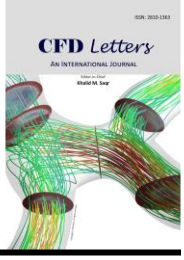
CFD Letters

Journal homepage:
https://semarakilmu.com.my/journals/index.php/CFD_Letters/index
ISSN: 2180-1363



Application of Polyhedral Mesh for Vortex Formation Study for Simple Single Pump Sump

Mohamad Lutfi Samsudin[1,2,*], Hasril Hasini[2]

[1] Malakoff Power Berhad, Level 11, Block 4, Plaza Sentral, Jalan Stesen Sentral 5, 50470 Kuala Lumpur, Malaysia
[2] Department of Mechanical Engineering, College of Engineering, Universiti Tenaga Nasional, Jalan IKRAM-UNITEN, 43000 Kajang, Selangor, Malaysia

ARTICLE INFO

Article history:
Received 28 June 2021
Received in revised form 10 August 2021
Accepted 11 August 2021
Available online 13 September 2021

Keywords:
Vortex; pump; CFD; polyhedral

ABSTRACT

Meshing of domain in CFD is an important step to ensure accuracy of the solution. In the past, hexahedral or tetrahedral mesh systems were commonly used, and both have their merits and demerits. For large and complex geometry, polyhedral is another option but its accuracy is claimed to be lacking. In this paper, the use of polyhedral mesh system by past researchers are reviewed. Evaluation on the application of polyhedral mesh system for the study of the vortex formation with a simple single pump sump model is made. Validation was made through the comparison of the results from hexahedral, tetrahedral and polyhedral mesh sizes and the experimental data from published data. The polyhedral mesh system was found to perform satisfactorily and was able to match the results from the hexahedral mesh system as well as the experimental data.

1. Introduction

Vortex formation is a common problem for submerged vertical pump. It occurs as surface or subsurface vortices and could lead to noise and vibration for the pump. The resulted premature failure of pump components reduces the plant availability and increase maintenance costs.

Typically, vortex formation is studied with experimental rigs during the design stage or during the evaluation of the anti-vortex device (AVD) design. These were highlighted in a number of previous works [1-5]. Sump dimension or approach flow condition is optimized to minimize or eliminate the chances for adverse flow patterns that could result in vortex formation. An alternative method for studying vortex formation which is through the computational fluid dynamics (CFD) is emerging and evolving. Results from this method had been proven to be acceptably in good agreement with the experimental method [6-12].

Rajendran *et al.,* [13] conducted both experiment and CFD simulation for his simple single pump sump model. Results from his work were used by past researchers to validate their models [14-17]. Rajendran *et al.,* [13] assessed the formation of surface and subsurface vortices at five locations

* *Corresponding author.*
E-mail address: mlutfi.samsudin@malakoff.com.my (Mohamad Lutfi Samsudin)

https://doi.org/10.37934/cfdl.13.9.1327

within the sump. Referring to Figure 1, with D being the diameter of the intake pipe, the referred locations were as follows:

i) Free surface vortex, as observed in a horizontal plane at 0.16D from the water surface, situated at the corner of the backwall and right sidewall.
ii) Floor vortex, as observed in a horizontal plane at distance 0.25D from the floor.
iii) Backwall vortex, as observed in a vertical plane at distance 0.23D from the backwall.
iv) Sidewall-1 vortex, as observed in a vertical plane at distance 0.15D from the left sidewall.
v) Sidewall-2 vortex, as observed in a vertical plane at distance 0.25D from the right sidewall.

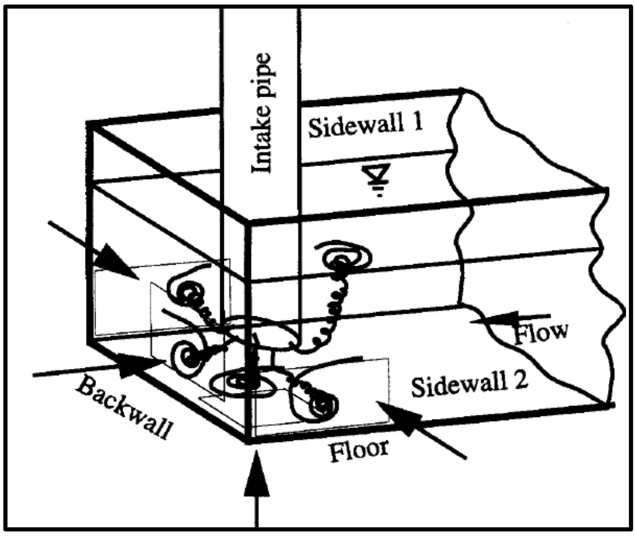


Fig. 1. Five locations for the vortex formation within the asymmetric pump sump [13]

Past researchers studying vortex formation employed either hexahedral mesh, or tetrahedral mesh [18-21]. Hexahedral mesh is preferred since it could give an accuracy level that is equivalent of typically 4 to 10 times the number of tetrahedral mesh [22]. Hexahedral mesh is also characterized by low numerical diffusion especially for the flow perpendicular to the volume faces [23]. However, it is significantly time consuming to generate such mesh for complex geometry, which makes tetrahedral mesh the likely choice. With this mesh, it is relatively easier and faster to generate the mesh but the elements cannot be stretched, thereby requiring a high number of elements. Each element would only have four neighbors which would lead to problems in computing the gradients and possess relatively higher numerical diffusion and converging errors. This, in effect, can result in poor accuracy of the obtained solutions.

Polyhedral mesh is an alternative type of mesh to both hexahedral and tetrahedral meshes. As with tetrahedral mesh, it can be easily applied and quicker to employ for complex bodies but with a lower number of mesh, thereby resulting in faster iterations. While Spiegel *et al.,* [24] employed polyhedral mesh for cerebral hemodynamic simulation, Sosnowski *et al.,* [23,25,27,28], and Sosnowski [26] employed this mesh for flow studies in the fluidized bed chemical loop combustion unit, air-water heat exchanger, an aerodynamics of a vehicle, building arrangement and granular packed bed. It was found that the use of polyhedral mesh is beneficial as the high number of neighbors and low number of elements and iterations can lead to converged and accurate solutions that is comparable to the hexahedral mesh. The application of boundary layer meshes with polyhedral mesh system had been shown not to cause any significant improvement for the accuracy of the near wall profile [24].

Past research with polyhedral mesh in other areas had been conducted with standard $k-\varepsilon$ turbulence model. For the study of pump intake vortex formation, this turbulence model had been known to be poor for the near wall flow characteristics, thus SST $k-\omega$ turbulence model had been proven to be a better model which accounts for both near wall and far field flow [17]. There had not been any past research with polyhedral mesh for the vortex formation, thus it is highly anticipated that the application of this mesh system, together with SST $k-\omega$ turbulence model, can greatly improve the overall CFD modelling and simulation works.

The purpose of this paper is to evaluate the application of polyhedral mesh system for the study of vortex formation in a simple single pump sump model. The results are compared against the hexahedral and tetrahedral mesh systems as well as against the experimental data from the published paper.

2. Methodology

2.1 Model Development

The numerical model for the study was developed based on the experimental model from the research by Rajendran *et al.,* [13], based on Figure 2. The model is for simple single pump model without bell mouth, pipe diameter, D, of 88mm and wall thickness of 6mm. The intake pipe was placed asymmetrically in a sump as intentionally to promote uneven approach flow distribution that would lead to vortex formation, both surface and subsurface vortices around the vicinity of the intake pipe. The intake pipe length for the model was extended to six times the intake pipe diameter, which is beyond the length of four times the diameter for measuring the swirl angle as specified in ANSI/HI 9.8 [29]. This would allow for developed flow and avoid any flow instability within the intake pipe.

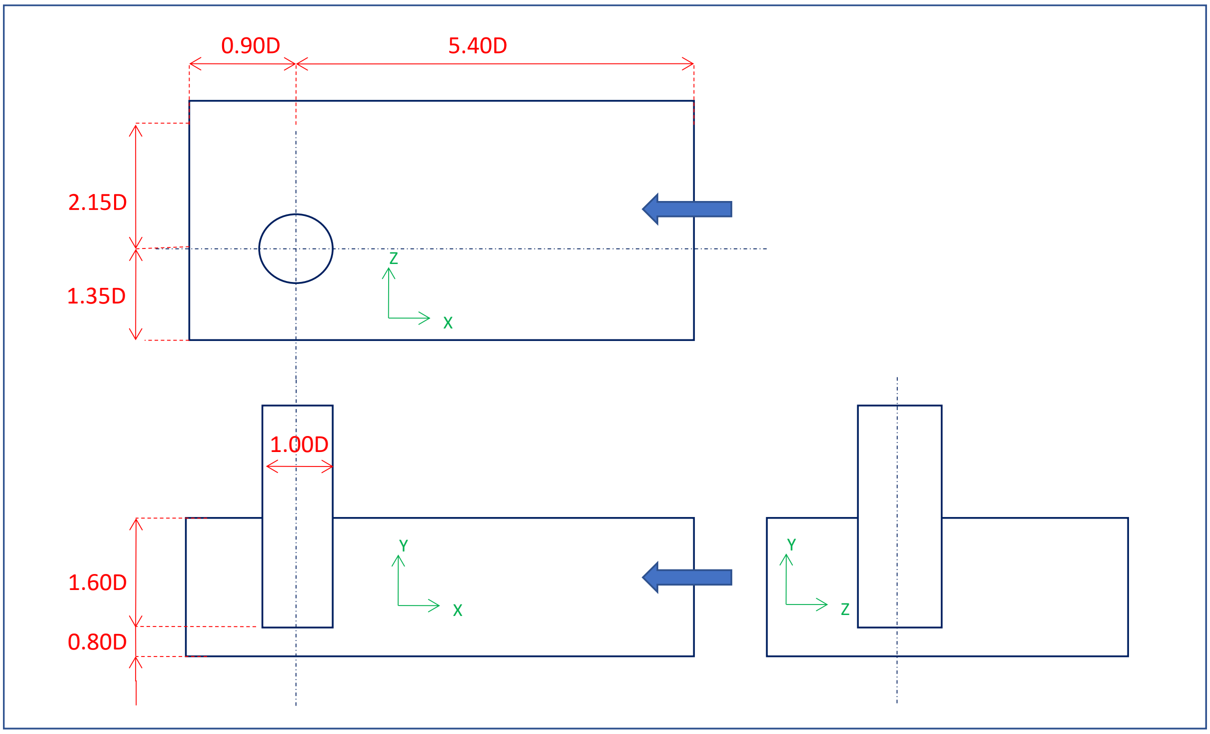


Fig. 2. Dimensions of the simple single pump sump, used by Rajendran *et al.,* [13]

A relatively high submergence of S/D = 1.6 was chosen to avoid the production of strong, air-entraining free surface vortices, with submergence, S defined as the vertical distance from the water level to pipe intake. The velocity of the water inside the vertical intake pipe of 0.492 m/s, the calculated Reynolds Number was 43,149, Froude number was 0.53 and Weber number was 295. These dimensionless parameters are acceptable as such to neglect the effects of surface tension forces (We > 120) and viscous forces (Re > 30,000) as per ANSI/HI 9.8 [29].

2.2 Model Meshing Scheme

The model was meshed with three types of mesh systems – fully hexahedral structured mesh, fully tetrahedral unstructured mesh and polyhedral unstructured mesh, converted from the fully tetrahedral unstructured mesh. The hexahedral mesh system was built based on the mesh count for the edges of multiblock configuration of the model while the polyhedral mesh system was based on the mesh interval size set for the tetrahedral mesh system prior to its conversion. Refer to Figure 3 for the sample comparison of the model with hexahedral and polyhedral mesh systems. Table 1 shows the details of the mesh generation for the mesh systems.

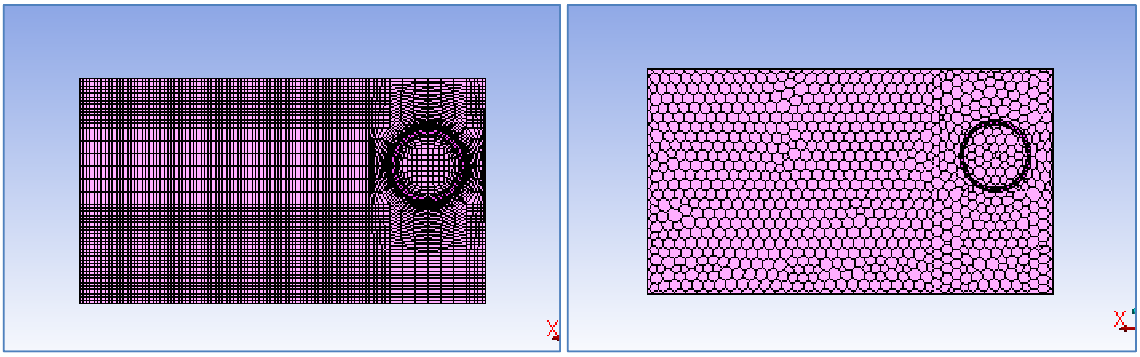


Fig. 3. Sample image for the simple single pump sump model meshed with hexahedral structured mesh (left) and polyhedral unstructured mesh (right)

Table 1
Details of the mesh generation for the simple single pump sump models

Mesh System	Mesh interval counts/sizes	Total no of mesh	Min face area (m^2)	Max face area (m^2)
Full hexahedral	3.6	221,370	2.654e-06	1.024e-04
	5.10	265,950	2.654e-06	6.590e-05
	8.16	338,520	2.654e-06	5.479e-05
	10.20	390,700	2.654e-06	5.479e-05
Hybrid tetrahedral	0.0025m	15,747,305	5.850e-07	1.461e-05
	0.0050m	2,151,959	1.908e-06	4.606e-05
	0.0075m	612,828	5.000e-06	9.264e-05
	0.0100m	266,412	1.090e-05	1.776e-04
	0.0150m	82,570	1.808e-05	3.385e-04
Full Polyhedral	0.0025m	2,708,873	9.880e-08	1.519e-05
	0.0050m	381,359	4.488e-07	4.311e-05
	0.0075m	113,043	1.035e-06	9.464e-05
	0.0100m	50,731	1.083e-06	1.745e-04
	0.0150m	16,935	3.382e-06	3.595e-04

2.3 CFD Simulation

The CFD simulation, which was conducted with ANSYS Fluent, was done based on single-phase and steady-state condition. This was made on the basis that the generated vortices are not strong enough to produce a vortex core filled with air, which is the vortex strength of lower than Type 4 as per ANSI/HI 9.8 [29]. This was practiced by past research which stated that several authors proved that while omitting phase interface at the water surface did not prevent the capturing of the vortices, adopting this would extremely reduce the computational time and complexity of the simulation [30]. The simulation employed SST $k-\omega$ model since it was found to be a better turbulence model in modelling vortex formation in pump sump compared to the $k-\varepsilon$ models such as standard, Realizable and RNG $k-\varepsilon$ models, with respect to the treatment at the near wall without refining the mesh sizes [17]. Rajendran *et al.,* [13], Spiegel *et al.,* [24] and Sosnowski *et al.,* [25] used standard $k-\varepsilon$ models for their research.

For the boundary conditions, water inlet was taken as velocity inlet with velocity magnitude of 0.382 m/s while water outlet was taken as outflow. Water surface was taken as symmetry and all walls were assumed to be without slippage.

3. Results and Discussion

3.1 Difference in the Intake Pipe Discharge Flow

A comparison chart was made as shown in Figure 4 for the total number of cells and the (absolute) percentage deviation in the intake pipe discharge flow at the plane of vertical distance 0.5m from the intake for all mesh systems.

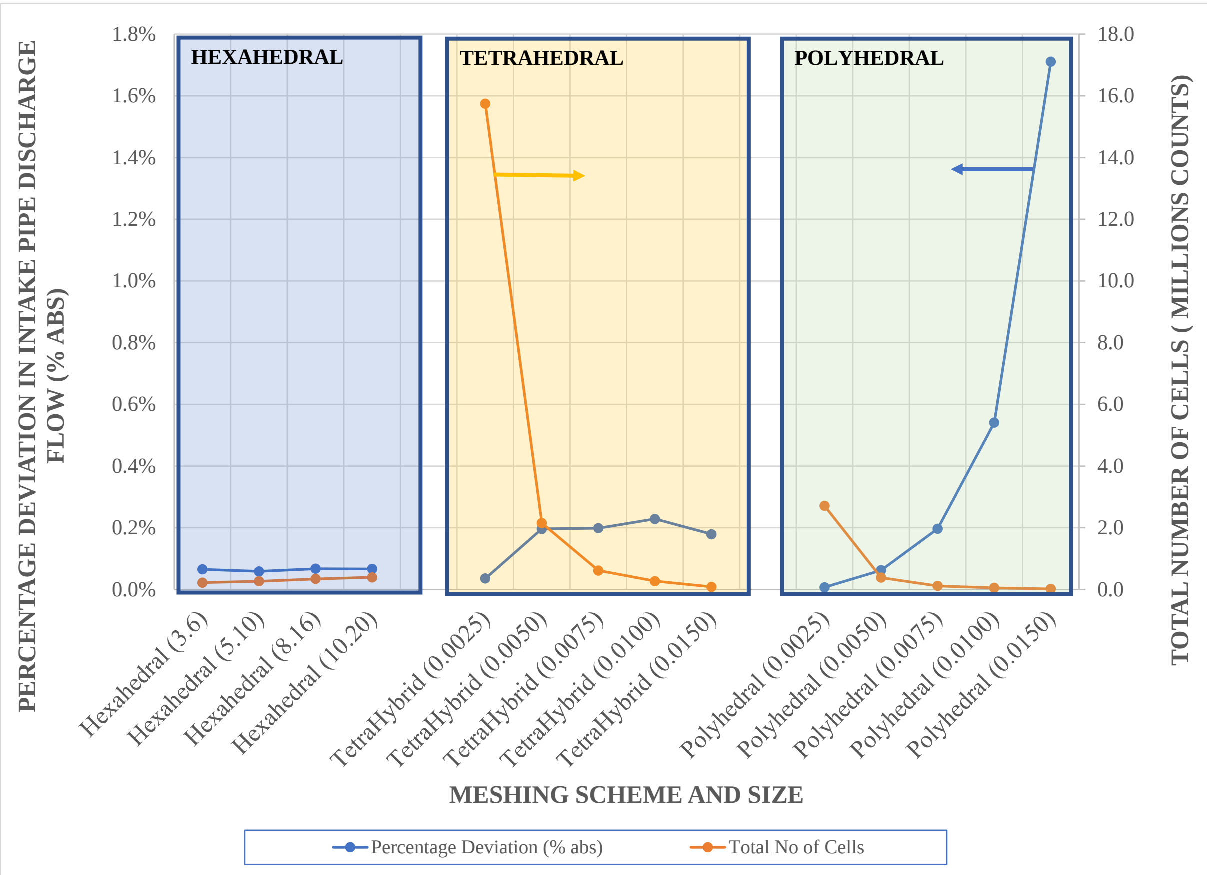


Fig. 4. Comparison of the performance of Hexahedral, Tetrahedral and Polyhedral mesh systems

For hexahedral mesh, the total number of cells increased with the refinement of the grid. Its performance, as reflected by the percentage deviation for the discharge flow, was almost unchanged throughout the varying grid sizes. On the other hand, for polyhedral mesh, the total number of cells decreased as the grid coarsened, thereby resulting in increased discharge flow deviation. The conversion from tetrahedral mesh to polyhedral mesh for the minimum interval size of 0.0050m caused significant drop in the total number of cells by 5.6 times and reduced the deviation by 0.13%. For polyhedral mesh converted from tetrahedral mesh with the minimum interval size of 0.0075m and above, even though the total number of cells is low, the deviation in the discharge flow increased significantly. This was attributed by the fact that the coarse grid at the intake pipe failed to capture the boundary layer flow.

Hexahedral mesh system performed better in comparison to tetrahedral mesh. However, polyhedral mesh converted from tetrahedral mesh with the minimum interval size of 0.0050m and below, performed at the same level and even better when compared to the hexahedral mesh system.

3.2 Comparison of Streamline Plots

The results from the CFD simulation were assessed at five locations of vortex formation for comparison with the work by Rajendran *et al.,* [13]. These locations are illustrated in Figure 1. Figure 5 to Figure 9 show the selected streamline plots obtained from CFD simulation of the simple single pump sump model using selected sizes of the hexahedral, tetrahedral and polyhedral mesh systems.

For backwall vortex as shown in Figure 5, the results obtained by Rajendran *et al.,* [13] shows the subsurface vortex core located closer to the pipe intake while the flow turning observed directly below it. The similar flow characteristics can be seen with the streamlines produced with both hexahedral and polyhedral mesh systems but not for tetrahedral mesh system which shows the vortex core position closer to the floor.

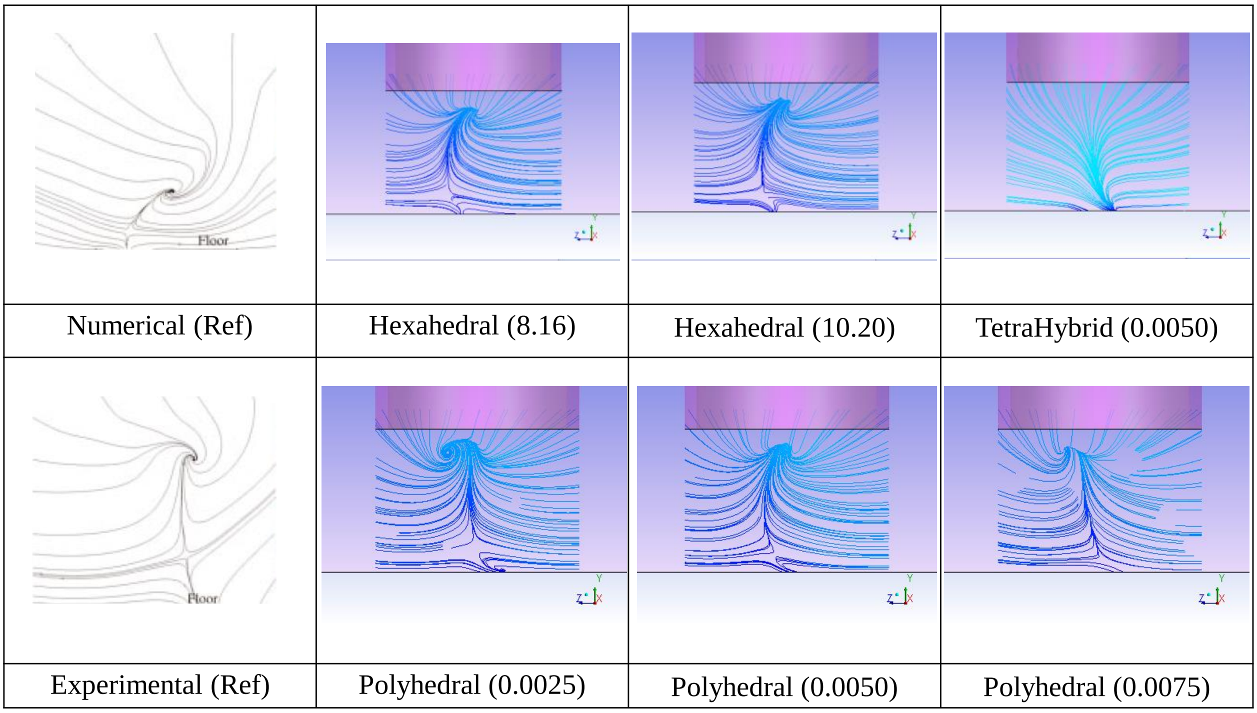


Fig. 5. Streamline plots for the backwall vortex, in comparison to the flow pattern obtained by Rajendran *et al.,* [13]

For floor vortex as shown in Figure 6, the flow characteristics shows similarity in the location of the vortex core which is below the pipe intake. However, the subsurface vortex tailing from the 10 o'clock position could not be seen for tetrahedral mesh system which shows the vortex to be confined within the area directly below the pipe intake.

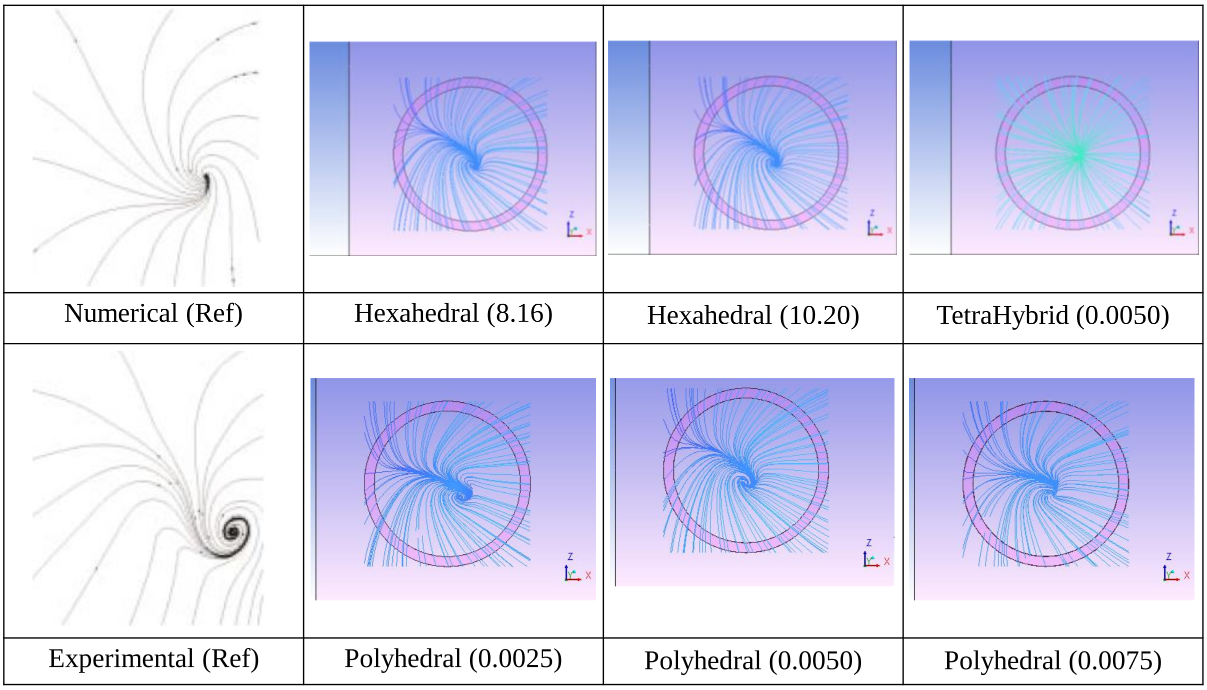


Fig. 6. Streamline plots for the floor vortex, in comparison to the flow pattern obtained by Rajendran *et al.,* [13]

Reviewing the free surface vortex as shown in Figure 7, the circulatory motion and presence of the vortex core can be distinctly seen for both hexahedral and polyhedral mesh systems, which are the same as the result obtained for the experimental work by Rajendran *et al.,* [13]. Finer mesh of polyhedral, for example polyhedral (0.0050) can closely match the results produced by hexahedral, for example hexahedral (8.16). On the other hand, for tetrahedral mesh system, the streamline shows the absence of distinct vortex core.

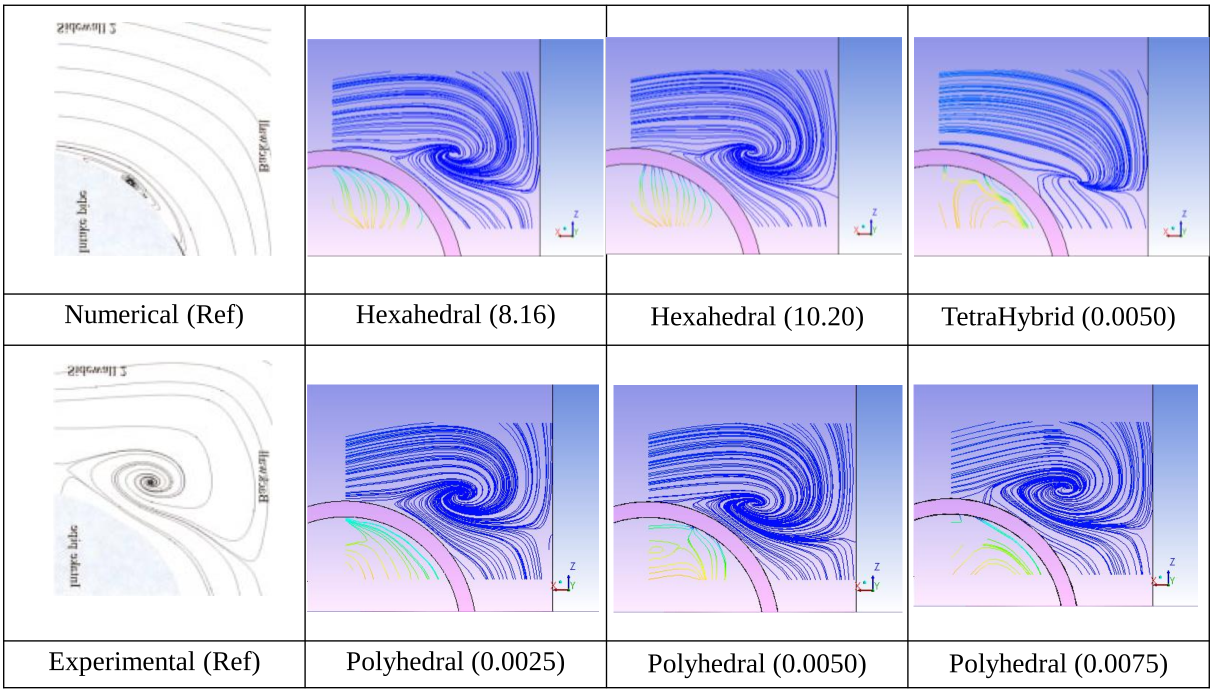


Fig. 7. Streamline plots for the free surface vortex, in comparison to the flow pattern obtained by Rajendran *et al.,* [13]

CFD Letters
Volume 13, Issue 9 (2021) 13-27

While the use of tetrahedral mesh system does not produce any vortex core for both sidewall-1 and -2 vortices as shown in Figure 8 and Figure 9, the use of hexahedral mesh system produces weak subsurface vortex core which is not distinctly seen. For polyhedral mesh system, a relatively coarser mesh, polyhedral (0.0075) shows an exaggerated swirl below the pipe intake.

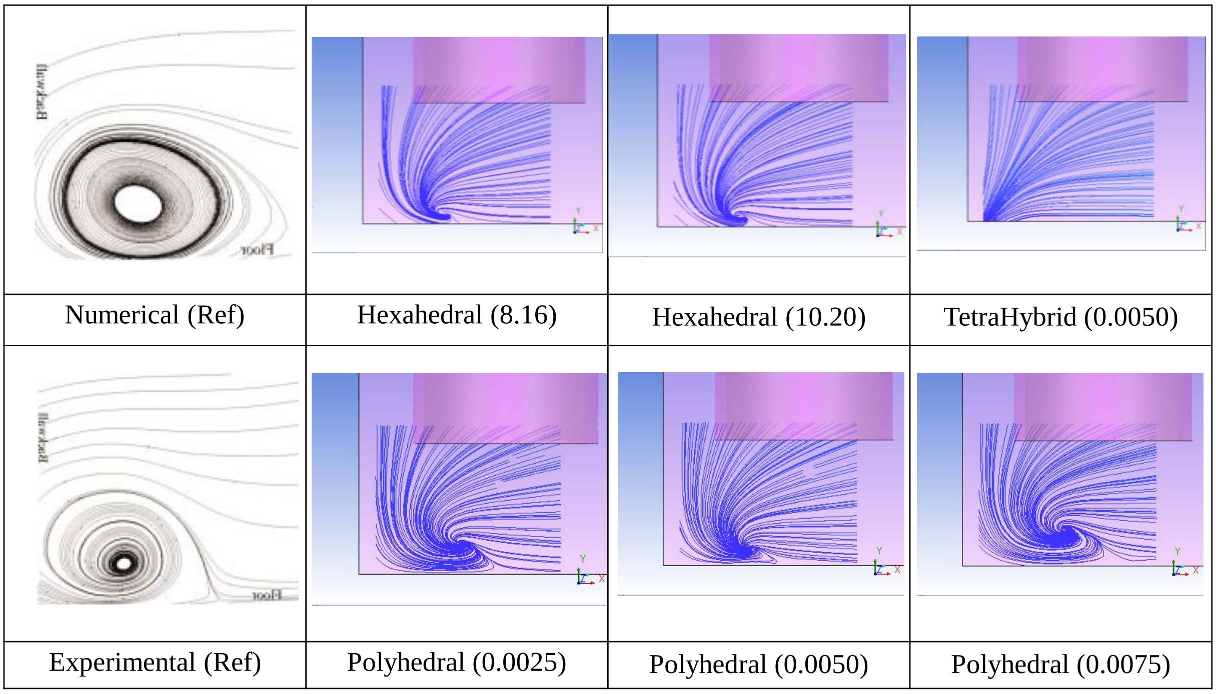


Fig. 8. Streamline plots for the sidewall-1 vortex, in comparison to the flow pattern obtained by Rajendran *et al.,* [13]

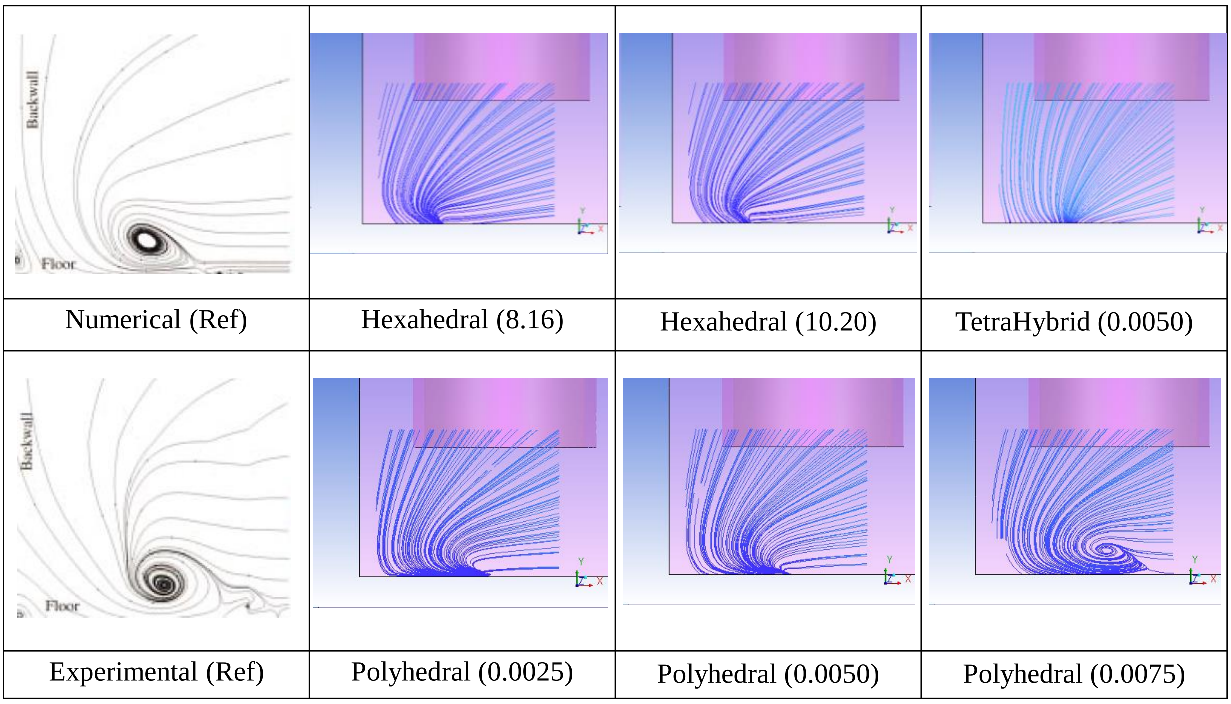


Fig. 9. Streamline plots for the sidewall-2 vortex, in comparison to the flow pattern obtained by Rajendran *et al.,* [13]

The plots at all locations produced by hexahedral mesh systems were qualitatively identical with those published by Rajendran *et al.,* [13]. The simulation done with tetrahedral mesh system produced flow patterns that were distinctly different from those with hexahedral mesh system. For free surface and sidewalls, the vortex and circulation flow were observed to be diminished from the plots. For polyhedral mesh system, the results produced by the 0.0050m interval size were similar with hexahedral mesh system at all locations while the other sizes had some slight differences in their respective plots.

3.3 Comparison of Tangential Velocity Plots

Results for the tangential velocity across the vortex planes at all five locations were plotted for selected sizes for the hexahedral and polyhedral mesh system and compared to the results from the published paper by Rajendran *et al.,* [13]. These are shown in Figure 10 to Figure 14.

The results for backwall vortex using hexahedral (10.20) and polyhedral (0.0050) were the closest compared to the experimental results by Rajendran *et al.,* [13]. Polyhedral (0.0025) and hexahedral (8.16) showed overpredicted results.

For floor vortex, the plots for hexahedral (8.16) and polyhedral (0.0025) were closest with the experimental results while the others underpredicted the vortex (weaker vortex).

Free surface vortex trend plots showed polyhedral mesh systems predicted better than the hexahedral mesh system. All polyhedral mesh systems and hexahedral (8.16) were the closest to the experimental results while hexahedral (10.20) underpredicted the vortex.

For sidewall-1 vortex, both hexahedral and polyhedral mesh systems prediction were far from the experimental results. Meanwhile, for sidewall-2, polyhedral mesh systems prediction was better and closer to the experimental results compared to the hexahedral mesh systems.

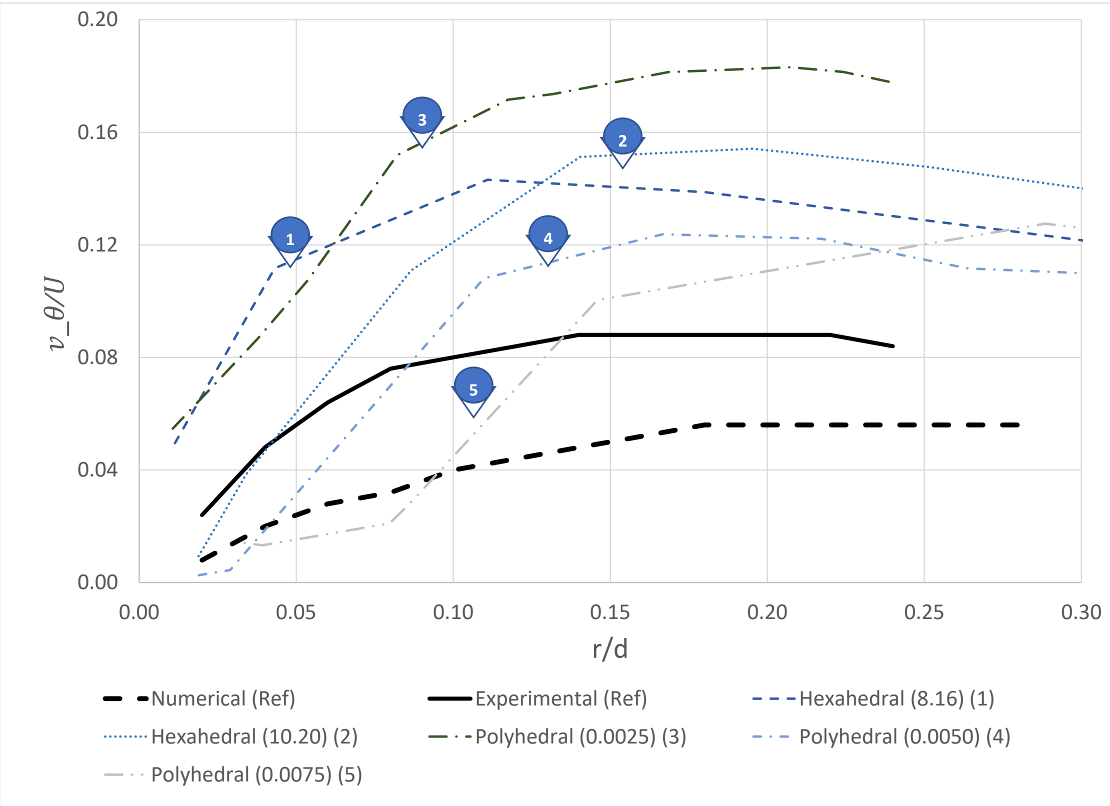


Fig. 10. Tangential velocity plots for backwall vortices for hexahedral and polyhedral mesh systems, in comparison to the results by Rajendran *et al.,* [13]

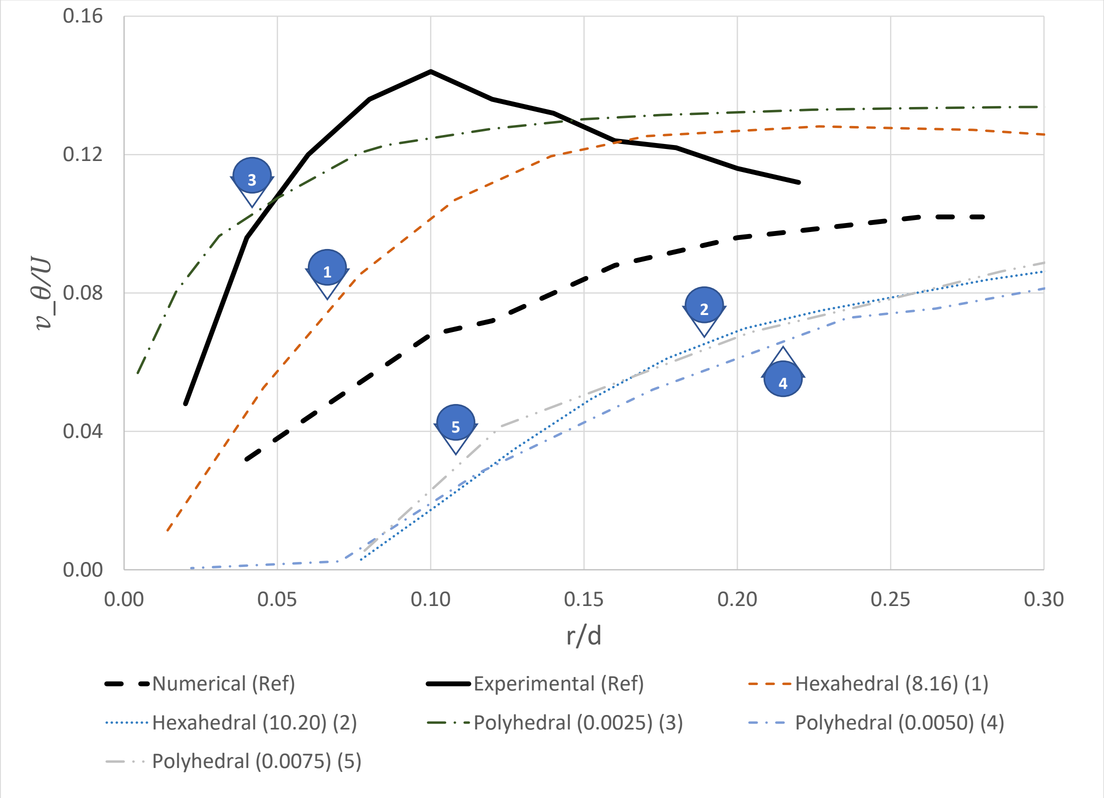


Fig. 11. Tangential velocity plots for floor vortices for hexahedral and polyhedral mesh systems, in comparison to the results by Rajendran *et al.,* [13]

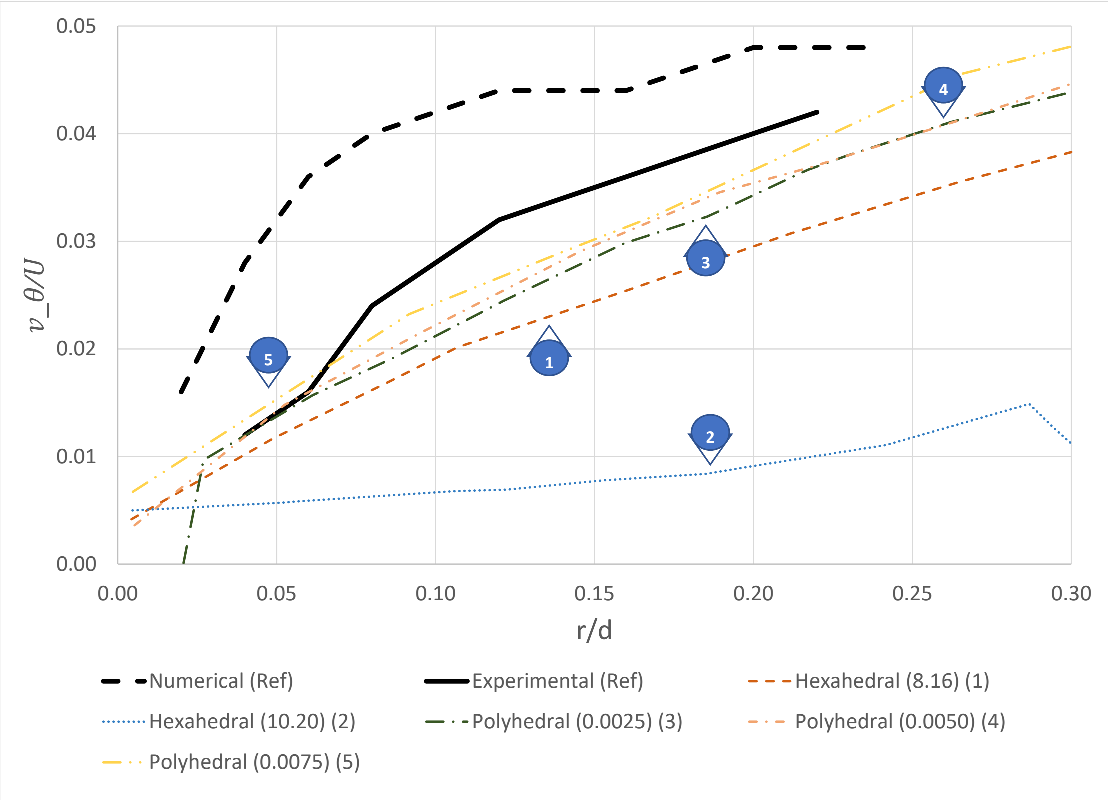


Fig. 12. Tangential velocity plots for free surface vortices for hexahedral and polyhedral mesh systems, in comparison to the results by Rajendran *et al.,* [13]

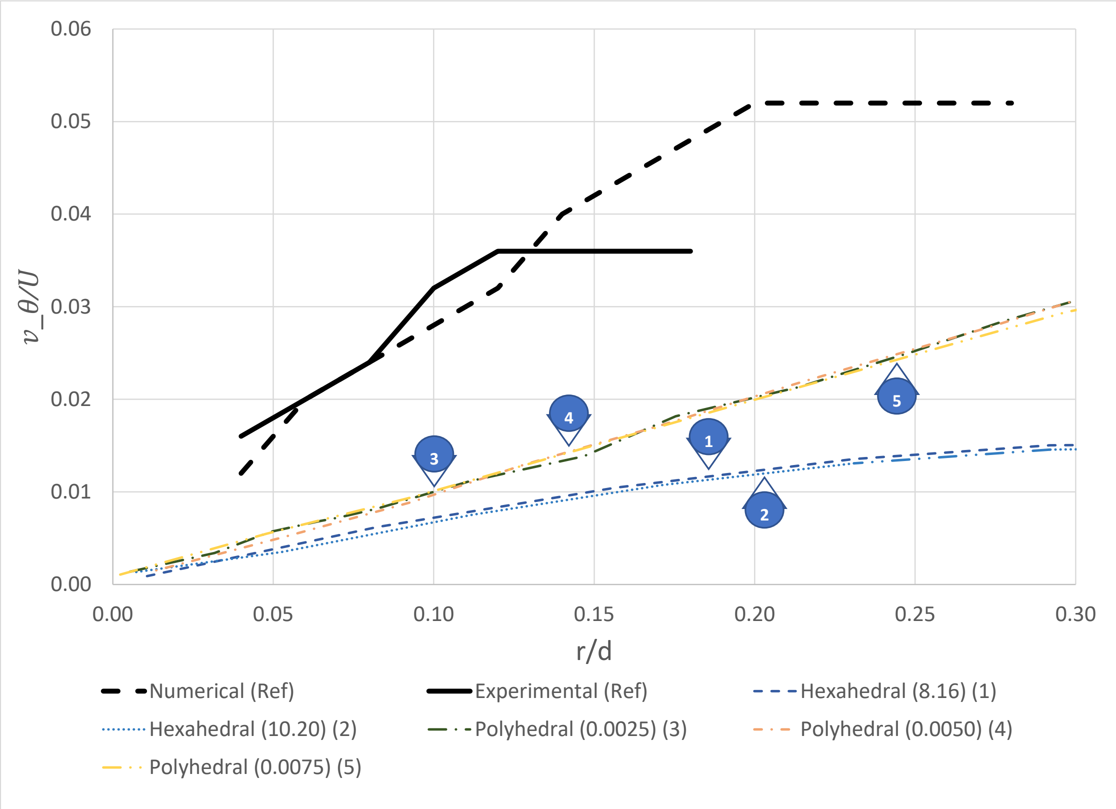


Fig. 13. Tangential velocity plots for sidewall-1 vortices for hexahedral and polyhedral mesh systems, in comparison to the results by Rajendran *et al.,* [13]

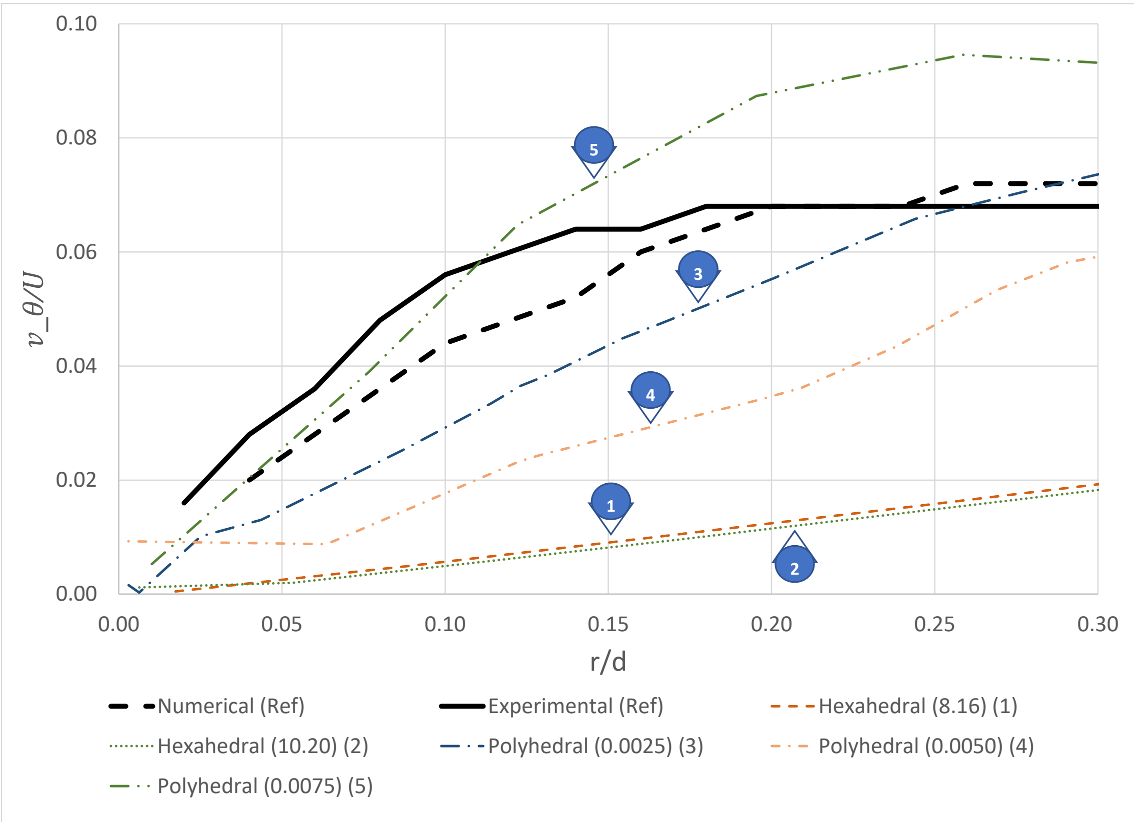


Fig. 14. Tangential velocity plots for sidewall-2 vortices for hexahedral and polyhedral mesh systems, in comparison to the results by Rajendran *et al.,* [13]

While it is clear that polyhedral mesh systems performance could match that with the hexahedral mesh systems, SST $k-\omega$ turbulence model satisfactorily predicts the vortex formation at all five locations to match the experimental results.

3.4 Comparison of Directional Velocity Plots

Plots of directional velocity across the inlet to the intake pipe were made for both hexahedral and polyhedral mesh systems to review the differences in the results. Figure 15 shows the plots of the axial velocity in the y-direction along the z-axis for select mesh systems. Figure 16 shows the same velocity plots at the center of the intake pipe where it is expected to see most velocity variation.

As can be seen, both mesh systems showed almost the same magnitudes for the directional velocity. Small slight deviations between the results by polyhedral and hexahedral mesh systems were observed within a few distance from the intake pipe wall, at maximum of 2.7% and within the center of the intake pipe, at maximum of 5.7%.

There was no difference between hexahedral (8.16) and hexahedral (10.20). Meanwhile the results for polyhedral (0.0050) and polyhedral (0.0075) were close to each other, but the results for polyhedral (0.0025) distinctly deviated from the other coarser mesh.

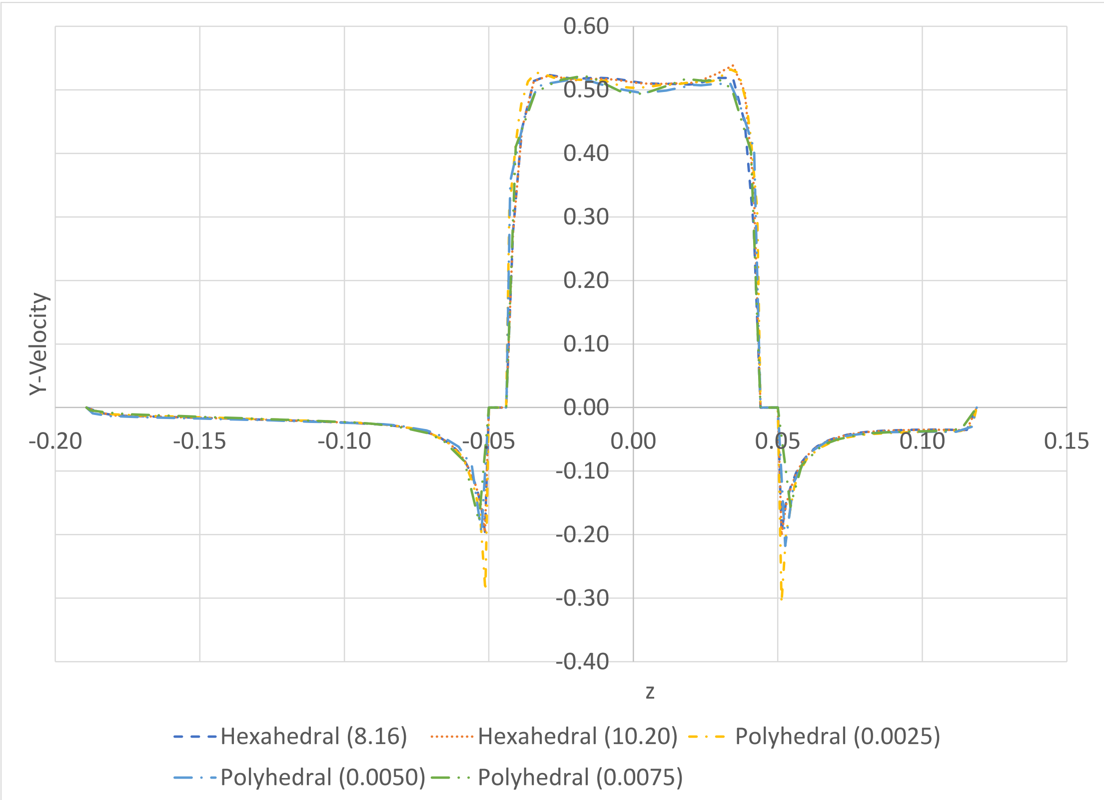


Fig. 15. Directional velocity plot in the y-direction along the z-line axis crossing the pipe intake and sump, covering both hexahedral and polyhedral meshes

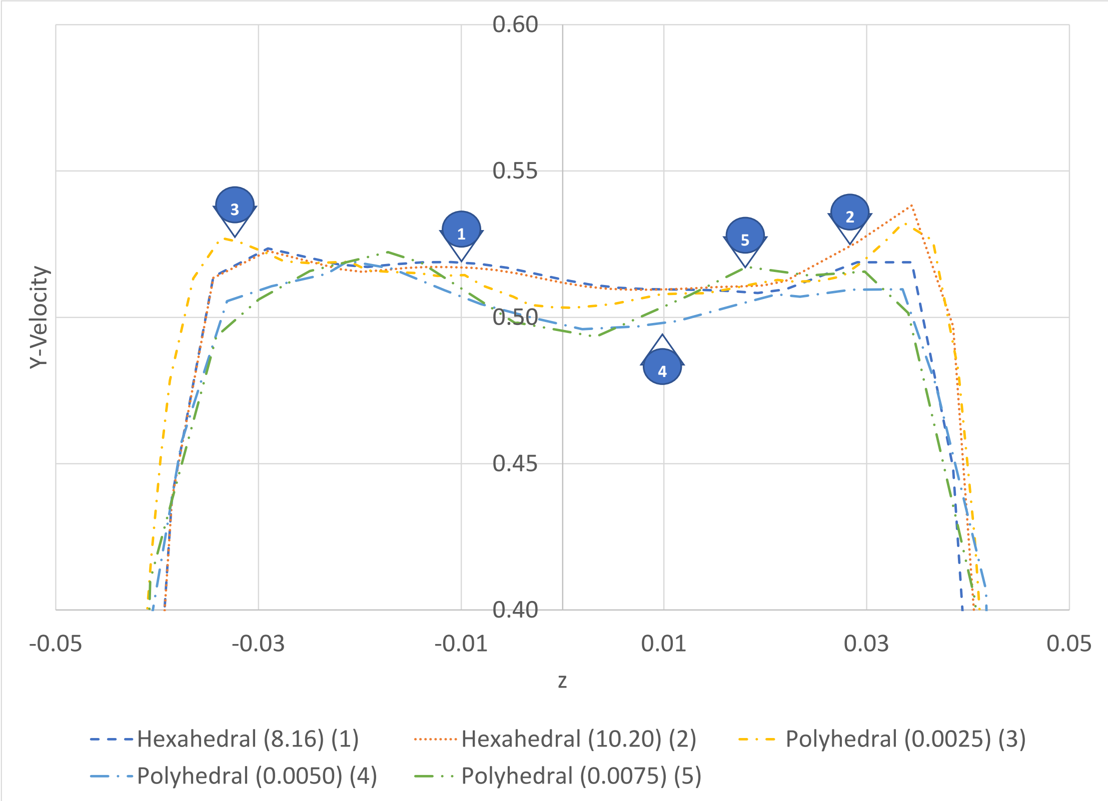


Fig. 16. Directional velocity plot in the y-direction along the z-line axis crossing the centre of the pipe intake, covering both hexahedral and polyhedral meshes

4. Conclusions

CFD simulation of pump intake using different mesh schemes namely hexahedral, tetrahedral and polyhedral are carried out. The focus on the analysis is to predict the surface and subsurface vortices at five different locations in a simple single pump sump model. The comparison of the predicted velocities and vortex cores is made with the experimental and numerical works of Rajendran *et al.,* [13]. From the analysis, the following conclusions can be made:

i. Polyhedral mesh system produces vortex prediction that is comparable with the experimental results. It also matches the results obtained with hexahedral mesh system, with added advantages of easier and faster mesh generation especially with complex geometries.
ii. A coarse polyhedral mesh could underpredict the vortex occurrence while a fine polyhedral mesh could overpredict it. The validation process of the results is essential to gain satisfactory confidence level on the analysis.
iii. SST $k - \omega$ turbulence model can be used with polyhedral mesh and performs satisfactorily to match the experimental results.

Acknowledgement

The authors would also like to acknowledge the financial support received from the Ministry of Education Malaysia under the FRGS scheme (20180110FRGS) that enable the work to be carried out.

References

[1] Shabayek, Shazy A. "Improving approach flow hydraulics at pump intakes." *International Journal of Civil & Environmental Engineering* 10, no. 6 (2010): 23-31.

[2] Ahn, Soo-Hwang, Yexiang Xiao, Zhengwei Wang, Xuezhi Zhou, and Yongyao Luo. "Numerical prediction on the effect of free surface vortex on intake flow characteristics for tidal power station." *Renewable Energy* 101 (2017): 617-628. https://doi.org/10.1016/j.renene.2016.09.021

[3] Batcha, Muhammad Azhar Akbar, Tajul Ariffin Norizan, Zulkhairi Zainol Abidin, Zambri Harun, and Wan Hanna Melini Wan Mohtar. "Vortex Measurement at Bell-Shaped Pump Inlet Using Particle Image Velocimetry." *Jurnal Kejuruteraan* 30, no. 2 (2018): 209-217. https://doi.org/10.17576/jkukm-2018-30(2)-11

[4] Norizan, Tajul Ariffin, Zambri Harun, Shahrir Abdullah, and Wan Hanna Melini Wan Mohtar. "Effects of floor splitter height on the effectiveness of swirl angle reduction in pump intake." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 57, no. 1 (2019): 32-39.

[5] Harun, Zambri, Tajul Ariffin Norizan, Shahrir Abdullah, and Mazharul Islam. "Subsurface Vortex Control Parametric Study at Submersible Pump Intake Using Plate-Type Floor Splitters." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 75, no. 3 (2020): 73-80. https://doi.org/10.37934/arfmts.75.3.7380

[6] Nagahara, Takahide, Toshiyuki Sato, Tomoyoshi Okamura, and Ryuichiro Iwano. "Measurement of the Flow around the Submerged Vortex Cavitation in a Pump Intake by Means of PIV." In *Fifth International Symposium on Cavitation*, pp. 1-7. 2003.

[7] Li, Songheng, Jose Matos Silva, Yong Lai, Larry J. Weber, and V. C. Patel. "Three-dimensional simulation of flows in practical water-pump intakes." *Journal of Hydroinformatics* 8, no. 2 (2006): 111-124. https://doi.org/10.2166/hydro.2006.015b

[8] Okamura, Tomoyoshi, Kyoji Kamemoto, and Jun Matsui. "CFD prediction and model experiment on suction vortices in pump sump." In *The 9th Asian International Conference on Fluid Machinery*, pp. 1-10. 2007.

[9] Zainol, Mohd Remy Rozainy Mohd Arif, I. Abutan, M. Z. Abdullah, and Mohd Ashraf M. I. "Application of Computational Fluid Dynamics (CFD) in Physical Model of Pump Sump to Predict the Flow Characteristics." In *Int Conf on Construction and Bldg Tech (ICCBT-2008)*, pp. 79-90. 2008.

[10] Lucino, Cecilia, Sergio Liscia, and Gonzalo Duro. "Vortex detection in pump sumps by means of CFD." In *XXIV Latin American Congress on Hydraulics Punta A Del Este, Uruguay, (IAHR)*. 2010.

[11] Desmukh, Tanweer S., and V. K Gahlot. "Simulation of flow through a pump sump and its validation." *International Journal of Research and Reviews in Applied Sciences* 4, no. 1 (2010): 7-17.

[12] Kang, Won-Tae, and Byeong-Rog Shin. "An Investigation of Surface Vortices Behavior in Pump Sump." In *11th Asian Int Conf on Fluid Machinery and 3rd Fluid Power Tech Exh* (2011): 592-595.

[13] Rajendran, V. P., S. G. Constantinescu, and V. C. Patel. "Experimental validation of numerical model of flow in pump-intake bays." *Journal of hydraulic Engineering* 125, no. 11 (1999): 1119-1125. https://doi.org/10.1061/(ASCE)0733-9429(1999)125:11(1119)

[14] Tang, X. L., F. J. Wang, Y. J. Li, G. H. Cong, X. Y. Shi, Y. L. Wu, and L. Y. Qi. "Numerical investigations of vortex flows and vortex suppression schemes in a large pumping-station sump." *Proceedings of the Institution of Mechanical Engineers, Part C: Journal of Mechanical Engineering Science* 225, no. 6 (2011): 1459-1480. https://doi.org/10.1177/2041298310393447

[15] Zhan, Jie-min, Ben-cheng Wang, Ling-hui Yu, Yok-sheung Li, and Tang Ling. "Numerical investigation of flow patterns in different pump intake systems." *Journal of Hydrodynamics, Ser. B* 24, no. 6 (2012): 873-882. https://doi.org/10.1016/S1001-6058(11)60315-6

[16] Tawade, M. S., and R. S. Maurya. "Numerical Study of Flow Structure Developing Around Double Suction Pipe of a Pumping System." *Procedia Engineering* 127 (2015): 809-815. https://doi.org/10.1016/j.proeng.2015.11.416

[17] Qian, ZhongDong, PengFei Wu, ZhiWei Guo, and WenXin Huai. "Numerical simulation of air entrainment and suppression in pump sump." *Science China Technological Sciences* 59, no. 12 (2016): 1847-1855. https://doi.org/10.1007/s11431-016-0237-8

[18] Issa, Abir, Annie-Claude Bayeul-Lainé, and Gérard Bois. "Numerical Simulation of Flow Field Formed in Water Pump-Sump." In *IAHR-24th Symposium on Hydraulic Machinery and Systems*, pp. 1-11. 2008.

[19] Chen, Z. M., Y. B. Lee, K. Y. Kim, S. H. Park, and Y. D. Choi. "Effect of suction pipe leaning angle and water level on the internal flow of pump sump." In *IOP Conference Series: Earth and Environmental Science*, vol. 49, no. 3, p. 032002. IOP Publishing, 2016. https://doi.org/10.1088/1755-1315/49/3/032002

[20] Shin, B. "Numerical Study of Effect of Flow Rate on Free Surface Vortex in Suction Sump." *Transactions of the Japan Society for Computational Engineering and Science* 2018 (2018).

[21] Zihao, Mi, Zhou Daqing, and Mao Yuanting. "Three-dimensional CFD simulation of inlet structure flow in pumping station based on Eulerian solid-liquid two-phase flow model." *Journal of Drainage and Irrigation Machinery Engnieering* 33, no. 6 (2015): 494-498.

[22] Shepherd, Jason F., and Chris R. Johnson. "Hexahedral mesh generation constraints." *Engineering with Computers* 24, no. 3 (2008): 195-213. https://doi.org/10.1007/s00366-008-0091-4

[23] Sosnowski, Marcin, Jaroslaw Krzywanski, and Renata Gnatowska. "Polyhedral meshing as an innovative approach to computational domain discretization of a cyclone in a fluidized bed CLC unit." In *E3S Web of Conferences*, vol. 14, p. 01027. EDP Sciences, 2017. https://doi.org/10.1051/e3sconf/20171401027

[24] Spiegel, Martin, Thomas Redel, Y. Jonathan Zhang, Tobias Struffert, Joachim Hornegger, Robert G. Grossman, Arnd Doerfler, and Christof Karmonik. "Tetrahedral vs. polyhedral mesh size evaluation on flow velocity and wall shear stress for cerebral hemodynamic simulation." *Computer Methods in Biomechanics and Biomedical Engineering* 14, no. 01 (2011): 9-22. https://doi.org/10.1080/10255842.2010.518565

[25] Sosnowski, Marcin, Jaroslaw Krzywanski, Karolina Grabowska, and Renata Gnatowska. "Polyhedral meshing in numerical analysis of conjugate heat transfer." In *EPJ Web of Conferences*, vol. 180, p. 02096. EDP Sciences, 2018. https://doi.org/10.1051/epjconf/201817002096

[26] Sosnowski, Marcin. "The influence of computational domain discretization on CFD results concerning aerodynamics of a vehicle." *Journal of Applied Mathematics and Computational Mechanics* 17, no. 1 (2018): 79-88. https://doi.org/10.17512/jamcm.2018.1.08

[27] Sosnowski, Marcin, Renata Gnatowska, Karolina Grabowska, Jarosław Krzywański, and Arkadiusz Jamrozik. "Numerical analysis of flow in building arrangement: Computational domain discretization." *Applied Sciences* 9, no. 5 (2019): 941. https://doi.org/10.3390/app9050941

[28] Sosnowski, Marcin, Renata Gnatowska, Jacek Sobczyk, and Waldemar Wodziak. "Computational domain discretization for CFD analysis of flow in a granular packed bed." *Journal of Theoretical and Applied Mechanics* 57, no. 4 (2019): 833-842. https://doi.org/10.15632/jtam-pl/112017

[29] American National Standards Institute. *American national standard for rotodynamic pumps for pump intake design*. Hydraulic Institute, 2018.

[30] Rudolf, Pavel, and Roman Klas. "Numerical simulation of pump-intake vortices." In *EPJ Web of Conferences*, vol. 92, p. 02077. EDP Sciences, 2015. https://doi.org/10.1051/epjconf/20159202077