provided the maximum water pressure and velocity in the volute (Figure 12-10).

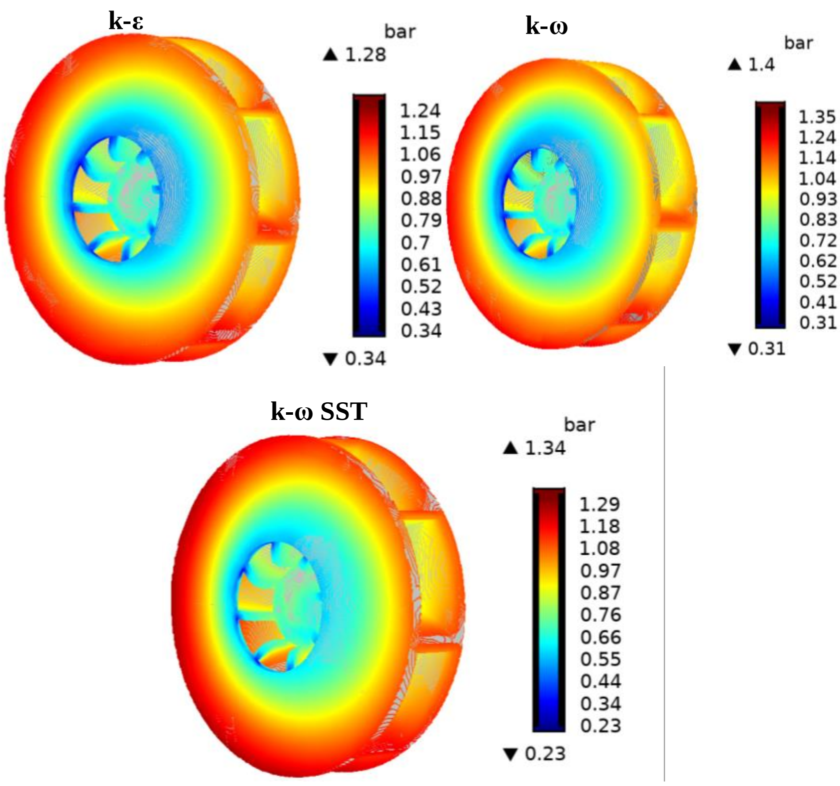

Figure 14. Static pressure contour in the impeller domain

Pump Performance

Figure 15 shows the pump performance curve. The total pressure at the inlet is expressed in terms of the pressure head, H, which is equal to (Comsol-multiphysics, 2020) :

$$H = \frac{\Delta p_{tot}}{\rho . g} \tag{13}$$

The highest head (10.85 m) was achieved by the k-w SST model. Considering that, he generated the greatest outlet pressure, As a result, the better water height was recorded

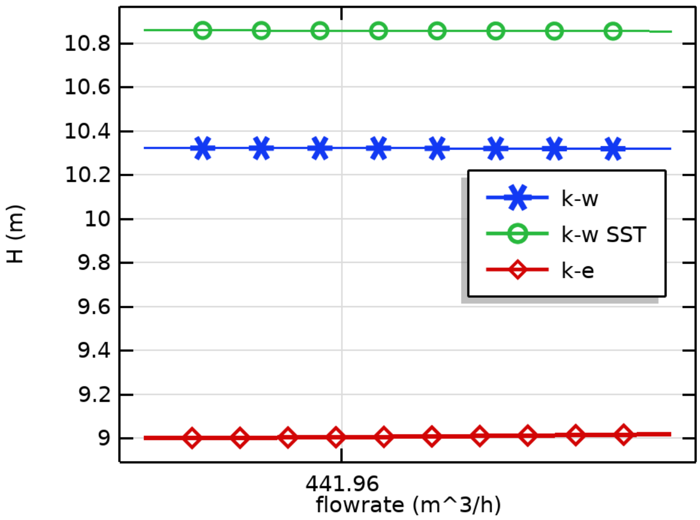

Figure 15. Pump performance curve for the three studied cases

Conclusion

The computational efficiency and performance parameters of three turbulence models were compared. First of all, their influence on the simulation convergence was highlighted. According to the results comparison, the calculation under k-ω SST was able to reach convergence after only 325 iterations. However, the other two models required more than 350 iterations. indeed, it turns out that the calculation results under k-ω SST generated the most optimal values. On the one hand, a low computational cost was recorded and on the other hand a better outlet pressure as well as a better discharge head were achieved. However, the k-ω and the k-ε model provided more precision at the expense of computational cost and the exploited memory.

Scientific Ethics Declaration

The authors declare that the scientific ethical and legal responsibility of this article published in EPSTEM journal belongs to the authors.

Acknowledgements or Notes

* This article was presented as an oral presentation at the International Conference on Technology, Engineering and Science (www.icontes.net) held in Antalya/Turkey on November 16-19, 2022.

References

Bonnet,A., & Luneu, J. (1989). *Aerodynamics theories of fluid dynamics*. Tolouse, France :Cépadués Edition.

Brozoski, A. (2018, October 10). *Solidedge centrifugal pump 01 - 09.* Retrived from https://www.youtube.com/watch?

El Behery, S. M., & Hamed, M. H.(2011). A comparative study of turbulence models performance for separating flow in a planar asymmetric diffuser. *Computers & Fluids , 44*, 248–257.

Liu, H. L., Liu, M. M., Dong, L., Ren, Y., & Du, H. (2012, November). Effects of computational grids and turbulence models on numerical simulation of centrifugal pump with CFD. In *IOP Conference Series: Earth and Environmental Science (Vol. 15*, No. 6, p. 062005). IOP Publishing.

Markus, H., Nicolas, C., Andreas, P., Sherwin, F., & Romuald, S. (2020). Assessment of scale-adaptive turbulence models for volute-type centrifugal pumps at part load operation. *International Journal of Heat and Fluid Flow, 85*, 1.

Menter, F. (1994). Two-equation Eddy-Viscosity turbulence models for engineering applications. *AIAA Journal, 32*(8), 1598-1605.

Menter, F.R.(1992). Performance of popular turbulence models for attached and separated adverse pressure gradient flows. *AIAA JOURNAL, 30* (8), 2072.

Menter, F.R., Kuntz,M., & Langtry, R. (2003). Ten years of industrial experience with the SST turbulence model. *Turbulence Heat and Mass Transfer, 4*. 625-632

Mikhail Pavlovich, B., & Pavel Victorovich, B. (2013). Comparison of turbulence models in the calculation of supersonic separated flows. *World Applied Sciences Journal , 27*(10), 1265.

Paul A, M., Dan, M., & Michael, G. (n.d.). *Fundamentals of Pumps.*Retrived from ez-pdh.com.

Selim, S. M., Hosien M. A., El-Behery, S. M., & Elsherbiny, M. (2016). Numerical analysis of turbulent flow in centrifigual pump. *Proceedings of the 17th Int. AMME Conference.* Cairo, Egypt.

Srivastava, S. (2020, Jun 14). *Centrifugal pump design and analysis using solidworks* . Retrived from skill-lync.com.

Wilcox, D. C. (1991). A half century historical review of the k-w model. *29th Aerospace Sciences Meeting*.

Wilcox, D. C. (1998). *Turbulence modeling for CFD* (2 nd ed.). La Canada, CA *:* DCW Industries.

Author Information

Abdessalam Otmani
Badji Mokhtar University
Annaba, Algeria
Contact e mail:*otmani@esti-annaba.dz*

Slimen Niou
Badji Mokhtar University
Annaba, Algeria

Azzeddine Dekhane
Badji Mokhtar University
Annaba, Algeria

Sahrane Sara
Badji Mokhtar University
Annaba, Algeria

Salahaddine Azzouz
Badji Mokhtar University
Annaba, Algeria

To cite this article:

Otmani, A., Niou, S., Dekhane, A., Sara, S., & Azzouz, S. (2022). CFD investigation on three turbulence models for centrifugal pump application. *The Eurasia Proceedings of Science, Technology, Engineering & Mathematics (EPSTEM), 21,* 202-213.