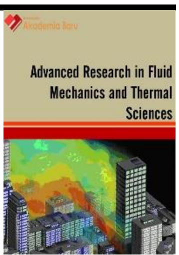
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Numerical Analysis of Calcium Carbonate ($CaCO_3$) Suspension Flow in Pentagon Spiral Pipe with Pitch Variation

Open Access

Gunawan[1,*], Yanuar[1], M. R. Setiawan[2], A. Arianda[2]

[1] Department of Mechanical Engineering, Universitas Indonesia, Jakarta, 16424, Indonesia
[2] Student of Department of Mechanical Engineering, Universitas Indonesia, Jakarta, 16424, Indonesia

ARTICLE INFO

Article history:
Received 14 October 2019
Received in revised form 17 December 2019
Accepted 17 December 2019
Available online 18 March 2020

Keywords:
Energy efficiency; Calcium carbonate suspension; Spiral pipe; drag reduction

ABSTRACT

Energy loss due to friction in fluid transportation was a serious energy efficiency problem. Then, calcium carbonate ($CaCO_3$) comes as a solution to solve this issue. The use of $CaCO_3$ in water flow through the pentagon spiral pipe was effective to create more efficient flow by promoting drag reduction (DR). This research aimed to optimize the pentagon spiral pipe geometry to promotes drag reduction by $CaCO_3$. The analysis has been done by using Computational Fluid Dynamics (CFD) with Ansys Fluent 19.2 in a steady-state flow. The flow model of k-ε RNG (Renormalization Group) was adopted to analyse the pitch variation effect on the drag reduction in various Reynolds numbers. The results showed a strong dependence of the drag reduction on the pitch variation and Reynolds number. The Pentagon spiral pipe geometry can be optimized with modification in pitch according to the fluid application condition.

1. Introduction

This present work shows a comparison between the predictions from a computational fluid dynamics (CFD) model of flow in pentagon spiral pipe and experimental data of drag reduction (DR) by calcium carbonate ($CaCO_3$) that already reported in Sealtial Mau *et al.,* [1]. Ghorai *et al.,* have conducted research that has an aim to propose a global empirical correlation for the interfacial friction factor and create local modeling of wavy stratified air-water flow in the pipe [2]. The result shows things that could not be seen in the experimental method such as velocity profiles in the liquid phase, velocity profiles in the gas phase as well as interfacial roughness in wavy stratified flow.

In early 2019, research about the new method of revealing the DR mechanism was conducted [3]. The research has the aim to establish a DNS framework of gas-liquid drag-reducing flow and use this framework to obtain numerical results and try to explain the mechanism of two-phase drag reduction from the results. Their new method used was Efficient interface tracking method—coupled volume-of-fluid and level set (VOSET) and typical polymer constitutive model Giesekus combined with the momentum equation of the two-phase turbulent flow. Results show that the DR mechanism

---

[*] *Corresponding author.*
E-mail address: gunawan_kapal@eng.ui.ac.id (Gunawan)

can be done due to local enhancement in the core region with local suppression near the walls of turbulent fluctuations. Then, as same as single phase flow, gas-liquid drag-reducing flow can be the global suppression of turbulent fluctuations. These results can be found only if we used the CFD model.

Drag reduction in internal flow also already conducted in 2013 [4]. They analyze the internal flow of the tube covered by the riblet structure using RANS (Reynolds-Averaged Navier-Stokes) and RNG (Renormalization Group) k-ε model to generate the governing equation. The result shows that the riblet structure weakens the velocity fluctuation and strength of turbulence burst in the boundary layer, macroscopically reducing the velocity gradient in near wall region, and consequently diminishing the skin-friction resistance. Another case of the k-ε model already used to analyze the fluid dynamic behaviors of the two-phase pipe flow in 2012 [5]. Results show a verified conclusion to the actual process in the term of crude oil loss of an oil tanker caused by VOC emission. RNG k-ε model itself was first researched to generate the equation for turbulent flow [6, 7].

In terms of passive control, superhydrophobic media is one of the famous ways. It reveals its high relation to drag reduction [8, 9]. Another famous way is spiral pipe. Yanuar *et al.,* investigated the effect of calcium carbonate nanoparticles on the flow in a pentagon spiral pipe. The result shows that the spiral pipe prevents the sedimentation of heavy particles. The geometry of the spiral pipe also generates circumferential flow resulting in the working fluids twisting at a certain Reynolds and it enhances the phenomenon of drag reduction [10].

The aim of this study is to optimize the pentagon spiral pipe geometry to promote drag reduction by $CaCO_3$. Analysis has been conducted by using Computational Fluid Dynamics (CFD) with Ansys Fluent 19.2 in a steady-state flow and RNG k-ε was used as the flow model to solve the governing equation.

2. Methodology

2.1 Mathematical Model

The mathematical model used in this study is obtained from the continuity and momentum equation that differentiated from the Navier-stokes equation that shows below.

$$\frac{d}{dt}(\alpha f \rho f) + \nabla.(\alpha f \rho f \boldsymbol{u} f) = 0 \tag{1}$$

$$\frac{d}{dt}(\alpha_f \rho_f) + \nabla.(\alpha_f \rho_f \boldsymbol{u}_f \boldsymbol{u}_f) = -\alpha_f \nabla p + \nabla.(\alpha_f \tau_f) + \alpha_f \rho_f g - f_{drag} \tag{2}$$

As flow characteristics observed in this research represent a turbulent flow, the flow is visualized by applying the standard and RNG k–ε equations.

$$\frac{d}{dt}(\alpha_f \rho_f k) + \nabla.(\alpha_f \rho_f \boldsymbol{u}_f k) = \nabla.(\alpha_f \Gamma_k \nabla_k) + \alpha_f G - \alpha_f \rho_f \varepsilon \tag{3}$$

$$\frac{d}{dt}(\alpha_f \rho_f \varepsilon) + \nabla.(\alpha_f \rho_f \boldsymbol{u_f} \varepsilon) = \nabla.(\alpha_f \Gamma_\varepsilon \nabla \varepsilon) + \alpha_f \frac{\varepsilon}{k}(c_1 G - c_2 \rho_f \varepsilon) \tag{4}$$

In this study, the equations are solved in ANSYS Fluent 19.2 to execute CFD simulations. Besides, the scientifically detailed flow issue is controlled exclusively by the Reynolds number.

2.2 Geometry Model

In this study, the model used is the pentagon spiral pipe that modeled in Autodesk Inventor 2019. The pentagon spiral pipe characteristic is same as spiral pipe used in the past study conducted by Sealtial Mau *et al.,* [1]. Figure 1 shows the geometry of spiral pipe being modeled.

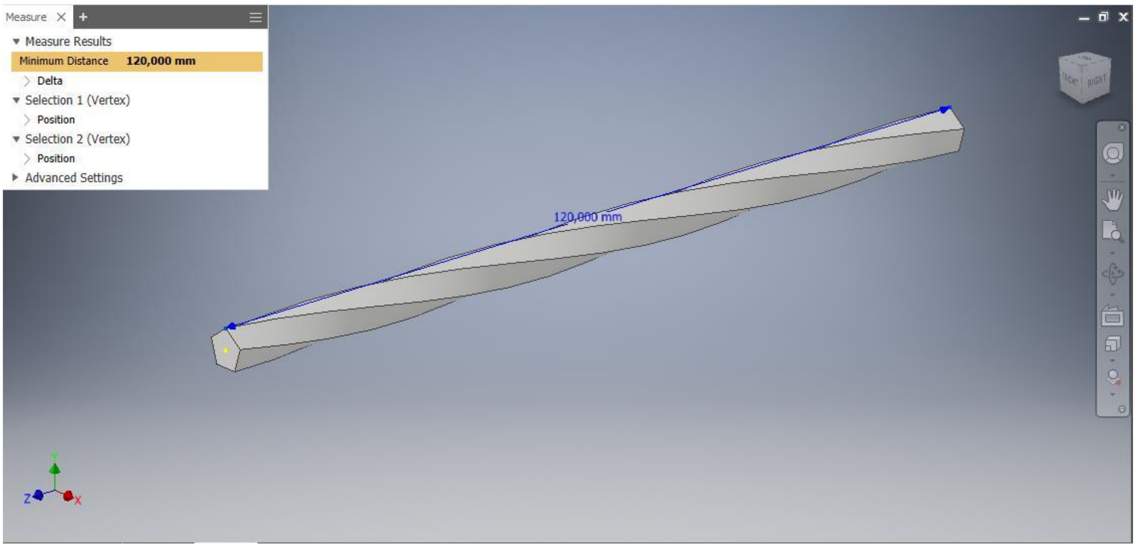

Minimum Distance 120,000 mm

Fig. 1. Pentagon spiral pipe model

Since we are focusing on our research on the pitch variation effects for drag reduction, we use three different geometry models with diameter ratio (P/D) of 7.1, 7.6, and 8.1 respectively. The P/D of 7.1 has been used in the previous experimental research conducted by Sealtial *et al.,* [1] for water flow drag reduction using $CaCO_3$. Next, P/D of 7.6 variation has been used because according to Yanuar *et al.,* [11] it is the best P/D for drag reduction in a three-lobed spiral pipe. The P/D of 8.1 is another variation that use to more clearly investigation the effect of P/D variations on the drag reduction phenomenon.

Table 1
Dimensions of spiral pipe

Pipe	P/D
Spiral 1	7.1
Spiral 2	7.6
Spiral 3	8.1

2.3 Viscosity Model

The viscosity model that will be used is the Power Law model due to the change of the fluid viscosity. Then the effective viscosity model is stated as:

$$\mu_{eff} = K(\frac{du}{dy})^{n-1} \tag{5}$$

where K is a consistency index. Moreover, shear stress, τ, is given by the power of n for the velocity gradient.

2.4 Verification and Validation

The verification has been done by using mesh independency study to determine the perfect number of elements for accurate results with efficient use of time. From the conducted mesh independency study (Figure 2), it can be concluded that from 600,000 elements total we can obtain a reliable result.

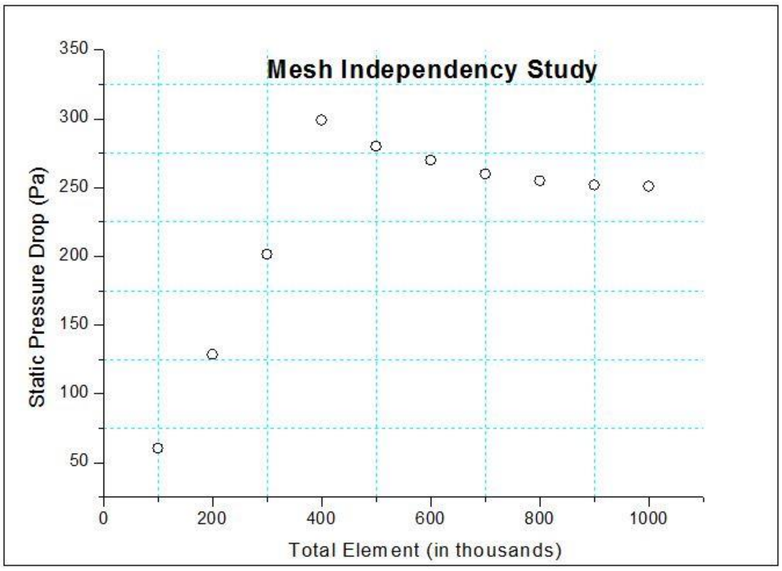


Fig. 2. Mesh independency study

In consideration of the mesh independency from 800,000 total elements onwards, we conduct the simulation with 600,000 elements in totals since according to Figure 3. the time consumed for analyzing time is increasing exponentially after 600,000 elements total. For the meshing model in this study, we adopted the hexahedral meshing one to investigate deeper about fluid complex flow [12].

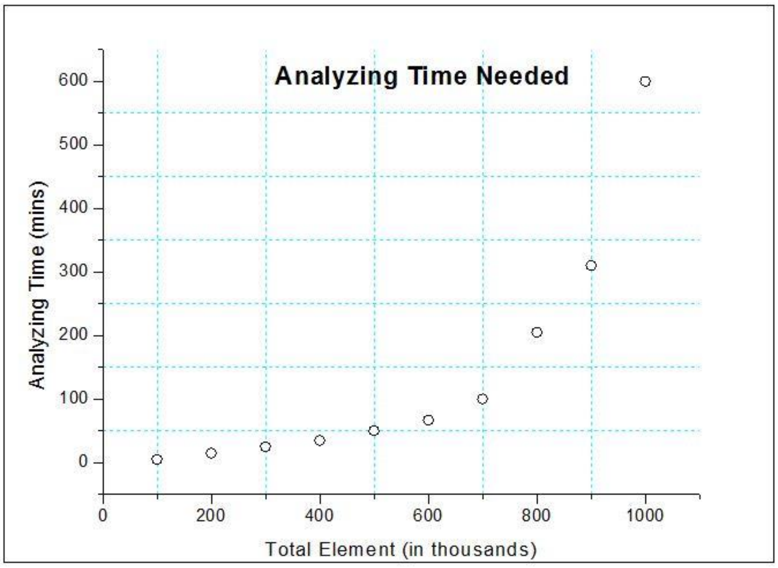


Fig. 3. Analyzing time

Validation of this research has been done by comparing the results with the previous experimental research conducted by Sealtial *et al.,* [1]. The simulation results are in good agreement with the previous experiment with less than a 5 percent margin of error. Figure 4 shows the validation result of this work.

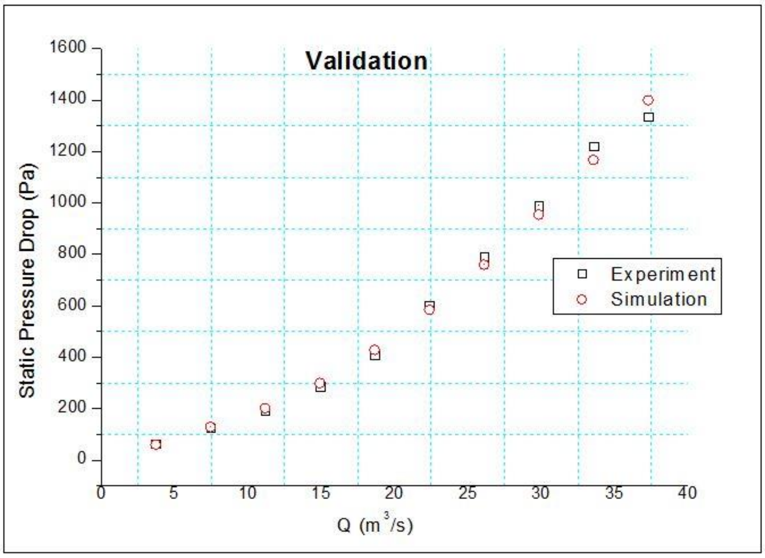


Fig. 4. Validation table

3. Results

Figure 5 shows that the pressure drops tend to rise as the number of ratios of diameter (P/D) rising. This phenomenon could be confirmed by Yanuar *et al.,* [11] who conducted research on pitch variation of crude oil flow by experimental method. The more pressure drops rising, the more value of drag reduction (DR) occurs.

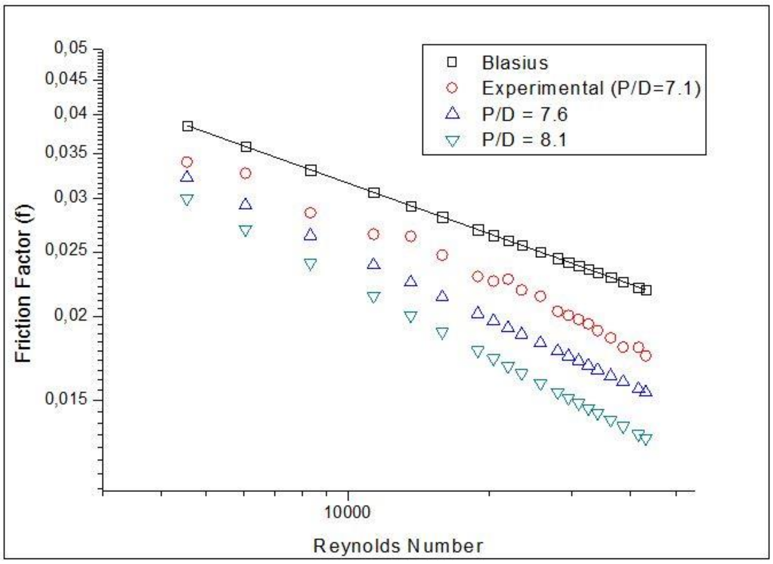


Fig. 5. Relationship between friction factor and Reynolds number

By positive correlation, it is also confirmed that a number of pitch ratio rising, DR% could tend to rise too. Figure 6 reveals that maximum drag reduction occurs at the number of 40% on P/D = 8.1. This happens because the spiral pipe with large P/D occurs smoother transition in the swirl formation.

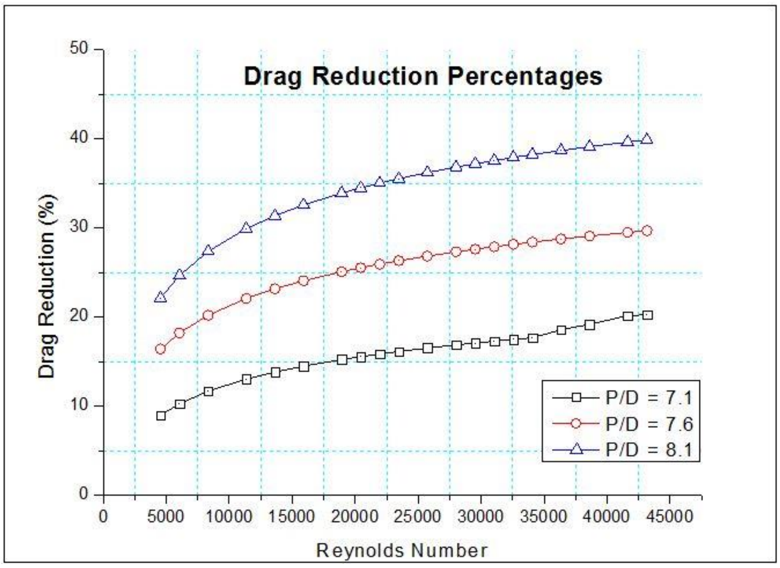


Fig. 6. Drag reduction

The highest DR% is achieved when the swirling effect of the spiral pipe is in full swing at over 40,000 in Reynolds number. Higher tangential velocity creates higher turbulence to increase the drag reduction effect [13].

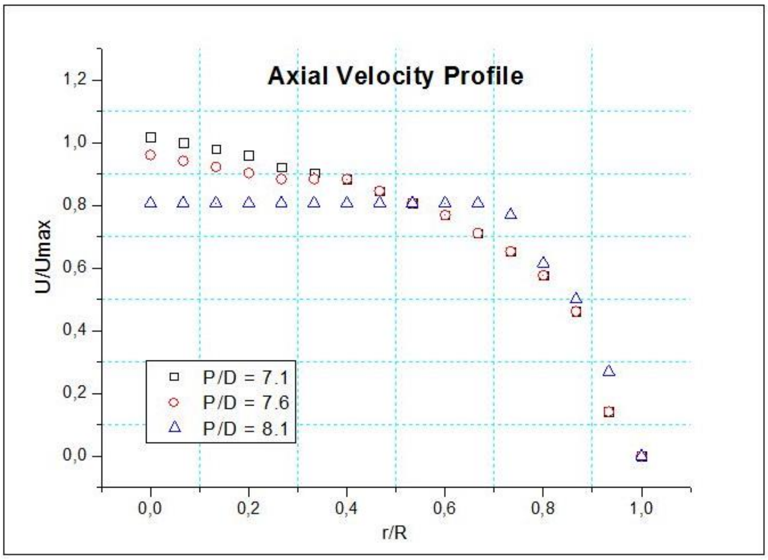


Fig. 7. Axial velocity profile

It can be seen from Figure 7 that different pitch frequency creates differences in the axial velocity profile. The best P/D variation creates a more evenly distributed local axial velocity magnitude. The axial velocity distribution result indicated the low drag occurred in this flow. This happens because of the swirling effect. Furthermore, the core of turbulence and axial velocity is highly correlated with swirl intensity [7].

4. Conclusions

The increase in P/D creates a better DR due to the increase of swirl number. This happens because smoother transition in swirl formation is happening in the pipe with higher P/D. Besides, the larger P/D creates a more effective flow based on the more evenly axial velocity distributed, it indicated the low drag occurred in this flow.

Furthermore, the major conclusion that can be pulled from this study is pitch variation has a massive impact in terms of drag reduction of water flow. This result also confirmed the past study that researched the same thing by the experimental method of crude oil flow [11]. The others, this study could complete the state of arts of past study and also confirm its conclusions [1].

Acknowledgment

This research was funded by a grant from Directorate of Research and Community Engagement Universitas Indonesia with PIT 9 Scheme. Contract number: NKB- 0083/UN2.R3.1/HKP.05.00/2019.

References

[1] Mau, Sealtial. "Effect of calcium carbonate solution on drag reduction in a pentagon spiral pipe." *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences* 46, no. 1 (2018): 41-48.

[2] Ghorai, Subhashini, and K. D. P. Nigam. "CFD modeling of flow profiles and interfacial phenomena in two-phase flow in pipes." *Chemical Engineering and Processing: Process Intensification* 45, no. 1 (2006): 55-65.

[3] Wang, Yi, Yan Wang, and Zhe Cheng. "Direct Numerical Simulation of Gas-Liquid Drag-Reducing Cavity Flow by the VOSET Method." *Polymers* 11, no. 4 (2019): 596.

[4] Wei, Zhi Guo, Yong Li, Zhi Wu Ke, and Xian Ling Li. "Numerical Simulation of Internal Flow Drag Reduction with Triangular Riblet Surface." In *Advanced Materials Research*, vol. 779, pp. 401-405. Trans Tech Publications, 2013.

[5] Kim, Woohyun, Munkyu Yoon, Moonyong Lee, and Sunwon Park. "CFD Analysis of Cavitation in a Crude Oil Pipeline to an Oil Tanker." In *Computer Aided Chemical Engineering*, vol. 31, pp. 580-584. Elsevier, 2012.

[6] Speziale, Charles G., and S. Thangam. "Analysis of an RNG based turbulence model for separated flows." *International Journal of Engineering Science* 30, no. 10 (1992): 1379-IN4.

[7] Chang, F., and V. K. Dhir. "Turbulent flow field in tangentially injected swirl flows in tubes." *International Journal of Heat and Fluid Flow* 15, no. 5 (1994): 346-356.

[8] Costantini, Roberta, Jean-Paul Mollicone, and Francesco Battista. "Drag reduction induced by superhydrophobic surfaces in turbulent pipe flow." *Physics of Fluids* 30, no. 2 (2018): 025102.

[9] Watanabe, Keizo, Yanuar Udagawa, and Hiroshi Udagawa. "Drag reduction of Newtonian fluid in a circular pipe with a highly water-repellent wall." *Journal of Fluid Mechanics* 381 (1999): 225-238.

[10] Yanuar, Ir, M. Eng, Marcus Alberth Talahatu, Sealtial Mau, Kurniawan Teguh Waskito, and Winda Wulandari. "The Effect of Calcium Carbonate (CaCO3) Nanoparticles on the Flow through a Pentagon Spiral Pipe." *Mechanical Engineering* 8, no. 7 (2017).

[11] Yanuar, Waskito KT, Budiarso Gunawan, André Luiz Salvat Moscato, Santiago del Rio Oliveira, H. Z. Hassan, Marcos Galante Boato, Ezio Castejon Garcia *et al.,* "Drag Reduction and Velocity Profiles Distribution of Crude Oil Flow in Spiral Pipes." *International Review of Mechanical Engineering* 9, no. 1 (2015): 1-10.

[12] Montazerin, Nader, Ghasem Akbari, and Mostafa Mahmoodi. *Developments in Turbomachinery Flow: Forward Curved Centrifugal Fans*. Woodhead Publishing, 2015.

[13] Fokeer, S., I. S. Lowndes, and D. M. Hargreaves. "Numerical modelling of swirl flow induced by a three-lobed helical pipe." *Chemical Engineering and Processing: Process Intensification* 49, no. 5 (2010): 536-546.