CFD analysis of turbulent flows transported by centrifugal pumps under low Re numbers

Análisis CFD de flujos turbulentos desplazados por bombas centrífugas con bajo número de Reynolds

https://doi.org/10.17981/ladee.01.01.2020.1

Fecha de Recibido: 27/06/2020 Fecha de Aceptado: 18/08/2020

José Sánchez De la Hoz
Universidad del Atlántico. Barranquilla (Colombia)
jmiguelsanchez@mail.uniatlantico.edu.co

Brando Hernández
Universidad del Atlántico. Barranquilla (Colombia)
bhernandezc@mail.uniatlantico.edu.co

Jorge Duarte Forero
Universidad del Atlántico. Barranquilla (Colombia)
jorgeduarte@mail.uniatlantico.edu.co

Para citar este artículo:
J. Sánchez-De la Hoz, B. Hernández y J. Duarte-Forero, "CFD analysis of turbulent flows transported by centrifugal pumps under low Re numbers", *LADEE*, vol. 1, no. 1, pp. 1–9, 2020. https://doi.org/10.17981/ladee.01.01.2020.1

Abstract— Numerical methodologies have presented an inexpensive solution of laminar and turbulent flows capable of predicting a wide range of mechanical devices in science and engineering. Computational tools have been employed in recent years to analyze the conservation equations behavior used to describe the interaction between turbulent and laminar flows used to transfer the energy needed to perform a complex mechanical system. Due to the above, this paper purpose the application of the numerical method linked to mathematical algorithms capable of generating an approximated solution of the partial differential equations system which determines pressure, and velocity values related to the centrifugal pump performance under low Re conditions in a virtual environment through OpenFOAM software, and Salome 8.3.0. An independence mesh analysis was computed to study the computational effort required to establish an approximated turbulence phenomena description performed by the centrifugal pump into the virtual environment supported by MRFSimpleFoam solver.

Keywords—CFD; turbulence models; mesh independence study; OpenFOAM; centrifugal pump

Resumen— Las metodologías numéricas han presentado una solución económica de flujos laminares y turbulentos capaz de predecir una amplia gama de dispositivos mecánicos en la ciencia y la ingeniería. En los últimos años se han empleado herramientas computacionales para analizar el comportamiento de las ecuaciones de conservación utilizadas para describir la interacción entre los flujos turbulentos y laminares utilizados para transferir la energía necesaria para operar un sistema mecánico complejo. Debido a lo anterior, este trabajo propone la aplicación del método numérico vinculado a algoritmos matemáticos capaces de generar una solución aproximada del sistema de ecuaciones diferenciales parciales que determina los valores de presión y velocidad relacionados con el rendimiento de la bomba centrífuga en condiciones de baja Re a través del software OpenFOAM y Salome 8.3.0. Se calculó un análisis de independencia de malla para estudiar el esfuerzo computacional requerido y así establecer una descripción aproximada de los fenómenos de turbulencia producidos por la bomba centrífuga en el entorno virtual que soporta el solver MRFSimpleFoam.

Palabras clave— CFD; modelos de turbulencia; estudio de independencia de malla; OpenFOAM; bomba centrífuga

I. Introduction

Computational Fluid Dynamics (CFD) has developed in the last years a wide range of solutions related to the numerical analysis of laminar, and turbulent flows in order to promote quick, and inexpensive methodologies applied to visualize, analyze, and predict the physical behavior of hydraulic pumps under determined conditions. Centrifugal pumps have defined as the main devices used to transport feedwater in nuclear power plants. CFD techniques were applied by Carpuso et al. [1] to generate a numerical design, and optimize the novel impeller based on the solution of the complex differential equation system used to model a turbulent flow [2]-[4]. In this sense, a multi-objective optimization was proposed based on the numerical validation of the simulated data against the experimental model generated by means of the experimental measurements [5]-[8].

These numerical studies may be applied to analyze the noise induced by the number of blades that perform the rotor-stator zone in a centrifugal pump. Yu-quin and Ze-wen [9] developed an unsteady simulation in order to analyze the turbulent flow into the centrifugal pump by means of the simple variable principle. A set of different blades was proposed to simulate the transport phenomena under different conditions. This numerical simulation was validated through the noise test platform to compare the level of noise produced under different working conditions. Zhang, Jiang, Gao & Liu [10] proposed a CFD analysis of unsteady flow structures and pressure pulsations under non-working conditions. In this sense, a Delayed Detached Eddy Simulation (DDES) approach was proposed to simulate a centrifugal pump in a virtual environment. These numerical results have shown an unsteady evolution of the stall structure, and pressure spectrum at a stall frequency of $0.25f_{o.}$

On the other hand, Yousefi, Noorollahi, Tahani, Fahimi & Saremian [11] developed a numerical simulation of viscous flows for various industries in order to improve oil pump performance. In this way, a three-dimensional model was proposed to analyze a turbulent two-phase flow confirmed by water and oil. The main effects of geometrical parameters were computed through the Finite Volume Method (FVM) to discretize and solve the set of the partial differential equations which govern the turbulent flow close to the complex zones of the centrifugal pump. In this sense, Wang, Luo, Li, Xia & Liu [12] improved the marine centrifugal pump performance based on the maximum weighted average efficiency and the minimum vibration intensity. The numerical results shown low amplitudes of pressure fluctuation values at different monitoring points of the centrifugal pump numerically studied.

The computing of two-phase flows in a virtual environment may be possible through the differential equations solution, which describes gas and liquid distributions into the centrifugal pump. Ge, He, Huang, Zuo & Luo [13] performed a numerical simulation based on the CFD-PBM method to compute gas and liquid distributions produced into the centrifugal pump. Simulation results were validated with experimental measurements to establish the better Eulerian-Eulerian model approximation to the real behavior of the two-phase flow. Numerical results have shown an increase of bubbles with a large diameter related to the increase of the inlet gas volume fraction. Deng, Li, Guan, Chen & Liu [14] proposed a numerical study on cavitation phenomena in a centrifugal pump by means of the Zwart-Gerber-Belamri Model applied in the turbulent flow simulation. The flow viscosity effects and surface tension on bubbles' expansion and compression were considered as a function of pressure and density. The vapor volume fraction was analyzed and predicted under different working conditions to compare diesel and water flows. In this sense, the pressure distribution in Diesel flow was more homogeneous than the pressure distribution in water flow, and these numerical results may be applied to perform the centrifugal pump conditions in a virtual environment.

Considering the above state of the art, the paper's content is described as follows: CFD methodology is described in section II. The numerical results of the RANS k-e turbulence model are presented in section III. The main conclusions are given in section IV.

II. CFD Methodology

A. *Experimental model*

An experimental model was defined in this research to define the main relationship between the pump response until different physical conditions [15]-[18]. The pump features were taking into account the main factor which influences the head pressure and mass flow rate needed to overcome the main restrictions of the system. In this sense, the mass flow rate restriction at the pump discharge and the atmospheric pressure were computed in the computational domain to relate numerical data computed in a virtual environment with the experimental measurements of the flow under real working conditions. Fig. 1 shows the main features of the experimental test bench performed in this research.

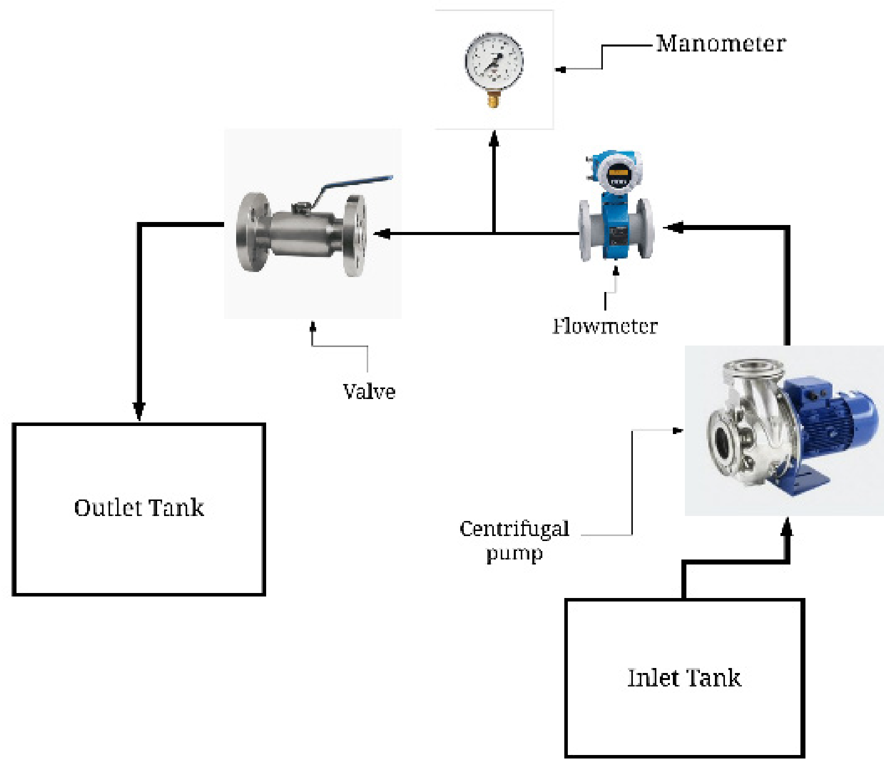

Fig. 1. DWT Decomposition model.
Source: Authors.

The experimental test bench was developed, taking into account the CFD analysis of centrifugal pumps previously studied. Therefore, an experimental model was developed through the mean values of the mass flow pressure and inlet velocity captured with the measuring instruments with a 95 percent probability supported by ISO 2858:1975. Pressure values of the flow into the test bench were measured with CPG500 digital manometer with a range of -1 to 16 bar, and the measurement accuracy of 0.25%.

The pressure and velocity values of the test bench were employed to define the numerical model employed to solve the dependent variables of the partial differential equation system by means of the boundary conditions of the model. Five main points of pressure and flow rate were employed to analyze the mass flow and pressure values generated by the test bench by the centrifugal pump. A good agreement between the numerical and experimental models was defined with a low error rate close to the 2%. The main centrifugal pump features were appreciated in Table I.

TABLE I.
EXPERIMENTAL VALUES OF CENTRIFUGAL PUMP.

Parameter	value	Units
Number of blades	5	-
Blade angle	25	Degree
Flow rate	32	m^3/s
Dynamic head pressure	125	m.c.a
Suction head pressure	5	m.c.a
Discharge diameter pump	6	In
Suction diameter pump	6	in
Centrifugal pump power	1.20	kW

Source: Authors.

B. *RANS k-e turbulence model*

Numerical methods are generally applied to analyze the laminar or turbulent flow generated by centrifugal pumps. In this sense, a numerical analysis may be thoroughly developed in complex zones of the industrial centrifugal pump in order to analyze the main features used to perform a centrifugal pump in a virtual environment. Computational Fluid Dynamics offers advanced techniques applied in the solution of transported flows by means of the solution of turbulence models used to complement an approximated solution of the Navier – Stokes equations. A set of partial differential equations may be defined and studied in a CFD code capable of approximating the mass, conservation, and energy equations to the flow behavior into the centrifugal pump. These equations are described as follows (1)(2)(3).

$$\frac{\partial\rho}{\partial t} + \nabla(\rho\vec{V}) = 0 \tag{1}$$

$$\frac{\partial(\rho\vec{V})}{\partial t} + \nabla\cdot\vec{V}(\rho\vec{V}) = -\nabla P + \nabla\cdot\prod \tag{2}$$

$$\frac{\partial(\rho\vec{E})}{\partial t} + \nabla\cdot\left(\rho\vec{E}\vec{V}\right) + \nabla\cdot\left(\rho\vec{V}\right) + \nabla\cdot\left(\prod\vec{V}\right) = -\nabla\cdot Q \tag{3}$$

RANS k-e turbulence models are characterized by the steady and unsteady solution of the differential equation system supported by the main turbulence variables, like the turbulent kinetic energy (k), and dissipation rate (e). Therefore, the main parameters used to define the hydraulic pump behavior may be described in the function of the turbulence phenomena described in (4) and (5).

$$\frac{\partial k}{\partial t} + \nabla\cdot\left(\vec{U}k\right) - \nabla\cdot(\nu+\nu_T)\nabla k = P_k - {}^{\varepsilon}\!/_{k} \tag{4}$$

$$\frac{\partial k}{\partial t} + \nabla\cdot\left(\vec{U}\varepsilon\right) - \nabla\cdot(\nu+\nu_T)\nabla\varepsilon = P_k - {}^{\varepsilon}\!/_{k} \tag{5}$$

Turbulent viscosity effects may be model into the virtual environment by means of the approximated solution of (6). In this way, the numerical model may capture the main effects of the turbulent flow, which influences the head pressure simulated in the virtual environment.

$$\nu_T = \frac{C_\mu\sqrt{k}}{\varepsilon},\quad C_\mu = 0.009 \tag{6}$$

This numerical simulation was improved through the numerical computing of two equations capable of modeling the velocity gradients of the turbulent flow into the virtual environment. Therefore, the rotor-stator interaction zone was influenced by (7) and (8), as follows:

$$\mathrm{LHS} = \left[c_1 - \frac{\eta\left(1-\frac{\eta}{\eta_0}\right)}{1+\beta\eta^3}\right]\cdot(P_k - c_2)\frac{\varepsilon}{k} \tag{7}$$

$$\eta = \frac{(S_{ij}S_{ij})^{\frac{1}{2}}k}{\varepsilon} \tag{8}$$

Constant values of this numerical model were computed by means of the Re-Normalization Group techniques with the statistical approximation of the constant values in this numerical model. Even though the above approach improves the accuracy of this numerical simulation, a mesh independency study was developed to improve the set of the limited nodes, and cell employed to model the computational domain into the virtual environment and discretize the set of the partial differential equations during the convergence of the numerical data.

III. Numerical results

The numerical prediction of the main results involves the solution of the partial differential equations computed in a CFD code with OpenFOAM software. This numerical approach was supported by means of powerful techniques used in the analysis of mechanical components coupled to energy transfer systems. The computational resource implemented was a PC with an Intel i7 P4-2.9 GHz and 16 GB of RAM memory. The solution of the partial differential equations influenced the inexpensive and quick description of the main effects which define the centrifugal pump performance under low Re conditions taking into account the turbulent transport phenomena defined in function of the turbulent kinetic energy, and dissipation rate parameters which may be used to describe the functionality of the centrifugal pump in a virtual environment. In this sense, the limited set of cells and nodes computed to simulate the centrifugal pump was performed through a mesh independence analysis in order to define the optimal computational resources needed to generate a high approximation of the turbulent flow into the centrifugal pump with a low Re simulated under different physical conditions. The mesh determined with computational tools is shown in Fig. 2, and the main parameters which perform the mesh (skew, orthogonal quality, and aspect ratio) are presented in Table II. A good agreement between the numerical data was defined with a good convergence criterion for 1.52×10^6 nodes to discretize the control volume of this numerical model (these results are present in Fig. 3).

Table II.
Mesh quality of the numerical model.

Parameter	value	Quality
Skew	0.813	Acceptable
Orthogonal quality	1.445	Acceptable
Aspect Ratio	5.264	Acceptable

Source: Authors.

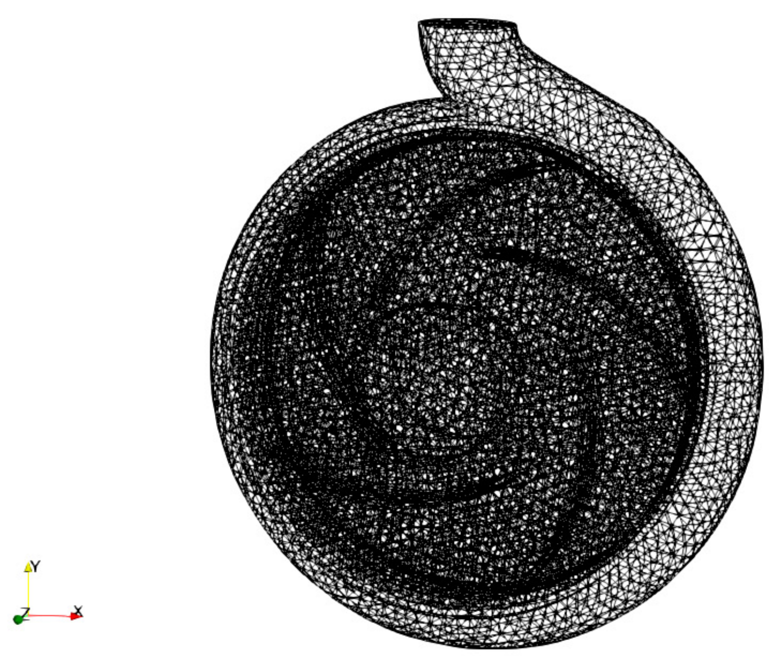

Fig. 2. Mesh density of computational domain.
Source: Authors.

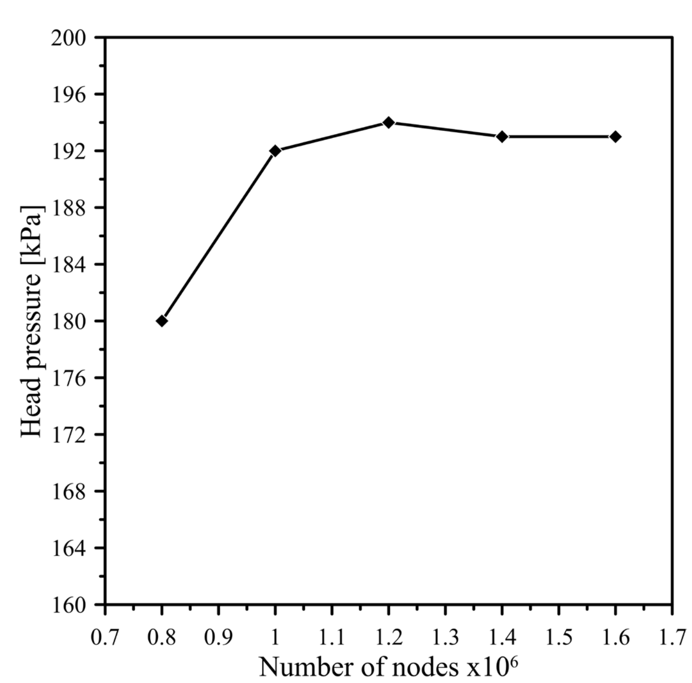

Fig. 3. Mesh independence study.
Source: Authors.

In this sense, the numerical data obtained in the function of the boundary conditions and the numerical solution of the partial differential equations were compared with the experimental measurements of the test bench described in Fig. 1. A good agreement was reached between the experimental model and numerical data (this approach is presented in Fig. 4).

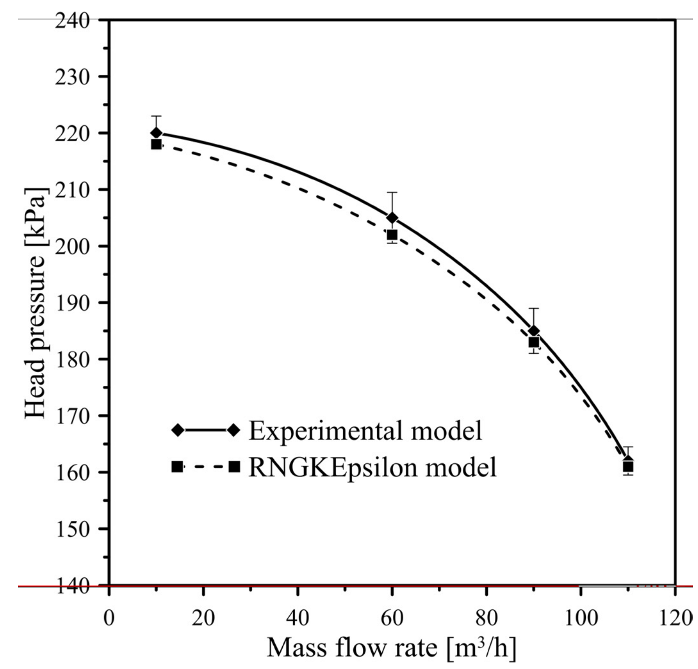

Fig. 4. Mesh density of computational domain.
Source: Authors.

On the other hand, the approximated solution of the partial differential equations with the main parameters of the mesh previously defined was employed to establish the velocity, pressure, and turbulent energy contours of the turbulent flow transported into the centrifugal pump in a virtual environment (as shown in Fig. 5, Fig. 6, and Fig. 7).

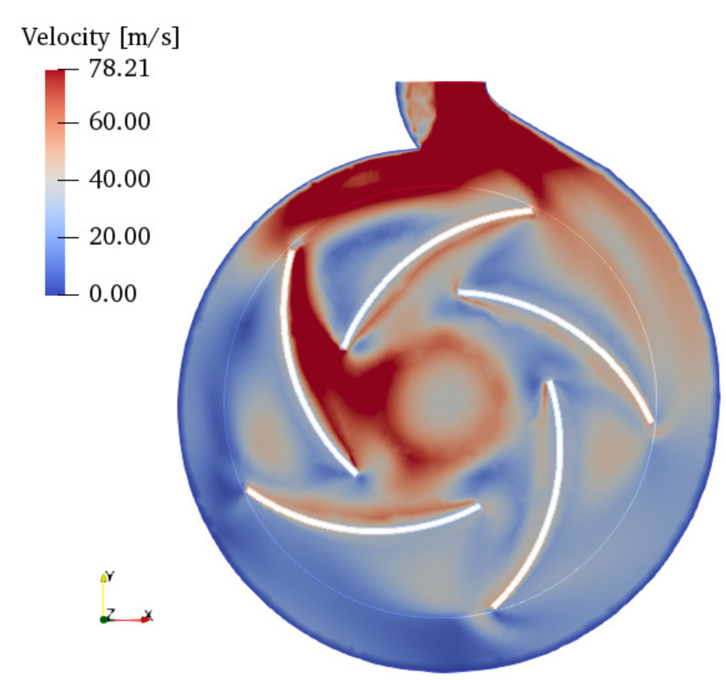

Fig. 5. Flow velocity simulated with RNGKEpsilon model.
Source: Authors.

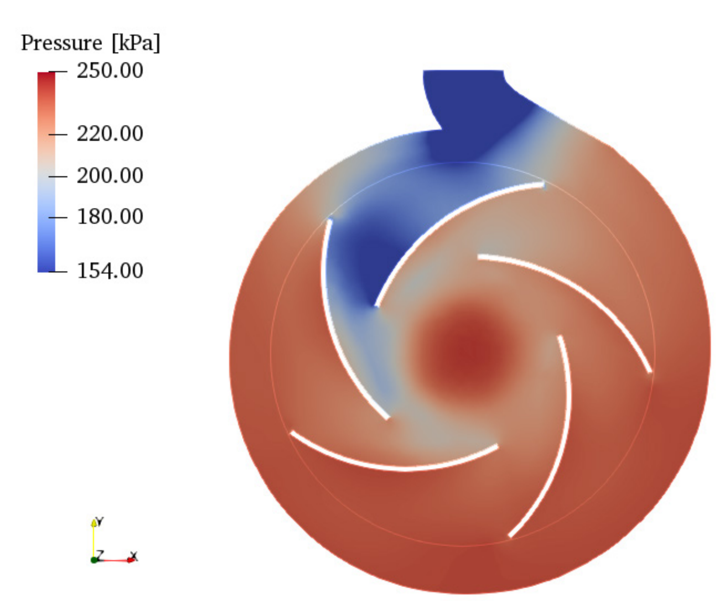

Fig. 6. Flow pressure simulated with RNGKEpsilon model.
Source: Authors.

This approach shows the energy transferred in the form of pressure in the mass flow transported to the hydraulic system in order to overcome the pressure restrictions of the system (Fig. 7).

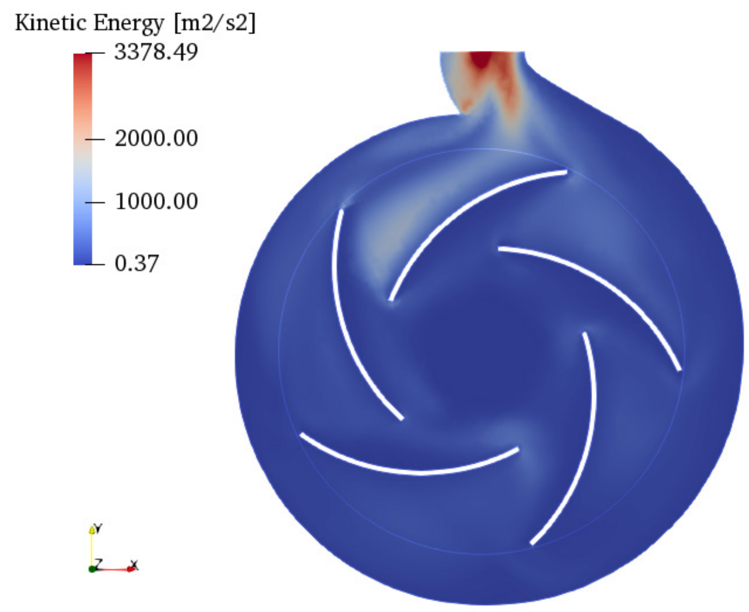

Fig. 7. Turbulent kinetic energy described with OpenFOAM.
Source: Authors.

A numerical description of the turbulence phenomena was computed with OpenFOAM software considering centrifugal forces, and the blades interaction with the turbulent flow displaced into the simulated centrifugal pump. Therefore, statistical values of the turbulent flow at the centrifugal pump outlet may be analyzed in function of the mass flow values, and the impeller angular velocity (as shown in the Fig. 8). This way shows the turbulent kinetic energy described with the energy transferred in form of pressure and their relation with the impeller angular velocity of the centrifugal pump, a higher value of 3800m²/s² was computed in this numerical study using a turbulence model with OpenFOAM software.

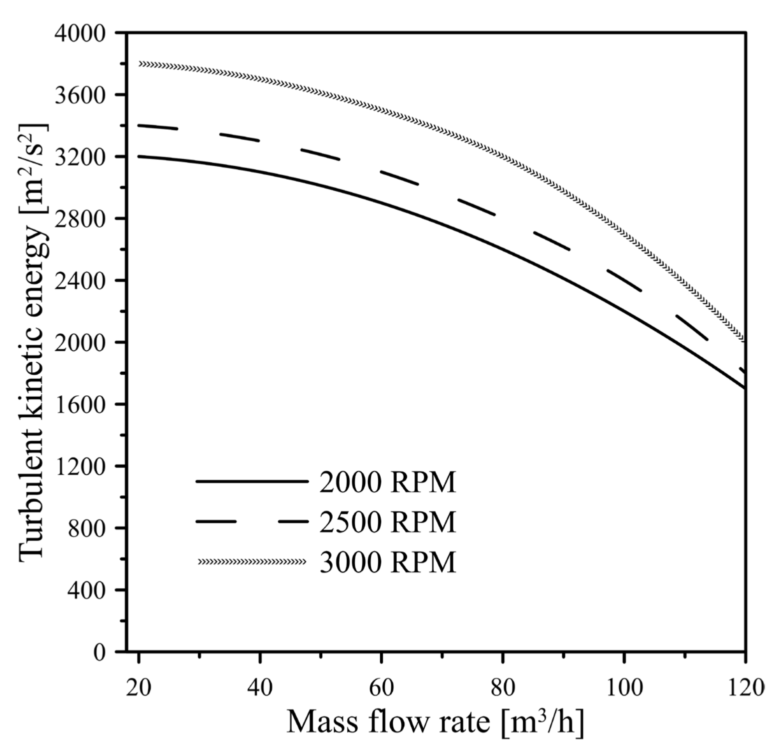

Fig. 8. Turbulent kinetic energy of centrifugal pump.
Source: Authors.

IV. Conclusions

A numerical model was developed in this research to simulate the physical values which describe the centrifugal pump performance in a virtual environment. Intel i7 P4-2.9 GHz and 16 GB of RAM memory were computational resources needed to simulate the centrifugal pump performance in a virtual environment. An independence mesh study was developed to minimize computational efforts, 1.5×10^6 nodes were defined to solve the set of the partial differential equations in a virtual environment considering the quality of the mesh developed with Salome 8.3.0 software in the function of 0.813 skews, 1.445 orthogonal quality, and 5.264 aspect ratio.

In this way, a low error rate was reached between the experimental and numerical data. These results have shown the high level of the approximation reached with powerful computational tools used to solve a numerical model with OpenFOAM software. For 2000 rpm, the maximum values of pressure and velocity were 154kPa, and 78 m/s, respectively. These numerical values were associated with the kinetic energy generation with a maximum value of 3.3 × 106 m^2/s^2.

This numerical analysis has shown a high response in the solution of the partial differential equations employed to simulate a turbulent flow in a virtual environment and relate this solution with the complex centrifugal pump geometry by means of the Finite Volume Method (FVM). For numerical purposes, 0.009 turbulent constant value maintained a good relation between the kinetic energy, dissipation rate, and physical values determined in the numerical model. A good convergence behavior was reached with 1×10^{-5} residuals for the CFD code solution.

Acknowledgment

The authors would like to acknowledge the Universidad del Atlántico, Universidad Francisco de Paula Santander, and Sphere Energy company for their support in the development of this research.

References

[1] T. Capurso, L. Bergamini & M. Torresi, "Design and CFD performance analysis of a novel impeller for double suction centrifugal pumps," *Nucl. Eng. Des.*, vol. 341, no. 1, pp. 155–166, Jan. 2019, https://doi.org/10.1016/j.nucengdes.2018.11.002

[2] S. Emani, M. Ramasamy & K. Z. K. Shaari, "Discrete phase-CFD simulations of asphaltenes particles deposition from crude oil in shell and tube heat exchangers," *Appl. Therm. Eng.*, vol. 149, pp. 105–118, Feb. 2019, https://doi.org/10.1016/J.APPLTHERMALENG.2018.12.008

[3] M. Cagnoli, A. de la Calle, J. Pye, L. Savoldi & R. Zanino, "A CFD-supported dynamic system-level model of a sodium-cooled billboard-type receiver for central tower CSP applications," *Sol. Energy*, vol. 177, pp. 576–594, Jan. 2019. https://doi.org/10.1016/J.SOLENER.2018.11.031

[4] X. Wang, M. Wang, J. Zhang & Y. Wu, "Study on the water seal formation process in advanced PWR pressurizer using CFD method," *Ann. Nucl. Energy*, vol. 135, p. 106949, Jan. 2020. https://doi.org/10.1016/J.ANUCENE.2019.106949

[5] O. M. Ilori, A. J. Jaworski & X. Mao, "Experimental and numerical investigations of thermal characteristics of heat exchangers in oscillatory flow," *Appl. Therm. Eng.*, vol. 144, pp. 910–925, Nov. 2018. https://doi.org/10.1016/J.APPLTHERMALENG.2018.07.073

[6] W. Qu, J. Xiong, S. Chen & X. Cheng, "High-fidelity PIV measurement of cross flow in 5 × 5 rod bundle with mixing vane grids," *Nucl. Eng. Des.*, vol. 344, pp. 131–143, Abr. 2019. https://doi.org/10.1016/j.nucengdes.2019.01.021

[7] T. Ziegenhein, D. Lucas, G. Besagni & F. Inzoli, "Experimental study of the liquid velocity and turbulence in a large-scale air-water counter-current bubble column," *Exp. Therm. Fluid Sci.*, vol. 111, p. 109955, Feb. 2020. https://doi.org/10.1016/j.expthermflusci.2019.109955

[8] W. Lyu & O. el Moctar, "Numerical and experimental investigations of wave-induced second order hydrodynamic loads," *Ocean Eng.*, vol. 131, pp. 197–212, Feb. 2017. https://doi.org/10.1016/j.oceaneng.2016.11.047

[9] W. Yu-qin & D. Ze-wen, "Influence of blade number on flow-induced noise of centrifugal pump based on CFD/CA," *Vacuum*, vol. 172, p. 109058, Feb. 2020. https://doi.org/10.1016/j.vacuum.2019.109058

[10] N. Zhang, J. Jiang, B. Gao & X. Liu, "DDES analysis of unsteady flow evolution and pressure pulsation at off-design condition of a centrifugal pump," *Renew. Energy*, vol. 153, pp. 193–204, Jun. 2020. https://doi.org/10.1016/j.renene.2020.02.015

[11] H. Yousefi, Y. Noorollahi, M. Tahani, R. Fahimi & S. Saremian, "Numerical simulation for obtaining optimal impeller's blade parameters of a centrifugal pump for high-viscosity fluid pumping," *Sustain. Energy Technol. Assessments*, vol. 34, pp. 16–26, Aug. 2019. https://doi.org/10.1016/j.seta.2019.04.011

[12] K. Wang, G. Luo, Y. Li, R. Xia & H. Liu, "Multi-condition optimization and experimental verification of impeller for a marine centrifugal pump," *Int. J. Nav. Archit. Ocean Eng.*, vol. 12, pp. 71–84, Aug. 2019. https://doi.org/10.1016/j.ijnaoe.2019.07.002

[13] Z. Ge, D. He, R. Huang, J. Zuo & X. Luo, "Application of CFD-PBM coupling model for analysis of gas-liquid distribution characteristics in centrifugal pump," *J. Pet. Sci. Eng.*, vol. 194, p. 107518, Nov. 2020. https://doi.org/10.1016/j.petrol.2020.107518

[14] S.-S. Deng, G.-D. Li, J.-F. Guan, X.-C. Chen & L.-X. Liu, "Numerical study of cavitation in centrifugal pump conveying different liquid materials," *Results Phys.*, vol. 12, pp. 1834–1839, Mar. 2019. https://doi.org/10.1016/j.rinp.2019.02.009

[15] E. Terzi, A. Cataldo, P. Lorusso & R. Scattolini, "Modelling and predictive control of a recirculating cooling water system for an industrial plant," *J. Process Control*, vol. 68, pp. 205–217, Aug. 2018. https://doi.org/10.1016/J.JPROCONT.2018.04.009

[16] T. C. Roumpedakis, S. Chapaloglou, P. Pallis, A-D. Leontaritis, K. Braimakis, S. Karellas & P. Vourliotisa "Experimental Investigation and CFD Analysis of Heat Transfer in Single Phase Subcooler of a Small Scale Waste Heat Recovery ORC," *Energy Procedia*, vol. 129, pp. 487–494, Sep. 2017. https://doi.org/10.1016/j.egypro.2017.09.166

[17] H. Quan, Y. Guo, R. Li, Q. Su & Y. Chai, "Optimization design and experimental study of vortex pump based on orthogonal test," *Sci. Prog.*, vol. 103, no. 1, pp. 1–20, Oct. 2019. https://doi.org/10.1177/0036850419881883

[18] Y. Gao, X. Fan & R. Dang, "Numerical characterization of the effects of flow rate on pressure and velocity distribution of pump as turbine," *Current science*, vol. 117, no. 1, pp. 57-63, Aug. 2019. https://doi.org/10.18520/cs/v117/i1/57-63

José Sánchez De la Hoz. Formación académica: Pregrado. Nivel de formación: Profesional. Universidad del Atlántico (Barranquilla, Colombia). https://orcid.org/0000-0002-1181-4785

Brando Hernández. Formación académica: Pregrado. Nivel de formación: Profesional. Universidad del Atlántico (Barranquilla, Colombia). https://orcid.org/0000-0002-9702-4150

Jorge Duarte Forero. Formación académica: Universitaria. Nivel de formación: Doctorado. Universidad del Atlántico (Barranquilla, Colombia). https://orcid.org/0000-0001-7345-9590