Jurnal Polimesin
Department of Mechanical Engineering
State Polytechnic of Lhokseumawe
http://e-jurnal.pnl.ac.id/polimesin

e-ISSN	: 2549-1999	No.	: 1	Month	: February
p-ISSN	: 1693-5462	Volume	: 22	Year	: 2024

Article Processing Dates: Received on 2023-08-28, Reviewed on 2023-09-08, Revised on 2023-10-22, Accepted on 2023-11-15 and Available online on 2024-02-29

Computational Analysis Of Pipe Bend Angle Effect On Pressure Drop

Muhammad Khoirul Akbar, Anis Roihatin*, Nur Fatowil Aulia
Department of Mechanical Engineering,
Politeknik Negeri Semarang, Semarang, 50275, Indonesia
*Corresponding author: anis.roihatin@polines.ac.id

Abstract

The air conditioning system is a significant energy source inskyscrapersfor supplying cool air to all rooms. However, the process has energy losses due to the ducting used. If the problem of energy loss can be solved, the air conditioning system will bring advantages in terms of energy efficiency and financial savings. A pressure drop in air duct pipe installations, such as pipe bends, is one type of energy loss. This research intends to use Computational Fluid Dynamics (CFD) to investigate the effect of pipe bend angles and velocity relationships on pressure drop in air duct pipe installations, which has previously been validated by experimental research with a 0.17% error percentage. This study focuses on square pipe bends with varying 45º, 60º, and 90º bend angles. The research showed that when testing the highest fluid velocity of 19.68 m/s, the highest pressure drop was 275.69 Pa on the pipe bend angle of 90º, while the lowest pressure drop was 256.41 Pa on the pipe bend angle of 45º. When testing the lowest fluid velocity of 9.77 m/s the highest pressure drop was 67.73 Pa on the pipe bending angle of 90ºwhile the lowest pressure drop was 62.98 Pa on thepipe bending angle of 45º. The simulation results indicate that the larger pipe bend angle results in a higherpressure drop, and vice versa.

Keywords:
Pipe bend angle, fluid velocity, pressure drop, Computational Fluid Dynamics (CFD).

1 Introduction

The number of skyscrapers is continuously increasing, including office buildings and apartments[1]. All rooms in these buildings require a source of energy to maintain the occupants' comfort. The study about energy usage in office buildings found that the highest energy usage was attributed to the air conditioningsystem accounting for 50%, 32% for office equipment, 13% for lighting systems, and 5% for other loads such as elevators and pumps[2].

Air conditioning system is a significant energy contributor, requiring substantial power to deliver cool air to all rooms[3]. However, this air circulation process results in energy losses in the duct. If the energy loss issue can be addressed, it will provide benefits in energy efficiency and financial savings.

Several studies have been conducted to improve energy efficiency and reduce pressure drop in fluid flow through pipe bends. Experiments have been carried out to investigate the effect of various bend angles on pressure drop, and varying the bend angles leads to changes in pressure. If the bend angle is larger, it will increase the pressure drop[4], [5].The effect of Reynolds number on pressure drop was studied by experimental and three-dimensional numerical simulation. The results showed that a higher Reynolds number reducespressur drop[6]. In turbulent flow through curved pipes, the influence of pipe curvature decreases as the Reynolds number increases, as observed in experimental and numerical studies[7]. Modifications to pipe bends were made by adding an inlet disturbance body, and numerical simulation research showed that the addition of an inlet disturbance body can reduce pressure drop[8].

Pressure drops occur due to flow phenomena in the pipe, such as friction loss and separation flow, which cause secondary flows. Friction loss occurs due to the friction between the air and the wall when passing through pipe bends[9]. Secondary flows occur because of the pressure distribution difference between the outer and inner walls of the pipe bend, leading to flow blockage at the bend angle and reducing the effective area through which air can pass[8].

The distribution and flow of fluids within pipe installations are not directly observable, making analysis challenging. To overcome this difficulty, Computational Fluid Dynamics (CFD) is necessary. With this method, further research on fluid flow in piping networks can be conducted, providing simulation results that closely approximate actual conditions.

The objective of this research is to determine the influence of large pipe bend angles (45°, 60°, and 90°) on pressure drop in air duct installations using Computational Fluid Dynamics (CFD).

2 Research Methods

The effect of pipe bend angles and velocity on pressure drop in air duct pipe installations was calculated and simulated by using computational software called Ansys Fluent program. The research steps were organized as a flowchart, as shown in Fig. 1, with each step then being described.

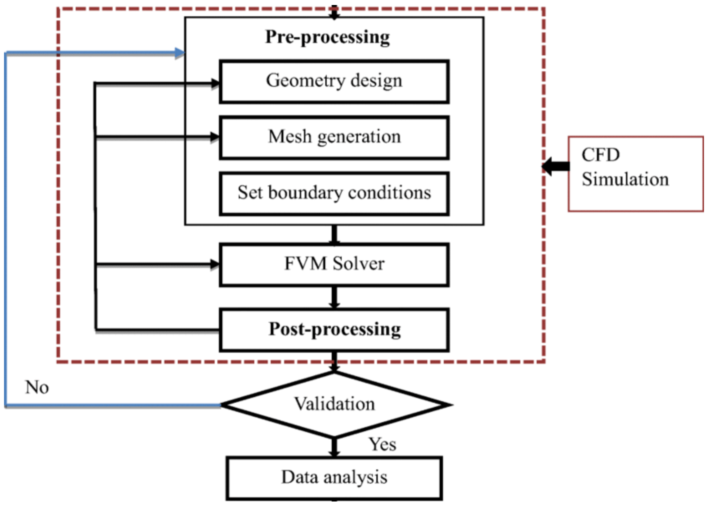

Fig. 1. Flowchart diagram of numerical modeling using Ansys Fluent.

2.1 Pre-Processing

The pipe bend was modeled using a 2D model created with AnsysSpaceclaim as shown in Fig. 2. The geometry used had D_h = 80 mm, L_{in} = 2000 mm, and L_{out} = 1600 mm. The pipe had a 90° bend with a curvature radius of R_a = 160 mm. The computational domain at the inlet was set as a velocity inlet, while at the outlet, it was set as a pressure outlet. The wall at the pipe bend was set as a no-slip wall.

Different pipe bend angles, namely 45°, 60°, and 90° werestudied in this research. The geometry of pipe bend angles is illustrated in Fig. 3.

2.2 Processing

The next step is carried out by setting up the simulation and solution. The simulation setup is configured with general settings selecting the pressure-based solver model, steady flow, velocityabsolute, and planar. The viscous model used is the k-epsilon realizable model with standard wall function.

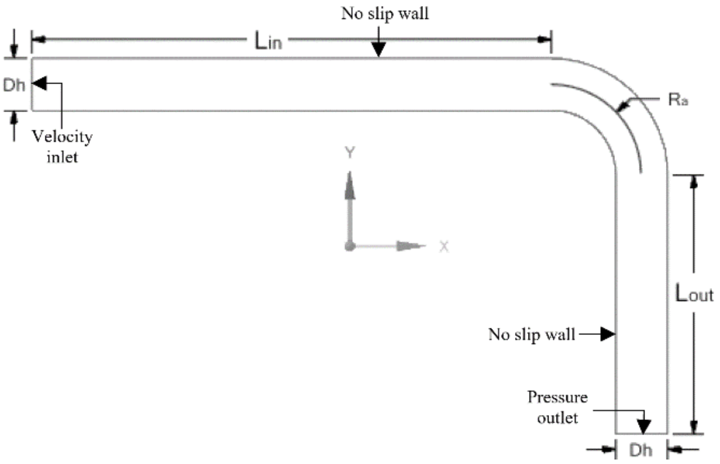

Fig. 2. Computational domain in pipe bends.

The simulation is performed using air as the fluid, with default values for density (ρ) at 1.225 kg/m^3 and viscosity at 1.7894×10^{-5} kg/ms. Boundary conditions are set for the flow passing through the test object at the inlet, outlet, and wall. The inlet is defined as a velocity inlet (m/s) with values of 9.77 m/s, 14.37 m/s, 15.85 m/s, 18.24 m/s, and 19.68 m/s (Re_{Dh} ≈ 5.35×10^4, 7.87×10^4, 8.68×10^4, 9.99×10^4, 10.78×10^4) [6]. The Reynolds number equation is utilized to determine the velocity inlet (Eq. 1).

$$Re = \frac{\rho v d}{\upsilon} \tag{1}$$

where:
ρ = density (kg/m^2)
v = velocity fluid (m/s)
d = diameter (m)
υ = kinematic viscosity (kg/ms)

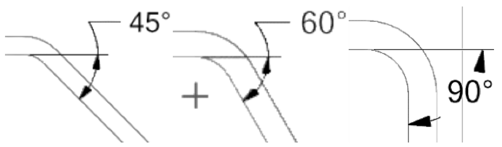

Fig. 3. Pipe bend geometry.

The turbulent specification method utilizes an intensity of 5% and a hydraulic diameter of 0.08 m. The outlet domain is defined as a pressure outlet with the same turbulent specification method as the inlet domain. As for the wall, it is set to the default no-slip condition with a constant roughness height of 0.5 and 0.00155 m.

After setting up the setup simulation, proceed with the solution simulation. A simple method is employed. The convergence criteria are set to 10^{-12}for continuity, x-velocity, y-velocity, z-velocity, k, and epsilon. Initially, the iterations start from the inlet side to achieve convergence. The iterations will reach convergence once the residual values reach 10-12.The meshing used is a quadrilateral mesh.

2.3 Post-Processing

The post-processing stage involves analyzing the results after the iterations. Results consist of both quantitative and qualitative data. The quantitative data includes the values of pressure drop (ΔP). The qualitative data includes the visualization of velocity profiles at each cross-section and the display of pressure and turbulence contour plots. Pressure drop values are determined based on the derivative of the Bernoulli equation for the specific case (Eq. 2).

$$H_L = \Delta P = P_1 - P_2 \tag{2}$$

where:
H_L = head loss
ΔP = pressure drop
P_1 = pressure inlet (Pa)
P_2 = pressure outlet (Pa).

3 Results and Discussion

3.1 Research Validation

The simulation of pipe bend was solved using the k-epsilon realizable turbulence model. This model is widely used because of its applicability to complex flows such as separation and secondary flows due to vortex formation[10], [11]. Validation of research was performed by conducting meshing tests in five different trials. Each meshing test was conducted with varying numbers of elements and mesh nodes. The selection of the mesh numbers was based on the validity of the compared data. The number of mesh used in each meshing test can be seen in Table 1.

Table 1. Number of mesh in the meshing test

Number of mesh	Element	Nodes
Mesh A	10307	11002
Mesh B	13846	14674
Mesh C	19374	20354
Mesh D	25419	26535
Mesh E	26535	36039

The pressure drop values of the pipe bend model with a bend angle of 90° were validated by comparing the results with the study conducted by Rup[6]. The validation results can be seen in the Fig. 4.

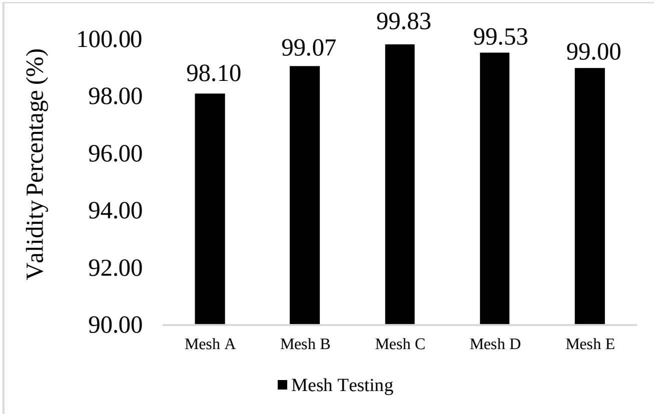

Fig. 4. Validity percentage diagram.

The mesh testing on mesh C yielded an average validity percentage of 99.83% or an average error percentage of 0.17%. As a result, the number of elements and mesh nodes used in the simulation is determined by the validation data, specifically mesh C with 19374 elements and 20354 nodes.

3.2 The Comparison of Velocity Profiles

Fig. 5 shows the velocity profile that occurs when incoming fluid flow enters the pipe and the velocity distribution between the near-wall and far-wall sides is approximately equal. This occurs because there is no disruption at this time. The fluid flow has not yet fully evolved in this section. The difference in velocity profile distribution between simulation and experiment is 2.14%. So, the velocity profile in section 1 is considered valid, because the error percentageis less than 10%.

Fig. 6 represents the velocity profile distribution of section 2. In this section, both the simulation and the experiment [6] show that the fluid flow is in the process of forming a fully developed flow. This occurs due to the friction between the pipe wall and the fluid flow, causing a difference in velocity profile between the fluid flow near the pipe wall and far from the pipe wall.

Thevelocity profile in section 2 has been considered valid because the error percentage is only 6.7%.

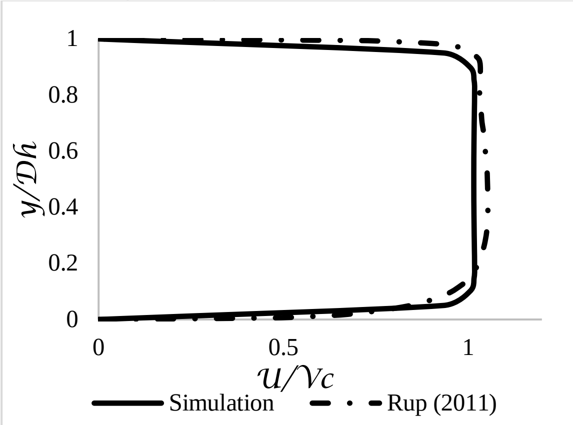

Fig. 5. The velocity profile of the simulation and Rup (2011) study in section 1, x/Dh = -10.0.

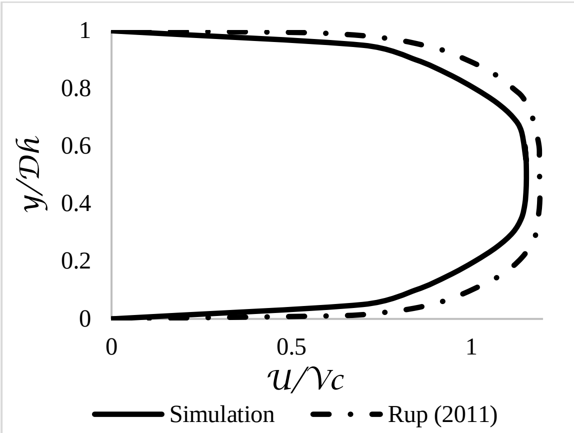

Fig. 6. The velocity profile of the simulation and Rup (2011) study in section 2, x/Dh = 1.0.

After the fully developed flow process occurs, the fluid flow moves toward the pipe bend as shown in Fig. 7. It can be observed from both the simulation and the experiment[6] that the fluid movement tends toward the side inner wall, which is caused by the adverse gradient pressure. In section 3, the velocity profile is considered valid because the error percentage is only 5.17%.

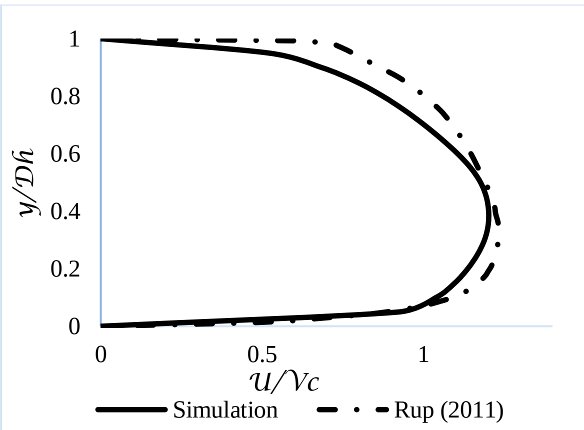

Fig. 7. The velocity profile of the simulation and Rup (2011) study in section 3, x/Dh = 14.

3.3 The Effect ofPipe Bend Angle on Pressure Drop

Fig. 8 shows the relationship between pipe bend angle and pressure drop based on Ansys Fluent simulation results. It can be observed that larger pipe bend angle comeswith a higher pressure drop, and vice versa. Table 2 shows the highest value of pressure drop at 275.69 Pa obtained at a pipe bend angle of 90° with a fluid velocity of 19.68 m/s. Furthermore, the lowest value of pressure drop at 62.98 Pa was obtained at a pipe bend angle of 45 degrees with a velocity of 9.77 m/s.

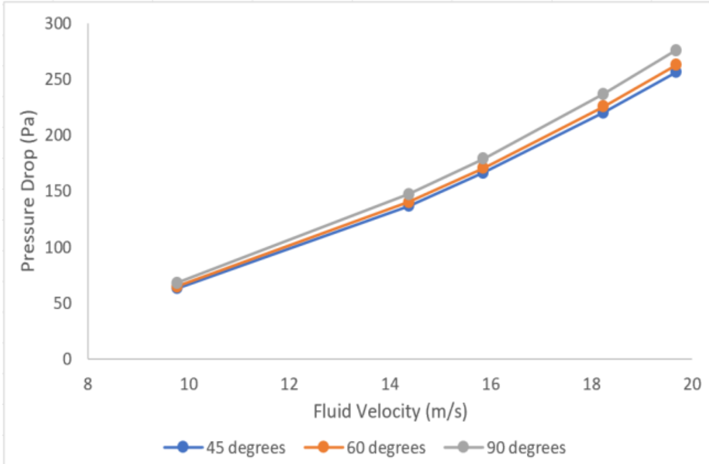

Fig. 8. The graph pressure drop in pipe bends.

Table 2. Values of pressure drop for various fluid velocities and pipe bend angles using Ansys Fluent simulation

V (m/s)	ΔP numeric (Pa)		
	45°	60°	90°
9.77	62.98	64.50	67.73
14.37	136.80	140.09	147.09
15.85	166.39	170.39	178.90
18.24	220.28	225.58	236.85
19.68	256.41	262.57	275.69

At pipe bends, there is a change in the direction of fluid flow. When fluid undergoes a sudden change in direction, there is an increase in pressureon theouter wall[12], as shown in Fig.9 – Fig.11.

This phenomenon is caused by the collision of the fluid flow with the pipe wall during the sudden change in direction. The interaction between the flow and the pipe bend wall results in a decrease in flow momentum, which leads to the inability to counteractthe adverse pressure gradient. This reduces the flow velocity and forms vortices around the pipe bend wall[13]. These vortices cause a reduction in the cross-sectional area of the main flow, resulting in flow acceleration and pressure drop.

Fig. 9 shows the contours of pressure for pipe bend angles 45 ° was degradedfrom blue (the inner wall) to green and yellow (the outer wall). At pipe bend angles 60 dan 90 as shown in Fig. 10 and Fig. 11, show the same phenomenon, but the yellow color gets wider. The larger the pipe bend angle comes the lower the flow momentum and higherpressuredrop[14].

Furthermore, fluid flow tends to generate more significant turbulence, resulting in a greater pressure drop. The larger the pipe bend angle, the greater the turbulence, which can affect an increased pressure drop.

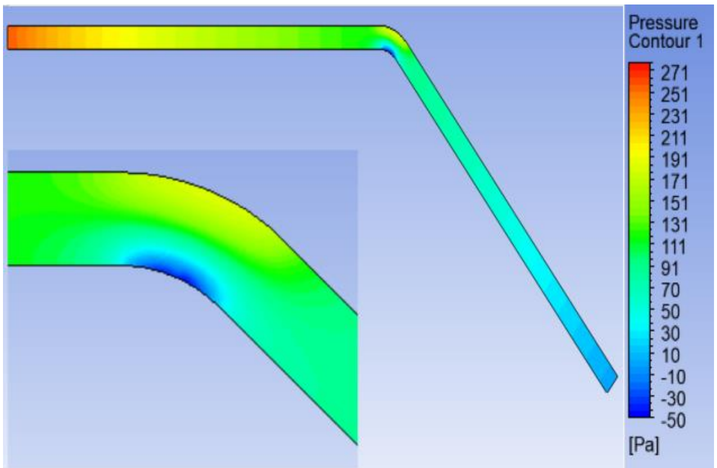

Fig. 9. Contours of pressure for various pipe bend angles 45°.

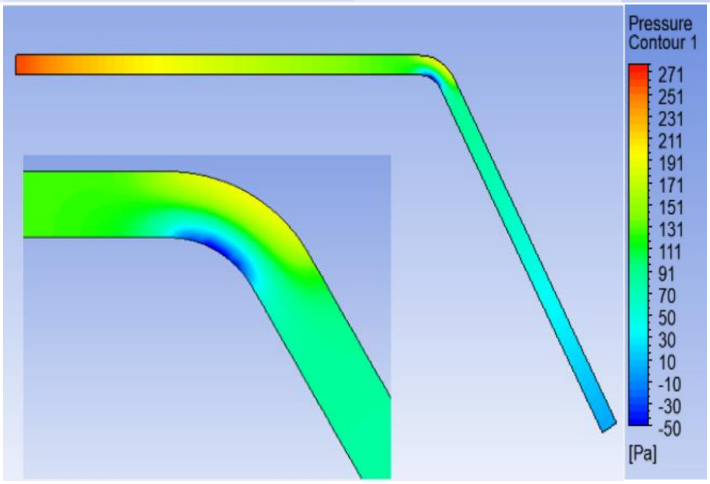

Fig. 10. Contours of pressure for various pipe bend angles60°.

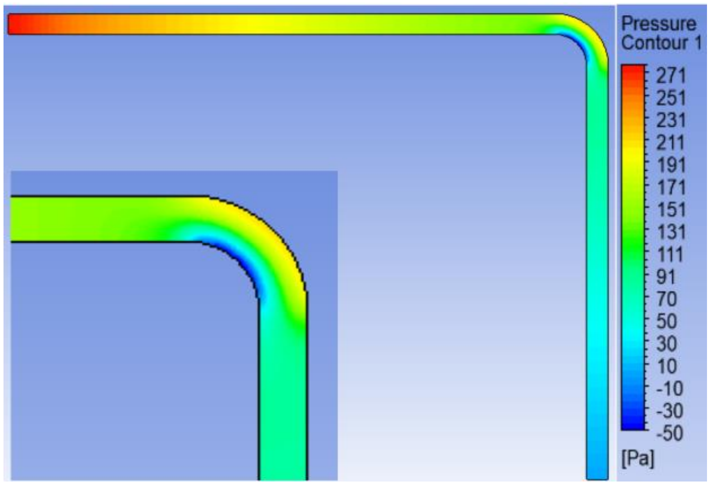

Fig. 11. Contours of pressure for various pipebend angles 90°.

The turbulence contours can be seen in Fig.12, where the red contours represent the areas with the highest turbulence. The pipe bend with a 90° angle exhibits the most intense red contours at the bend angle compared to the 60° and 45° pipe bends. This occurs because the moving fluid collides with the pipe bend wall, leading to turbulence.

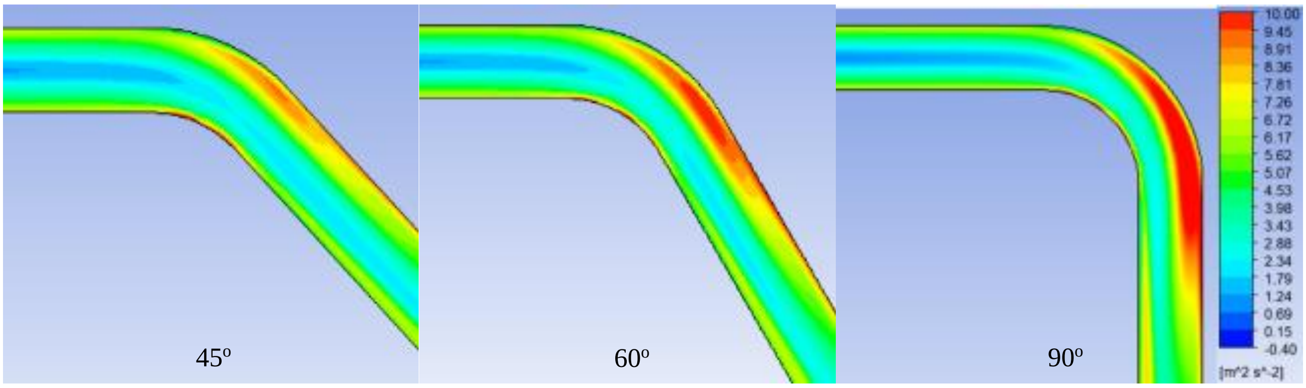

Fig. 12. Contours of kinetic turbulence for pipe bend angle variations.

3.4 The Influence of Velocity on Pressure Drop

The relationship between fluid flow velocity and pressure drop be observed in Fig.8. The graph shows a linear increase in pressure drop with increasing velocity, assuming a constant pipe bend angle. As the flow velocity increases, turbulence in the outer wall also increases, leading to increased flow separation and friction losses that reduce the flow momentum. The decreasing flow momentum results in the flow's inability to counteract adverse pressure gradients, leading to a reduction in flow velocity and the formation of vortices around the pipe bend wall. These vortices cause a decrease in the cross-sectional area of the main flow, leading to flow acceleration and a larger pressure drop.

The contours of pressure drop with varying velocities, assuming a constant pipe bend angle (90°), can be seen in Fig. 13–Fig. 17. It is observed that the highest pressure drop occurs at a fluid velocity of 19.68 m/s, as this velocity represents the highest among the tested variations. As the fluid velocity increases, the fluid flow colliding with the pipe bend wall becomes stronger, resulting in increased turbulence, which in turn affects a larger pressure drop[15].

Fig. 13 shows the contour of pressure at fluid velocity9.77 m/s is almost blue indicating low pressure drop. At fluid velocity 14.37 m/s (Fig. 14), the contour of pressure shows degradation from blue to green indicating pressure drop increases. Fig. 15 also shows the same phenomenon at a fluid velocity of 15.85 m/s. At fluid velocity 18.24 m/s, the contour of pressure shows degradation from blue and green to yellow as shown in Fig. 16. Its indicated pressure drop gets higher. At the highest fluid velocity 19.68 m/s, the contour of pressure in Fig. 17 shows a wider yellow color, indicating a higher pressure drop.

The turbulence contours of various velocities can be observed in Fig. 18. The contours that become increasingly red indicate higher turbulence, while the contours that become increasingly blue indicate lower turbulence. The contours with the densest red color correspond to the highest fluid velocity variation, which is 19.68 m/s, while the contours dominated by blue color correspond to the lowest fluid velocity variation, which is 9.77 m/s. Therefore, the highest turbulence occurs at a fluid velocity of 19.68 m/s, and the lowest turbulence occurs at a fluid velocity of 9.77 m/s. The significant turbulence at the pipe bend corner is caused by the fluid flow colliding with the pipe bend wall. When the fluid flow rate is high, the turbulence formed at the pipe bend corner becomes more intense.

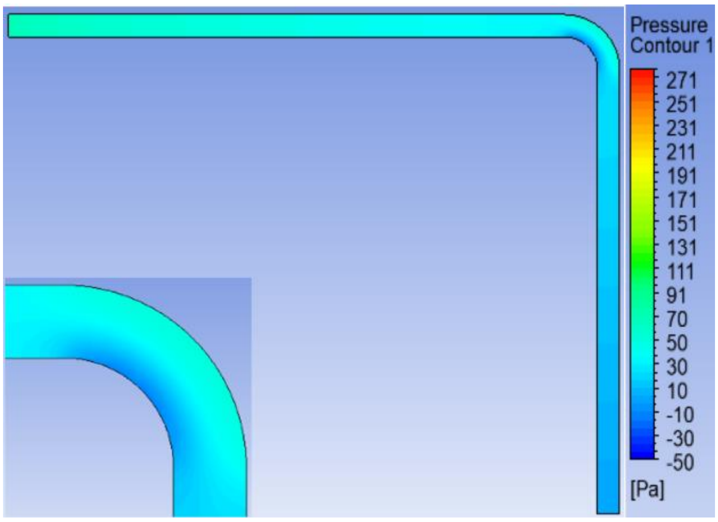

Fig. 13. Contours of pressure for various fluid velocities 9.77 m/s.

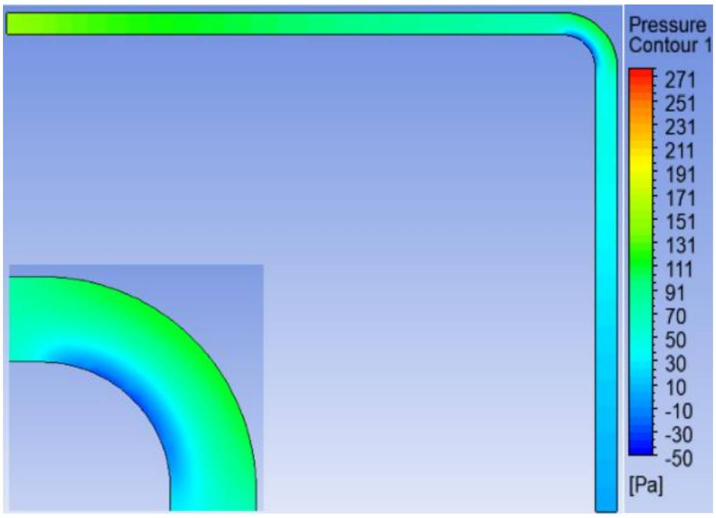

Fig. 14. Contours of pressure for various fluid velocities14.37 m/s.

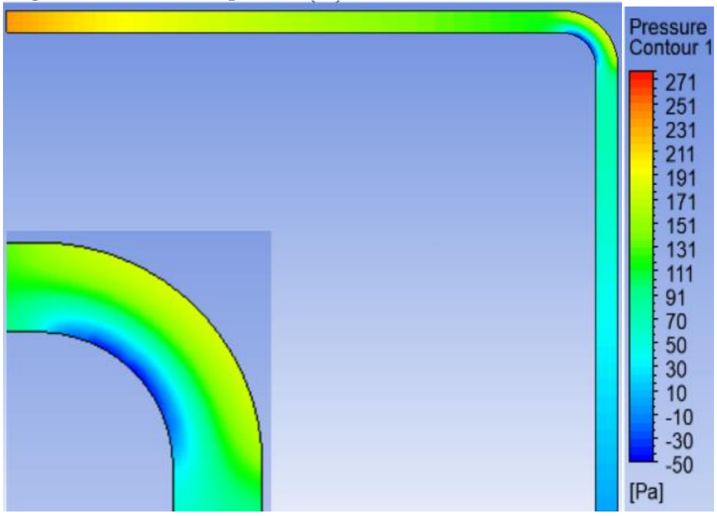

Fig. 16.Contours of pressure for various fluid velocities 18.24 m/s.

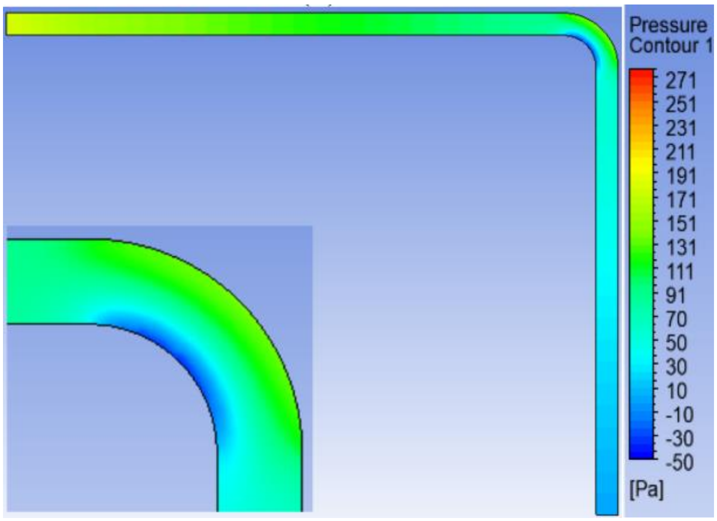

Fig. 15. Contours of pressure for various fluid velocities15.85 m/s.

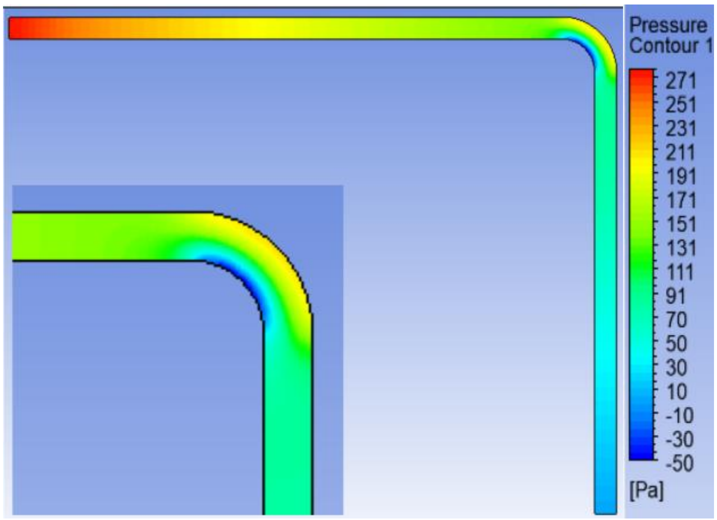

Fig. 17. Contours of pressure for various fluid velocities19.68 m/s.

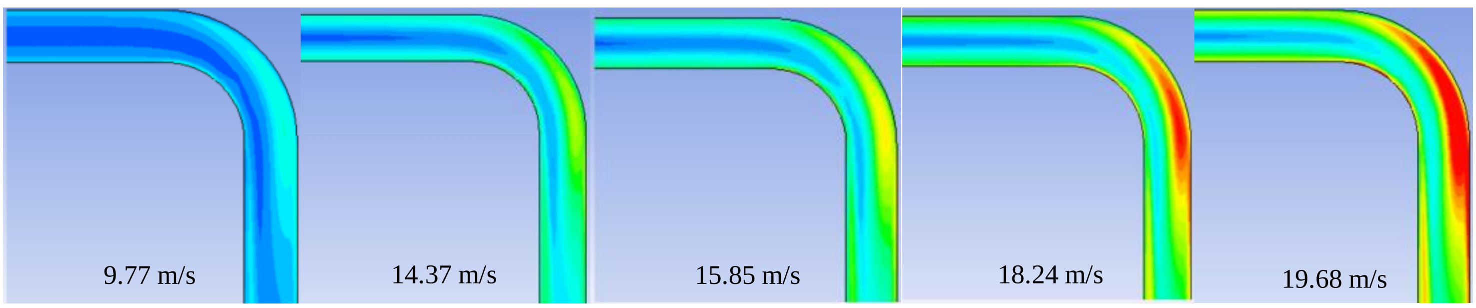

Fig. 18. Contour of kinetic turbulence forvelocity variations.

4 Conclusion

The results showed that when testing the highest fluid velocity of 19.68 m/s, the highest pressure drop was 275.69 Pa on the pipe bend angle of 90º, while the lowest pressure drop was 256.41 Pa on the pipe bend angle of 45º. When testing the lowest fluid velocity of 9.77 m/s the highest pressure drop was 67.73 Pa on the pipe bending angle of 90º while the lowest pressure drop was 62.98 Pa on thepipe bending angle of 45º.

There is a direct relationship between pressure drop and the pipe bend angle. A larger pipe bend angle comes with a higher pressure drop, and vice versa. In pipe bends, there is a change in the direction of fluid flow. When the fluid undergoes a sudden change in direction, there is an increase in pressure and turbulence, particularly on the side of the bend angle.

References

[1] T. Ahmad, A. Aibinu, and M. J. Thaheem, "The Effects of High-rise Residential Construction on Sustainability of Housing Systems," in *Procedia Engineering*, Elsevier Ltd, 2017, pp. 1695–1704.

[2] Salimi, S., &Hammad, A,."Critical review and research roadmap of office building energy management based on occupancy monitoring". Energy and Buildings, vol. 182, pp. 214-241, 2019.

[3] F. Birol, *The Future of Cooling Opportunities for energy-efficient air conditioning Together Secure Sustainable*.International Energy Agency, 2018.

[4] Nikmah, A., Amalia, R., &Satrio, D.,"Analysis of the Effect of Bend Angle Outlet Main Steam Line on the Steam Flow Characteristic" . In *IOP Conference Series: Earth and Environmental Science*, Vol. 972, No. 1, p. 012064). IOP Publishing.

[5] Gajbhiye, B. D., Kulkarni, H. A., Tiwari, S. S., &Mathpati, C. S.,"Teaching turbulent flow through pipe fittings using computational fluid dynamics approach", Engineering Reports, vol. 2(1), e12093, 2020.

[6] K. Rup, Ł. Malinowski, and P. Sarna, "Measurement of flow rate in square-sectioned Duct Bend," *Journal Of Theoretical And Applied Mechanics*, vol. 49, pp. 301–311, 2011.

[7] P. Dutta and N. Nandi, "Effect of Reynolds number and curvature ratio on single phase turbulent flow in pipe bends," *Mechanics and Mechanical Engineering*, vol. 19, no. 1, pp. 5–16, 2015.

[8] R. N. Faila and Sutardi, "StudiNumerikKarakteristikAliranFluidaMelalui Rectangular Elbow 90 derajatdengan Diamond Inlet Disturbance Body," InstitutTeknologiSepuluhNopember, Surabaya, 2018.

[9] R. Nanda Parely and A. FirdausSudarma, "Pengaruh Guide Vanes TerhadapAliranUdarapadaSalurandenganVariasiKecepatanAliranMenggunakanAnsys Fluent," *JurnalSaindanTeknik*, vol. 5, no. 1, Jan. 2023.

[10] I. W. Yudhatama and M. I. Hidayat, "Simulasi Computational Fluid Dynamics (CFD) ErosiPartikelPasirdalamAliranFluida Gas Turbulenpada Elbow PipaVertikal - Horizontal," *JurnalTeknik ITS*, vol. 7, no. 2, pp. 134–139, 2018.

[11] M. I. FadhilHendrawan, D. Danardono, and S. Hadi, "Studisimulasipenggunaan airfoil naca 6412 sebagaisudupadaturbinangincrossflowmelaluipemodelan CFD 2 dimensi," *JurnalTeknikMesin Indonesia*, vol. 13, no. 1, pp. 28–31, 2018.

[12] Nurnawaty and Sumardi, "AnalisisPerubahanTinggiTekananAkibatSudutBelokan 90 derajatdan 45 derajatdenganMenggunakan Fluid Friction Apparatus," *JurnalTeknikHidro*, vol. 13, no. 1, pp. 28–37, 2020.

[13] Du, X., Wei, A., Fang, Y., Yang, Z., Wei, D., Lin, C. H., & Jin, Z. "The effect of bend angle on pressure drop and flow behavior in a corrugated duct."ActaMechanica, vol. 231(9), pp.3755-3777, 2020

[14] M. Saifuddin and P. H. Adiwibowo, "EksperimentalKarakteristik Pressure Drop PadaSambungan T (Tee) Contraction UntukPosisiSearahDenganVariasiSudutKemiringan," *JTM*, vol. 1, pp. 74–82, 2013.

[15] I. Eka Putra, S. Sulaiman, and A. Galsha, "AnalisaRugiAliran (Head Losses) padaBelokanPipa PVC," in *PerananIpteksMenujuIndustriMasaDepan (PIMIMD-4)*, ITP Press, Jul. 2017, pp. 34–39.