Analysis of the Effect of Bend Angle Outlet Main Steam Line on the Steam Flow Characteristic

A Nikmah¹, R Amalia¹, D Satrio²*

¹Power Plant Engineering, Politeknik Elektronika Negeri Surabaya, Surabaya 60111, Indonesia
²Department of Ocean Engineering, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia

*E-mail address: dendy.satrio@its.ac.id

Abstract. Pressure drop is one of the problems that can occur due to bend components in the piping system. To minimize the pressure drop, modification of the bend pipe component was made at the bend outlet of the main steam line. In this study, an analysis of the effect of the bend angle on the pressure drop was carried out on the pressure and velocity steam distribution. The angle variations used are 30°, 45°, and 60° with the Computational Fluid Dynamics (CFD) method simulated by CFD software. The simulation results show that the lower the angle of curvature of the pipe, the lower the pressure drop and the lowest pressure drop is obtained in the 30° bend domain of 26.2 kPa. The pressure distribution pattern shows that the pressure value will increase from the inner wall to the outer wall. The greater the bend pipe angle, the greater the possibility of flow separation and pressure drop value. The velocity distribution pattern shows that the flow velocity becomes non-uniform after passing through the bend pipe. The greater the bend angle, the sharper the flow direction so that the time taken is longer. In addition, the greater the bend pipe angle, the greater the flow stagnation.

1. Introduction

Indonesia’s electricity needs are produced mainly by coal-fired power plants. Energy requirements in the scope of this system need to be taken into account to minimize the emergence of problems in each component used. Many problems occur in power plants, both in fuel processing and from the power plant parts. Trisnayanti et al. and Satrio et al. researched the fuel side, where the study aimed to determine the effect of fuel replacement on performance and fuel consumption in a steam power plant [1,2]. In contrast, Kurniawati et al. researched one of the crucial components in the power plant, the condenser. This study aims to determine the cooling water on the performance of the condenser and the efficiency of the power plant cycle, which is influenced by the temperature and flow rate of cooling water [3].

In the power generation industry, the piping system is one of the most critical components. Piping systems are used to transfer fluids (in liquid or gas) from one place to another. The most critical piping system in a power plant is the main steam line. The main steam line connects the boiler with the main turbine chamber (high-pressure turbine), where this pipe transports steam with a high temperature and pressure. There are several components in steam piping: pipes, flanges, fittings, joints, and valves. In the piping system, there are always losses caused by the parts of the piping system.
The fluid that passes through the pipe will experience losses. These losses are caused by friction between the fluid and the pipe walls and piping components. Viscous can also cause shear stress when moving, so this shear stress will change some energy of the flow. Into other forms of energy such as heat, sound, and others. This change in the form of energy causes the loss of energy on the system. In the main steam line, energy losses often occur, one of which is a pressure drop. In the steam piping system, the pressure drop is a contribution from fluid height (hydrostatic pressure differences), fluid velocity (dynamic pressure differences), pressure in the pipe, and pressure loss from other pipe components. For fluids with gas type, the head loss can be expressed in the form of pressure, known as pressure loss.

Fadhli and Madjid experimentally studied pipe bends (elbows) variations on fluid flow velocity and pressure losses. The author states that pressure changes occur in the pipe area, which has a significant bend curvature. The stagnation causes it in that position, so the existing velocity head is converted into a pressure head. On the pressure head, it is found that the greater the degree of curvature of the turn, the greater the pressure head (∆h) that occurs \[4\].

Bayu Septian has conducted numerical simulation methods on the flow profile and pressure drop at pipe bend angles to analyze the significant difference in pressure drop. The fluid that passes through the bend or elbow always experiences energy loss due to fluid flow separation in the pipe. Bayu conducted research using numerical simulation methods on flow profiles and flow pressure drops at pipe bend angles to analyze significant differences in pressure drop. Bayu stated that the bigger the pipe bend angle, the higher the pressure drop \[5\].

Zainudin et al. experimental studied the effect of the angle of the bend connection on head losses and pressure drop on pipe flow. The authors state that using the variation of the turning angle will affect the head losses and pressure drop. The greater the turning angle used, the greater the value of head losses and pressure drop. Calculating the test data is obtained if the significant the turning angle used, the velocity will be inversely proportional to the head losses \[6\].

Lupi et al. performed a numerical simulation using Direct Numerical Simulation to investigate the flow stability properties in a 90°-bend pipe. It is because the flow stability in the bending pipe (bend) also needs to be considered. The authors perform a numerical simulation using Direct Numerical Simulation (DNS) by varying the Reynolds number, which runs between 2000-3000. The study results showed that for Reb 2500, the flow was stable with two main pairs of symmetrical, opposite rotating vortices in the pipe section downstream of the bend. Two recirculation areas were detected inside the bend: one on the outer wall and the other on the inner side. For Reb 2500, the flow shows periodic and oscillatory behaviour. It is concluded that the instability is related to the strong shear by the backflow phenomenon \[7\].

Dutta et al. performed numerical on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling, aiming to determine the characteristics of flow separation with high Reynolds number in the bend of the pipe. From the simulation results, the authors state that with increasing Reynolds number, the velocity profile in the inner core of the pipe bend tries to recover its fully developed shape with deceleration and acceleration effects on the outside and inside the pipe bend, respectively. As the Reynolds number increases, the peak values become more significant. They are located further away from the wall, where the overall velocity fluctuation becomes more significant in the separation region near the core of the bend. The flow becomes very complex, unstable, and coherent downstream of the bend because the flow separation exhibits three flow motions \[8\].

Based on the explanation above, this study analyzes the effect of the bend angle of the main steam outlet on the characteristics of the steam flow using the Computational Fluid Dynamics (CFD) numerical method. The simulated reference angle is 90°. Variations in the bend angle of the main steam outlet selected are 30°, 45°, 60°. The angle variation aims to determine the effect of the bend angle on the pressure drop value that occurs at the outlet bend pipe and its effect on the distribution pattern of pressure, velocity, and temperature of superheated steam.
2. Numerical Method
This research uses computational fluid dynamics (CFD) numerical methods by CFD software to determine the effect of bend pipe angle variations on pressure drop, pressure distribution patterns and velocity pattern steam flow. Simulation is done by 3D modelling. From this research, the changes in pressure drop and the contours of pressure and velocity distribution of all variations will be taken and then compared to the phenomena that happened.

2.1. Geometry modelling
SpaceClaim draws geometry domains modelling with size details shown in Table 1. Each of the domains is shown in Figure 1. The mesh used is a global mesh of hexahedral type, with several other mesh settings such as method, inflation, and face meshing. The results of the meshing process are shown in Figure 2.

| Description                  | Value    |
|------------------------------|----------|
| Inner diameter (D_i)         | 224.6 mm |
| Horizontal length (HL)       | 1796.6 mm|
| Radius inner bend (R)        | 1063 mm  |
| Reference angle bend (θ)     | 90°      |
| Vertical length (VL)         | 420 mm   |

Table 1: Reference geometry data.

Figure 1. Geometry domains modelling.
Figure 2. Mesh structure.

The grid independence test needs to be carried out so that the parameter results are not affected by changes in the number of elements. The grid independence test also aims to find the best mesh results. The data taken to perform the grid independence test is the pressure coefficient \( (C_p) \) value on the inner wall, where the pressure coefficient \( (C_p) \) is a dimensionless number that represents the increase in static pressure to the dynamic pressure of the inlet bend [9].

Pressure coefficient \( (C_p) \) formula [10]

\[
C_p = \frac{P - P_\infty}{\frac{1}{2} \rho_\infty V_\infty^2}
\]  

(1)

Where:
- \( P \): static pressure at the point at which pressure coefficient is being evaluated
- \( P_\infty \): static pressure in the freestream
- \( \rho_\infty \): freestream fluid density
- \( V_\infty \): freestream velocity of the fluid

The results of the grid independence test can be seen in Figure 3. It can be seen that a change in the number of elements in the geometry will bring a difference in the pressure coefficient \( (C_p) \) value. From the grid independence test results, it can be seen that the pressure coefficient \( (C_p) \) at the 4th, 5th, and 6th points can be said to have no change in value as the number of mesh elements increases. Meshing 4 was chosen considering the number of elements being smaller and having a pressure coefficient value close to the 5th point.

Figure 3. Grid independence test.

2.2. Simulation process

This section is to solve numerical problems in the computational domain. In a simulation, the process must do two steps at the processing stage, including setup and solution. It is necessary to set the turbulence model, material, operating conditions, boundary conditions at this stage. The input
parameters into this process are the inlet temperature of the main steam pipe, inlet pressure, outlet pressure, and the properties of superheated steam, as shown in Table 2 and Table 3 [9].

Table 2. Operation data on simulation.

| Boundary condition | Type            | Operation data                  |
|--------------------|-----------------|---------------------------------|
| Inlet              | Pressure inlet  | Pressure: 8,991,130 Pa          |
|                    |                 | Temperature: 783 K              |
| Outlet             | Pressure inlet  | Pressure: 8,759,623 Pa          |
|                    |                 | Temperature: 781.83 K           |
| Wall               | Stationary wall | Heat flux: 150.40 W/m²          |

Table 3. Steam properties.

| Properties          | Value  | Unit   |
|---------------------|--------|--------|
| Density             | 26.7   | Kg/m³  |
| Cp                  | 2,518.68 | J/kg.K |
| Thermal conductivity| 0.07542 | W/m.K  |
| Viscosity           | 2.926x10⁻⁵ | Kg/m.s |

Three-dimensional bend pipe domain Reynolds-Average Navier-Stokes (RANS) with turbulence viscous Realizable k-ε are performed in the present study. The solution method SIMPLE scheme is chosen for pressure coupling and second-order upwind for spatial discretization. Standard initialization is used from the inlet side by entering the temperature and pressure values of 783 K, and 8765900 Pa. Residual convergence monitor used 10⁻⁶ for all parameters.

2.3. Validation data

After doing the grid independence test, the simulation data validation is carried out by reference. Figure 4 presents the comparison of Cp to x/L simulation results with reference. x/L indicates a dimensionless parameter from the location of the data collection position (the radius under consideration) from the geometry, while Cp as a dimensionless number representing the increase in static pressure to the dynamic pressure of the inlet bend. The radius under consideration is on the bend geometry’s inner wall and outer wall [9].

![Figure 4. Graph of CP against x/L inner, x/L outer.](image-url)
The value of \( C_p \) on the inner wall and outer wall reference will meet at the point \( x/L = 1.55 \) after passing the bend pipe. In the simulated \( C_p \) value, the inner wall and the outer wall also meet \( x/L = 1.7 \). From the simulation results, the minimum \( C_p \) value is at \( x/L_{in} = 1.43 \) at -0.3445, and the maximum \( C_p \) value is at \( x/L_{out} = 1.61 \), the \( C_p \) value is 0.5808. From the reference, the maximum \( C_p \) value is at \( x/L = 1.5 \) with a \( C_p \) value = 0.3814, while the minimum \( C_p \) value is at \( x/L = 0.59 \) with a \( C_p \) value = -0.3984. The difference value occurs due to the extension of the geometry on the outlet and inlet sides to prevent backflow during the simulation.

The difference in the value of \( C_p \) on the inner wall and outer wall shows a difference in the value of the static pressure of the two areas. This indicates the occurrence of a pressure gradient. One of the effects of the pressure gradient is flow separation. The pressure drop value (\( \Delta P \)) is calculated to validate the simulation results that have been carried out by reference. The pressure drop value in the reference is 31.7 kPa, while the pressure drop from the simulation results is 33.2 kPa. From the comparison of the pressure drop, an error of 4.77% is generated. Based on comparing the pressure drop (\( \Delta P \)) from the simulation with the reference, it can be said that the simulation is valid.

3. Results and Discussion

In this section, an analysis of the simulation results will be presented regarding the effect of the bend angle on the pressure drop that occurs along with the main steam pipe domain. The analysis includes the effect of the bend angle on the pressure drop that occurs in the main steam pipe, the effect of the bend angle on the pressure distribution, and the distribution of fluid flow velocity.

3.1. The effect of variations in bend angle on the pressure drop

The effect of bend pipe angle on pressure drop (\( \Delta P \)) is shown in Figure 5. The pressure drop value is obtained from the difference in static pressure at the inlet and outlet domains. From the picture, there is a change in the value of each variation.

![Figure 5. The effect of bend angle on \( \Delta P \).](image)

The \( C_p \) value in Figure 5 decreases as the bend pipe angle decreases. Values at 90°, 60°, 45°, and 30° bend angles are 33.2 kPa, 32.8 kPa, 29.4 kPa, and 26.2 kPa, respectively. The slight difference in values at the bend angle of 90° and the bend angle of 60° can be caused by the steep bend angle of the pipe so that the pressure drop that occurs is still significant. Meanwhile, the bend angle of 45° and 30° has a gentle bend angle so that the value can be suppressed enough.

The pressure drop occurs due to changes in the fluid path area and the friction effect of the fluid flowing against the pipe wall. In addition, the viscous effect also applies to energy considerations. The energy in the fluid will continue to decrease in the direction of the fluid flow, and friction occurs so that the mechanical energy will continue to decrease [11].

The simulation results that have been carried out also show the same results as the reference [6], although there are differences in the fluid used. The pressure drop (\( \Delta P \)) occurs because of the difference in the pressure values at the inlet and outlet in the fluid domain, increasing, followed by the sharper flow...
direction. In addition to the friction effect, changes in the direction of flow can affect changes in pressure and velocity of the flowing fluid. It will be explained further in the analysis of the contour distribution.

3.2. The effect of bend angle variation on the pressure distribution

The results of the pressure distribution contour in the bend pipe domain for each variation are shown in Figure 6. The contour of the pressure distribution on the bend pipe shows that from the inner wall to the outer wall, there is an increasing trend of colour contours, where each variation has different maximum and minimum pressures. However, all contours show that the pressure value is getting more significant from the inner wall to the outer wall.

![Figure 6. The contour of pressure distribution on bend pipe.](image)

Figure 6 present that the 30° bend pipe domain has a maximum and minimum pressure of 8802300 Pa and 8749260. The 45° bend pipe domain has a maximum of 8805290 Pa and a minimum pressure of 8750850 Pa. The 60° bend pipe domain has a maximum pressure value of 8803300 Pa and a minimum pressure of 8711380 Pa. The 90° bend pipe domain has a maximum pressure of 8820320 Pa and a minimum pressure of 8738170 Pa.

In the description of the pressure distribution from the inner wall to the outer wall, it can be said that the pressure distribution on the horizontal plane is in the direction of the streamline. This indicates the validity of Euler’s equation. Euler’s equation states that the flow through the bend will experience an increase in pressure in the radial direction inside the bend, proportional to the magnitude of the radius of the bend. So, it can be said that the greater the curvature (radial direction), the greater the pressure value.

Euler’s equation can only express the trend of increasing pressure, not for specific fluid pressure values, and this equation has certain limitations. The difference in pressure in the two areas also allows flow separation. The more significant the difference in static pressure, the greater the possibility of flow separation [9,11].

It can also be seen each domain has different inlet and outlet pressure values. This pressure distribution also shows that changes in the flow direction also cause changes in pressure in the fluid crossing the bend pipe. This can be seen where the fluid pressure is not uniform when crossing the bend pipe. Even after passing through the bend pipe, the fluid pressure is not uniform even though 8Di has extended geometry.

The greatest maximum pressure value occurs in the bend pipe, which has the most significant angle of curvature due to a sharp change in direction, causing the greater the possibility of flow separation, the more significant the difference in pressure of the incoming and outgoing flow, and the greater the pressure drop.
3.3. The effect of bend angle variation on the velocity distribution

Figure 7 shows the contour of the fluid velocity distribution that passes through the bend pipe. The figure shows that when the flow passes through the bend pipe, the maximum velocity is on the inner wall and switches to the outer wall when the flow has passed through the bend pipe. And each variation of the bend angle has a different maximum speed.

![Figure 7. Contour of velocity distribution on bend pipe.](image)

It is known that the maximum speed based on Figure 7 in 90°, 60°, 45°, and 30° domains is 131.351 m/s, 131.552 m/s, 132.256 m/s, and 133.859 m/s. And it's can be seen that the maximum velocity is in the inner wall area when the flow passes through the bend pipe and will shift to the outer wall area after passing the bend pipe. The overall minimum speed is the same, 0 m/s, and is near the pipe wall.

The description of this contour distribution can be said that the fluid flow velocity when passing through the bend pipe is not uniform. It is indicated an increase in the velocity value in the radial direction in the pipe, from the inner wall to the outer wall. So, velocity distribution contour can say that this phenomenon applies Euler's equation.

Where the Euler equation states, the speed will decrease with an increase in pressure and vice versa. The fluid particles experience a net pressure force; the particles will accelerate towards low pressure and slow down as they approach the high-pressure area [11]. The energy loss in the fluid passing through the bend pipe is more significant than in the straight pipe (minor losses > major losses).

Apart from the friction effect, a sudden change in direction will result in a secondary flow that can cause a pressure drop. It is also influenced by the length of the track area and the magnitude of the angle of curvature. From these results, it can be said that the greater the angle of curvature, the slower the flow velocity, so the time required for the fluid to pass is longer. The longer the time to pass, the greater the energy loss that occurs in the fluid. From the velocity distribution on the bend pipe, it has been shown that the larger the bend pipe angle, the greater the stagnation. This significantly contributes to the flow separation so that the velocity head is converted into a pressure head.

Changes in velocity due to differences in flow direction also cause changes in steam temperature. There are eight temperature measurement points in each domain shown in Figure 8. Figure 9 has shown a decrease in temperature (ΔT) in each bend pipe domain.
Figure 8. Temperature measurement point.

Figure 9. Temperature distribution on bend pipe.

The decrease in temperature (\(\Delta T\)) for each domain of bend pipe 90°, 60° is 0.00336 K, 0.00324 K, while in the bend pipe domain 45° and 30° it is 0.00315 K relatively small. The decrease in temperature occurs due to changes in shear stress caused by fluid friction on the wall surface.

Fluid flow through a curved pipe will create a velocity gradient, where the velocity gradient depends on the path distance and the coefficient of friction and surface frictional stress. The greater the trajectory distance, the greater the loss of energy due to friction.

4. Conclusions

The main conclusion from this study is:

- The bend pipe angle affects the pressure drop in each bend pipe domain. It has been proven that the pressure drop will decrease as the bend pipe bend angle is slight. The values of in the 90°, 60°, 45°, and 30° bend pipe domains are 33.2 kPa, 32.8 kPa, 29.4 kPa, and 26.2 kPa, respectively. The best results were obtained from each of these domains, namely in the domain of variation of the bend angle of 30°.

- The pressure distribution contour on the bend pipe shows a pressure change in a radial direction, and the pressure value will increase from the inner wall to the outer wall. The difference in flow direction plus the bend pipe bend angle allows for greater flow separation and increasing pressure drop.

- The velocity distribution pattern on the bend pipe shows that the flow velocity becomes non-uniform during and after passing through the bend pipe. The greater the angle of curvature, the greater the stagnation of flow, make the longer it takes for the fluid to pass and decrease in temperature.
Acknowledgments
The authors would like to convey their appreciation to the Politeknik Elektronika Negeri Surabaya and Institut Teknologi Sepuluh Nopember.

References
[1] Trisnayanthi N N A I, Pratilastiarso J and Satrio D 2020 Effects of Fuel Replacement on the Performance and Fuel Consumption of Steam Power Plant 2020 International Electronics Symposium (IES) pp 67–73
[2] Satrio D, Trisnayanthi N N A I and Pratilastiarso J 2021 Analysis of the Effects of Fuel Type Selection on the Performance and Fuel Consumption of a Steam Power Plant Int. J. Adv. Sci. Eng. Inf. Technol. 11 2046–54
[3] Kurniawati I, Pratilastiarso J and Satrio D 2020 Analysis of the Effect of Cooling Water Condenser to Power Plant Cycle Using Cycle-Tempo Software 2020 International Electronics Symposium (IES) pp 37–42
[4] Fadhli F and Madjid S 2017 Studi Eksperimental Pengaruh Variasi Belokan Pipa (Elbow) Terhadap Kecepatan Aliran Fluida Dan Kerugian Tekanan Iltek J. Teknol. 12 1717–21
[5] Septian B 2019 Simulasi Numerik Profil Aliran Dan Penurunan Tekanan Aliran Terhadap Sudut Belokan Pipa
[6] Zainudin Z, Adi Sayoga I M and Nuarsa M 2012 Analisa Pengaruh Variasi Sudut Sambungan Belokan Terhadap Head Losses Aliran Pipa Din. Tek. Mesin 2 75–83
[7] Lupi V, Canton J and Schlatter P 2020 Global stability analysis of a 90°-bend pipe flow Int. J. Heat Fluid Flow 86
[8] Dutta P, Saha S K, Nandi N and Pal N 2016 Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling Eng. Sci. Technol. an Int. J. 19 904–10
[9] Sasmitra D K 2016 Analisa Aliran Uap Panas Dari Secondary Superheater Ke High Pressure Turbine (Studi Kasus: PLTU UNIT 1 PT PJB UP GRESIK)
[10] Abbott I H and von Doenhoff A E 2012 Theory of Wing Sections: Including a Summary of Airfoil Data (Dover Publications)
[11] Fox , Pritchard, Philip J., McDonald, Alan T., R W 2011 Fox and McDonald’s introduction to fluid mechanics. (Hoboken, NJ; Chichester: John Wiley & Sons, Inc.; John Wiley [distributor])