International Journal of Emerging Engineering Research and Technology
Volume 3, Issue 11, November 2015, PP 23-28
ISSN 2349-4395 (Print) & ISSN 2349-4409 (Online)

An Experience with Simulation Modelling for Radial Flow Pump

Mohd Azlan Ismail[1], Al Khalid Othman[2], Hushairi Zen[3]

[1]*Lecturer, Faculty of Engineering, University Malaysia Sabah, Malaysia*
[2]*Associate Professor, Faculty of Engineering, University Malaysia Sarawak, Malaysia*
[3]*Senior Lecturer, Faculty of Engineering, University Malaysia Sarawak, Malaysia*

ABSTRACT

The aim of this paper is to present the simulation modelling process of an end suction centrifugal pump using commercial Computational Fluid Dynamics software. The study pump is a radial flow pump with specific speed of 1075.0 (mm/mm). Three distinct models, called Volute, Impeller and Draft Tube, were generated based on the geometry and dimension of the pump. Unstructured tetrahedral mesh were applied to all flow domains, utilizing the advantage of good meshing capability for complex surface geometries. The governing equations that were used are steady-state three-dimensional Navier-Stokes equations coupled with the k-epsilon turbulence model. It can be observed that the simulation result yield comparable performance when compared with the manufacturer data sheet. Pressure distribution, velocity vector and velocity streamline visualization show local losses inside the pump. It is believed that the finding from this study complement the ongoing research relevant to turbomachinery modelling and will be useful to investigate hydraulic characteristic inside the pump.

Keywords: centrifugal pump, computational fluid dynamic, performance curve

INTRODUCTION

An end-suction centrifugal pump is a mechanical device used to covert mechanical energy to hydraulic energy. It is one of the most commonly-used pump types in the world. The applications cover many industry sectors such as building services, irrigation, water supplies and industrial plants. The construction consists of four main parts; casing, bearing, shaft and impeller. The impeller is coupled with an electric motor that rotates at its designed rotational speed, centrifuging the liquid in the circumferential direction. The flowing liquid enters from the impeller eye and increases in static pressure along the impeller blades due to the increase of cross-section area. The velocity is further reduced in the volute, thus increasing the static pressure at the outlet of the pump.

Investigation of pump hydraulic performance and characteristics is highly complex and extremely difficult. The interaction between stator and rotor generates a complex flow field known as flow separation, boundary layers, turbulence, recirculation, and vortex dynamics. In addition, a large number of interdependent variables are involved in a pump operation. A traditional technique to design the centrifugal pump is based on theoretical framework, empirical correlation, prototyping, experimental work and engineering experience [1].

Detailed experimental work with accurate sensors and instrumentations will give definite results, but is typically time-consuming and prohibitively expensive. Furthermore, if geometry refinement is performed, it will become extremely difficult to pinpoint the absolute source of the changes in the performance. With the advancement of computation power and robustness of numerical algorithms, computational fluid dynamics (CFD) has the capacity to solve and analyse fluid flows for turbomachinery, both for designing and performance prediction. This allows researchers and engineers to predict and investigate numerous parameters, such as pressure distribution and velocity distribution which is difficult to accomplish by experimental work.

Stickland et. al. (2002) undertook experimental investigation on the flow pattern of particles between impeller blades and compared the result with simulation modelling [2]. High-speed digital video and

Address for correspondence:

lanz_mr@ums.edu.my

particle image velocimetry was used to visualize and examine the flow pattern between the impeller blades. Visualization from CFD simulation results showed reasonably similar flow patterns with the moving particle from the experimental analysis.

Maitelli et. al. (2010) presented simulation analysis of a centrifugal pump with electrical submersible pumping for oil and gas exploration applications [3]. The results from three simulation analysis with different impeller connector configurations were compared with pump manufacturer data sheets. The outcome from the findings shows that all the performance curves have the same performance characteristic when compared with the manufacturer data sheets. Adjustments by adding diffusers were proposed by the author in order to reduce the convergence problem and mitigate the software limitations. Cheah et. al. (2007) and Munish et. al. (2011) presented a simulation analysis on the pressure and velocity distribution at the flow path between impeller blades [4, 5]. They pointed out that flow separation at the leading edge intensifies during off-design operation.

Substantial amounts of simulations studying pumps with different specific speeds have been reported, but the pumps are diverse in geometry and configuration. This paper is a complement to the ongoing simulation analysis on centrifugal pumps. The methods of analysis were adapted from other researcher's approaches. The aim of this study is to observe the simulation accuracy and visually study the hydraulic behaviour inside the pump.

RESEARCH METHODOLOGY

Firstly, the CAD model was generated by using Solid work and Blade Gen software. Then, the CFD simulation analysis was performed with ANSYS CFX software. The best efficient point from the simulation analysis is compared with the manufacturer data sheet to validate the accuracy. This was followed by the presentation of pressure gradient and velocity distribution inside the pump post-processing illustrations. The flow of work for the simulation process is shown in Figure 1.

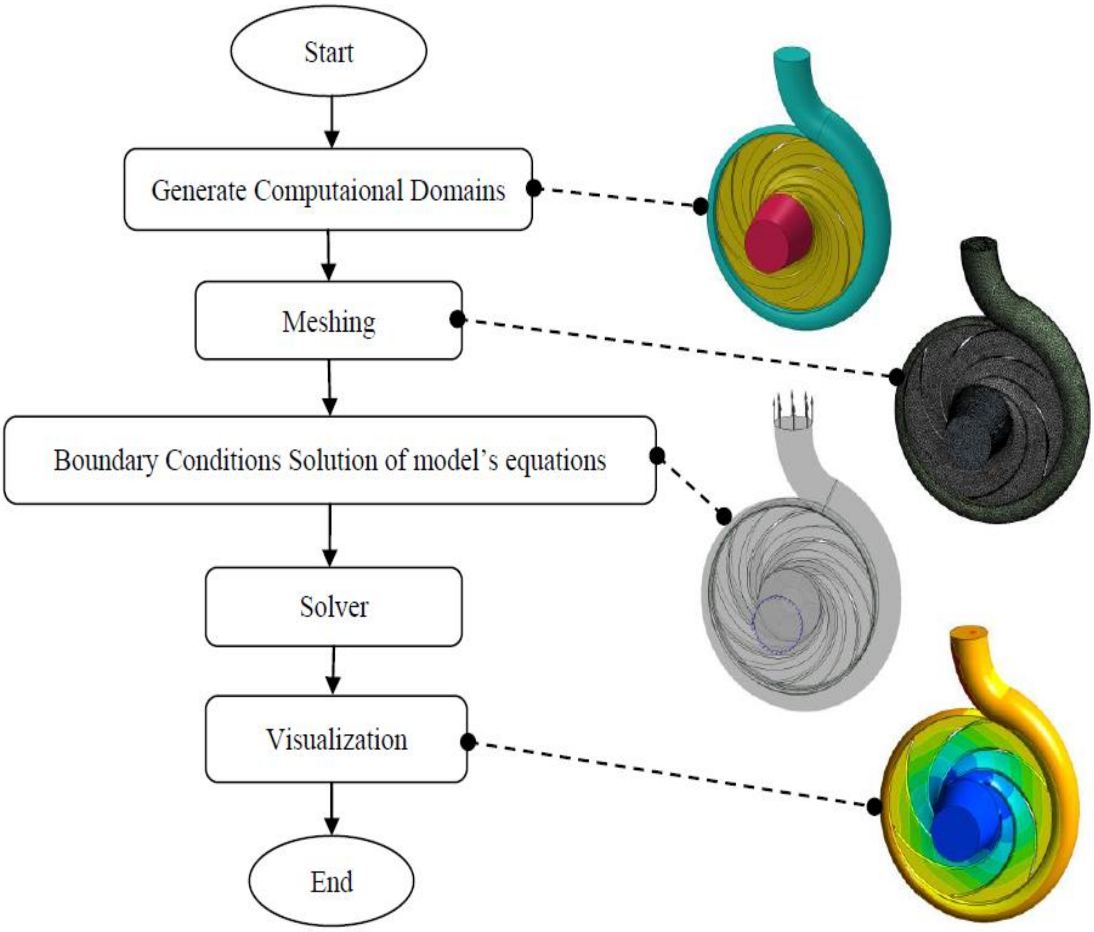


Figure1. *Flow of Work for the Simulation Process*

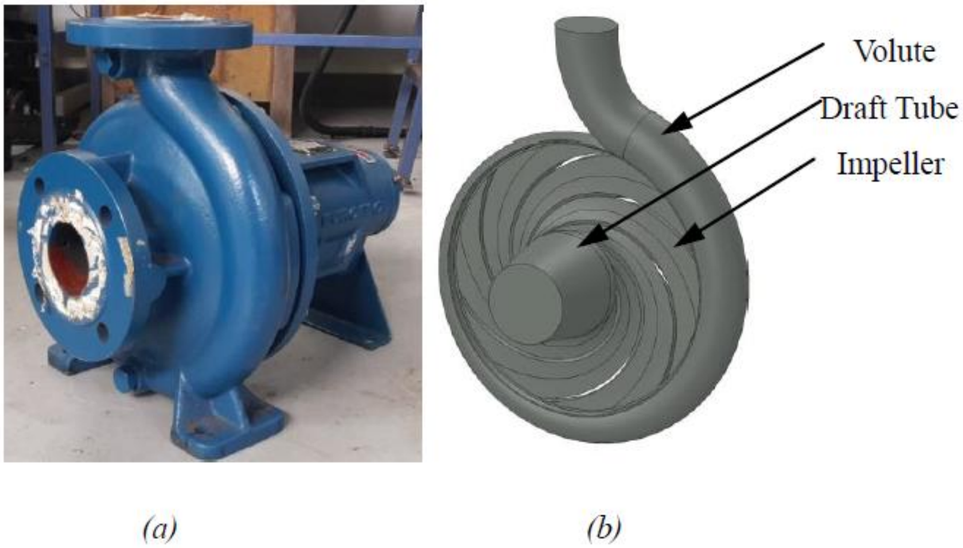


Figure2. *(a) The Study Pump (b) CAD model of Computational Domains*

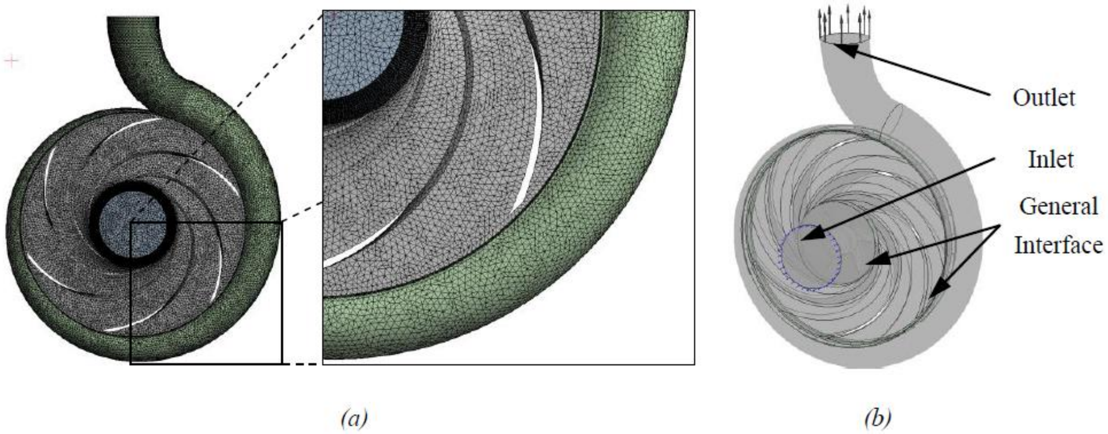


Figure3. *(a) Mesh Topology at the Computational Domain Surfaces (b) Boundary Conditions and Interfaces*

Study Pump

One radial flow pump with a specific speed of 1075.0 (mm/mm) was selected for the simulation analysis, as shown in Figure 2 (a). The EuroFlo pump has the model number EU50-20. The pump has a shrouded type impeller with a diameter of 214.0 mm and 6 blade number. The inlet and outlet of the pump is 65.0 mm and 50.0 mm, respectively. The pump is designed to couple with a four-pole induction motor, and rotates at 1450 RPM with a power rating of 2.2kW. The best efficient point is at 8.0 l/s, producing 14.0 meters of pressure head at the pump outlet. The pump dimension and performance is in accordance with EN733/DIN24255.

Computational Domains

The CAD model based on geometry of the cavity inside the pump, shown in Figure 2 (b). The model represents computational domains for the fluid flow between inlet and outlet of the pump. Three distinct models, called Volute, Impeller and Draft Tube, are based on the geometry and dimension of the pump was generated by CAD software. This action will allow the process of assigning rotational regions to the impeller computational domain, while excluding volute and draft tube. The impeller computational domain was obtained by BladeGen software, known as a design tool for rapid 3D design of rotating machinery components. The front and rear cavity between impeller and pump housing is not included as part of computational domains.

Meshing

The meshing shape was generated by using an unstructured tetrahedral shape for all the domains. This allows fast mesh production with good mesh density control. Mesh independent analysis was performed by observing one sensitive parameter while refining the mesh size. This action helped to decide on optimum mesh size with acceptable level of accuracy. The total number of elements was 248740, 246615 and 83537 for Volute, Impeller and Draft Tube, respectively. The corresponding mesh topology is shown in Figure 3 (a).

Boundary Conditions, Interfaces and Solver

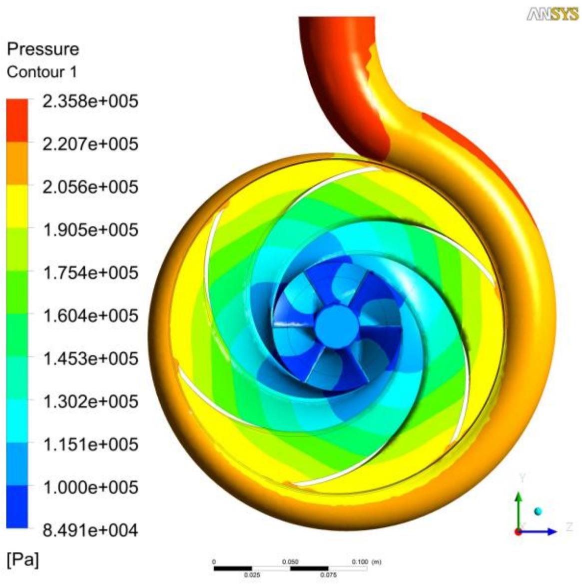
The boundary conditions were set according to their physical flow. The inlet at the impeller eye was set as an opening with a reference pressure of 1 atmospheric pressure. The outlet is located at the end opening of the volute with a boundary condition of mass flow rate, with flow rate at 8.0 l/s. Interfaces between the computational domains were set as general interfaces for both Draft Tube-Impeller and Impeller-Volute. All other surfaces were set as rough walls with surface roughness of 100 micron. The illustrations of boundary condition and interfaces is shown in Figure 3(b). The governing equation used in the simulation consists of steady-state, three-dimensional incompressible Navier-Stokes coupled with k-epsilon as the turbulence model. Water was used as the fluid for the simulation with density of 1000 kg/m3 and at a temperature of 25^0C. The Root Mean Square (RMS) that was set to 0.0001, in order to ensure good convergence for the numerical iteration.

RESULT AND DISCUSSION

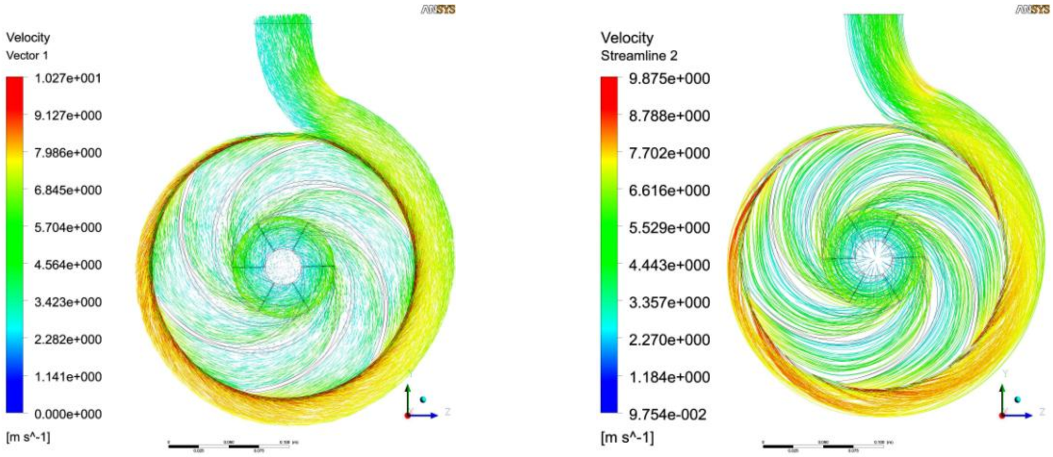
Steady-state simulation results are shown in Table 1, for the flow rate, pressure, power and efficiency. The highest difference was recorded for the torque generated by the impeller. This can be explained by the act of excluding the flow domains between the impeller and pump housing; excludes the energy losses that contribute to the volumetric and disk friction losses, thus explaining the lesser power obtained to achieve the best efficient point. In addition, the limitation of CFD software which does not take into account the mechanical contact friction at the pump shaft and mechanical seal pack.

Table1. *Simulation Results and Manufacturer Data Sheet at 8.0 l/s*

	HBEP (m)	P shaft (kW)	η (%)
Manufacturer Data Sheet	14.0	1.70	65.0
Simulation Result	13.4	1.45	72.4



(a)



(b) (c)

Figure4. *(a) Pressure distribution (b) Velocity vector (c) Velocity Streamline*

Figure 4 (a) shows the static pressure distribution at the surfaces of the flow domains. From the figure, it can be observed that static pressure increases steadily from the impeller eye to the outer diameter of the impeller.

The blue, dark region at the eye of the impeller is where cavitation normally takes place. If the pressure lower than saturation pressure, the water will emit vapour bubbles and abruptly collapse as water flows along the vanes. In this case, the static pressure at the eye of the impeller is higher than net positive suction head (NPSH). The static pressure at the pressure surface of the blades is higher than at the suction surface of the blades. The static pressure continues to increase as it leaves the volute toward the pump outlet.

Figure 4 (b) illustrated the velocity distribution which give an insight the trajectory of fluid particles inside the pump. It can be observed that there is a flow separation at the leading edge of the impeller. The phenomenon take place when boundary layer detaches from the surface of the impeller blade. This changes the tangential velocity entry of the water to the impeller blades and led to hydraulic losses in the pump and affect the flow field between the impeller blades. The losses intensify during off-design operations due to the change of the inlet velocity, as reported by Cheah et. al. [5].

Figure 4 (c) shows the velocity stream line starting from the pump inlet. The velocity stream line reveals that the water flow path has a smooth trail following the blade profile. The stream line shows swirl flow at the draft tube and impeller inlet. This intensify the non-tangential entry at the impeller leading edge and contribute to hydraulic losses.

CONCLUSION

In this study, a simulation analysis of a single-stage, end-suction centrifugal pump with a specific speed of 1075.0 (mm/mm) by using commercial CFD software was presented. The result obtained from the simulation shows good correlation with the manufacturer data sheet. Furthermore, the interaction of fluid by means of pressure gradient, velocity distribution and velocity stream line was also presented. For future work, the simulation model will be used to study off-design operations by adjusting the flow rate. This will further clarify the steep performance of pump efficiency and enhance academic and practical understanding of pump hydraulic performance.

ACKNOWLEDGEMENT

Special thanks are given for the financial support for this work provided by Research and Innovation Management Centre (RIMC), University Malaysia Sarawak under PhD Student Fund (UNIMAS/TNC(PI)-04/09-01/22Jld1(31).

REFERENCES

[1] J. F. Gülich, *Centrifugal Pump*. Berlin: Springer, 2007.

[2] M. T. Stickland, T. J. Scanlon, J. F. Blanco, and J. Parrondo, "A Numerical and Experimental Analysis of Flow in a Centrifugal Pump," in *ASME Joint US-European Fluids Engineering Conference*, 2002.

[3] C. W. S. Maitelli, V. M. Bazerra, and W. da Mata, "Simulation of Flow in a Centrifugal Pump of ESP System Using Competational Fluid Dynamics," *Brazillian ournal of Petroleum and Gas*, vol. 4, pp. 001-009, 2010.

[4] M. Guptaa, S. Kumarb, and A. Kumarc, "Numerical Study of Pressure and Velocity Distribution Analysis of Centrifugal Pump," *International Journal of Thermal Technologies*, vol. 1, 2011.

[5] K. W. Cheah, T. S. Lee, S. H. Winoto, and Z. M. Zhao, "Numerical Flow Simulation in a Centrifugal Pump at Design and Off-Design Conditions," *International Journal of Rotating Machinery*, 2007.

AUTHORS' BIOGRAPHY

Mr. Mohd Azlan Ismail

Mohd Azlan Ismail is a PhD candidate in Mechanical Engineering at University Malaysia Sarawak, Malaysia. His research interest is in renewable energy focused on microhydro power for rural electrification. He graduated from University Malaysia Sabah and has a Master Degree in Mechanical Engineering Practice from University of Wollongong, Australia. Mohd Azlan Ismail is a graduate member of Institute of Engineers, Malaysia.

Dr. Al Khalid Othman

Dr. A.K. Othman was born on July 12, 1971, in Kuching Sarawak, Malaysia. He is graduated his first degree in BEng (Hons) in Electrical and Electronic Engineering in Nottingham Trent University, U.K in 1995 and Masters in Information Technology (Digital) at Nottingham University, U.K. in 1996. He gained PhD in Engineering, specialized in Underwater Acoustic Network at Newcastle University, U.K. in 2007. He became a Member (M) of Board of Engineers Malaysia since 16 November 1999. His research interests include underwater communication, wireless protocol, networking and renewable energy.

Dr. Hushairi Zen

Dr. Hushairi Zen obtained his Doctor of Philosophy (PhD) in 2009 at Edith Cowan University, Perth in Wireless Local Area Network. His area of expertise include wireless communication, data and voice networking, remote monitoring and control, and renewable energy.