Numerical simulation on the effect of damper opening angle in square duct 90-degree elbow

L Hakim and E T Asmorowati
Teknik Mesin, Universitas Islam Majapahit, Jl. Raya Jabon, Gayaman, Mojoanyar, Mojokerto, Jawa Timur 61364, Indonesia

E-mail: hakim@unim.ac.id

Abstract. New results in the form of velocity distribution in the square duct elbow 90-degree have been obtained from the simulation results. The flow characteristics through which are analyzed using commercial CFD software. The Reynolds averaged Navier Stokes (RANS) equation will be solved numerically using the finite difference method and using the SIMPLE algorithm. The test model to be used in this study is square duct which is installed in the inlet section 90-degre elbow with a double damper placed at a distance of $x / D_h = 2$ after outlet elbow. In this paper, the effect of the damper opening angle on the velocity profile on the 90-degree square duct is simulated in simulation at the opening angle of 30-degree clockwise direction and 30-degree counter-clockwise direction. The simulation results in the form of velocity profile distribution in each position are shown through the graph and obtained a maximum velocity magnitude value of 26.143 m/s which is exactly the place of the damper.

1. Introduction
Square air ducts with square elbows have many pairs in several office installations, expenditure centres and server rooms for air humidity safety. The air is straightened with a straight line and the elbow has a special level of difficulty. Meanwhile, the air passing through the channel has a very complex speed and pressure. The amount of air passing through the elbow has a different inertial force between the inner and outer sides of the wall, because of the different gradients between the two sides. Secondary and spatial flow causes losses to the airways that form elbows [1-4]. Therefore study of the characteristics of flow pattern in a duct elbow is of great significance in duct elbow design.

Secondary flow occurs due to the distribution of pressure on the outside and inside of the wall while the separation occurs due to the direction of the fluid through the channel so that it is able to resist the adverse pressure gradient. The flow will be separated and diverted The direction of the flow then forms a vortex. The pressure on the outside wall is bigger than the inner side, allowing fluid particles on the outer wall to move inward. The inner wall is a side that has a smaller radius than the outside wall. The movement of these particles causes the fluid flow rate to be blocked. The presence of separation and secondary flow contribute greatly to causing drooping pressure [4]. This secondary flow results a very high frictional loss in pipe bends than in straight pipes under similar conditions [5]. This secondary flow is superimposed on its primary axial flow leading to high velocity at the outer core of the pipe bend.

Research on elbow pipes and pipes has also been carried out by Dutta and Nandi [5]. The study was conducted numerically using the $k-e$ turbulence model by varying the difference in curvature ratio
\( R_c/D = 1 \) to 5. Where \( R_c \) is the mean curvature radius and \( D \) is the diameter of the pipe. The Reynolds number variation is specified, which is between \( 1 \times 10^5 \) to \( 110 \times 10^{10} \). From the simulation results obtained characteristics of flow rate and dependence on intensity of turbulence at Reynolds numbers and curvature ratio of pipe turns 90°. In general, the secondary flow in pipe bends are influenced by different parameters such as the curvature ratio, defined as the pipe diameter over bend, radius of curvature, Reynolds number, inlet flow distributions condition of the entrance flow, ie; laminar or turbulent [6].

In addition, many researchers have observed the phenomenon of flow passing through valves. The study was conducted to compare the value of valve flow coefficient \((C_v)\) and valve loss coefficient \((K)\) which passes through a single butterfly valve and double butterfly valve numerically and experimentally. Valve flow coefficient is an important feature in investigating valve performance and is determined by the difference between upstream and downstream pressure. Valve loss coefficient is a representation of differences in pressure, fluid density, and fluid velocity. In addition, from the numerical results in the velocity path line that shows that when the valve opening is 50%, the flow on the double butterfly valve has a more complex phenomenon of flow in the form of recirculating eddies [7]. Another experiment was carried out by varying the Reynolds number \( 5.0 \times 10^4 < Re < 10^6 \). From the velocity profile generated shows that the straight pipe contained in the valve flow will reach fully developed earlier compared to the straight pipe added by elbow, this is because in the elbow a secondary flow has occurred before going through the valve [8]. From literature studies, It was observed that investigating secondary movements of turbulent flow in pipe bends was easier to learn numerically.

In contrast to previous research, this study focuses on observing the phenomenon of flow in the channel-shaped airways 90° which has been added to double dampers after simulation. Silencer used to control airflow in heating, ventilation and air conditioning systems can be adaptively adapted to measure air velocity with a unique synergistic moment. In this case, double dampers varying at the opening angle 30° clockwise (CW) and 30° counter clockwise (CCW). Flow velocity through square duct 90° elbow and double dampers are the focus of the analysis in this study.

2. Method

The study was conducted in a simulation using commercial CFD software. The geometry of the model used in this work is shown in Figure 1. Geometry duct that has been equipped with double dampers, with opening angle 30° clockwise (CW) and 30° counter clock wise (CCW). This paper assumes that the fluid is incompressible water. The upstream straight channel \( L_i = 750 \) mm, downstream straight channel \( L_o = 2125 \) mm, Elbow inner radius \( R_i = 125 \) mm, Elbow Outer radius \( R_o = 250 \) m.

![Figure 1. Geometry duct with double damper and 90-degree elbow](image-url)
Three types of boundary conditions has been specified for the computational domain. At the inlet, the measured inlet velocity, At the wall boundaries, the non-slip conditions has been applied. At the outlet, the out flow. A summary of the physics for the flow simulation is shown in Table 1.

### Table 1. Summary of the Model Physic for simulation.

| Type               | hexahedral |
|--------------------|------------|
| Boundary Condition | Inlet: velocity inlet, Outlet: velocity outlate, Upstream, downstream, dan elbow: wall |
| Solver             | Pressure Based |
| Time               | Steady |
| Formulation        | Implicit |
| Space              | 3D |
| Material           | Air |
| Solution Pressure  | Second Order |
| Solution Momentum  | Second Order Upwind |
| Solution Turbulent Kinetic | Second Order Upwind |
| Energy             | |
| Solution Dissipation Rate | |
| Pressure-Velocity Coupling | SIMPLE |
| Initialize         | Inlet |
| Residual           | 1e-6 |

The Reynolds averaged Navier Stokes (RANS) equation will be solved numerically using the finite difference method and using the SIMPLE algorithm. This algorithm is effectively used in solving complex computational fluid dynamics equations [9]. To solve this equation, turbulence model selection is needed [10]. The selection of the right turbulence model is needed to solve 3D turbulent flow problems because it requires accurate modeling [5]. The governing equations for incompressible fluid flow with constant properties are conservation of mass and momentum [5, 11-14].

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho u) = 0 \quad (1)$$

$$\rho \left( \frac{\partial u}{\partial t} + u \cdot \nabla u \right) = -\nabla p + \mu \nabla^2 u + f \quad (2)$$

The turbulence model $k - \varepsilon$ was chosen in this study as a turbulence model between single phase flow streams in square duct elbow [5, 15-17]. The transport equation for $k - \varepsilon$ (RNG) is as follows.

$$\frac{\partial (pk)}{\partial t} + \frac{\partial (pk u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \frac{\mu_k}{\sigma_k} \frac{\partial k}{\partial x_j} \right] + 2 \mu_t E_{ij} E_{ij} - \rho \varepsilon \quad (3)$$

$$\frac{\partial (p\varepsilon)}{\partial t} + \frac{\partial (p\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \frac{\mu_{\varepsilon}}{\sigma_{\varepsilon}} \frac{\partial \varepsilon}{\partial x_j} \right] + C_{1\varepsilon} \varepsilon \frac{E_{ij} E_{ij}}{k} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} \quad (4)$$

Where $u_i$ represents velocity component in corresponding direction, $E_{ij}$ represents component of rate of deformation, $\mu_t$ represents eddy viscosity. The Eq. (3) and (4) also consist of some adjustable constants, these are as follows:

$$C_\mu = 0.009 \ \sigma_k = 1.00 \ \sigma_\varepsilon = 1.30 \ C_{1\varepsilon} = 1.44 \ C_{2\varepsilon} = 1.92$$
3. Result and Discussion

The main objective of the present study is to characterize the effect of damper Opening Angle in Square Duct Elbow 90-Degree through numerical simulation. The results of the velocity vector and velocity profile are presented in this section.

3.1. Validation of velocity profile

The numerical simulation of the Navier-Stokes is performed by the following conditions, i.e., the Reynolds number of $2 \times 10^5$, variations in the damper opening angle 30-degree clockwise (CW) direction and 30-degree counter clockwise (CCW) direction. In addition, the mesh on the area near the wall is used by the boundary layer mesh, in the other zone the hexahedral mesh type is used. Double dampers are placed in the downstream after outlate 90° elbow ($x/D_h = 0$), $D_h$ is the hydraulic diameter of the duct. While the parameters compared are the average speed ($\bar{U}$) in the downstream $x/D_h = -1$. The results obtained are compared with the results of research conducted by Rup and Sarna [1] numerically and by Sudo et al. [2] is experimentally used as validation. The speed profile generated is shown in Figure 2.

![Figure 2. velocity profile at $x/D_h = -1$](image)

Figure 2. Shows a comparison of the velocity profile of the model produced with experimental and numerical results at position $x/D_h = -1$. The velocity profile for each mesh was produced to compare with the results published by previous researchers. From the results of the validation, it shows that the resulting model and the method currently used are close to the results that have been published by [1, 2], so this model and method will be used for further analysis. The ordinate of the graph in Figure 2 is a representation of the value of local velocity divided by the velocity value on the inlet side, while the abscissa in the graph represents the vertical position of the velocity profile on the channel. The value $z/D_h = -0.5$ in the abscissa in the graph shows the position of the inner elbow, while $z/D_h = 0.5$ indicates the position that is in the direction of the outer elbow. The average comparison ($U/U_{Ref}$) with some previous studies in Table 2. At the out late of the bend ($x/D_h = -1$) the flow has detached from the inner wall, and the separated flow region grows as the fluid moves downstream, causing flow acceleration along the outer wall as expressed by Dutta and Nandi [5].

| Table 2. The comparison of the average ($U/U_{Ref}$) of $R_e = 2 \times 10^5$ |
|---------------------------------|----------------|----------------|
| Present Study                  | Num. R. Sarna [1] | Exp. K. Sudo [2] |
| 0.705                          | 0.777           | 0.906           |
3.2. Velocity profile distribution

The simulation results show the velocity profile, speed path line, velocity vector on the cross section with the double damper installed. According to Figure 3, the uniform mean velocity on the inlet side, then shifts towards the inner elbow just when going to elbow. The flow is normalized again after outlet elbow to double damper.

![velocity profile](image)

**Figure 3.** velocity profile of damper opening angle: (a) 30 CW (b) 30 CCW

The downstream channel after outlet elbow is the focus that is observed. The inner elbow side is a channel side that has a smaller curvature radius than the outer elbow side. From Figure 3(a) and 3(b), it can be seen that, the lowest velocity occurs on the inner elbow side, on the inner wall side there is flow separation right after the outlet elbow. Inertial forces are the cause of adverse pressure gradients which will cause the flow to form a secondary stream [4, 5, 18]. The flow will reverse from the main flow (back flow), then form a vortex. The flow separation that occurs results in an effective flow area decreasing, so that it can cause pressure drop along the flow. After the outlet elbow, then the fluid flow leads to the double damper that is installed afterwards ($x/D_{h} = 0$)

**Table 3.** Maximum of velocity at damper opening angle 30 CW and 30 CCW

| $x/D_{h}$ | Damper Opening Angle | $U_{Max}$ (m/s) |
|-----------|----------------------|-----------------|
|           |                      | -2  | -1  |  0  |  1  |  2  |  3  |  4  |  5  |
| 30 CW     |                      | 12.139 | 11.824 | 26.143 | 20.373 | 18.289 | 15.559 | 14.136 | 13.902 |
| 30 CCW    |                      | 12.065 | 11.457 | 23.400 | 21.725 | 17.677 | 16.710 | 15.692 | 14.836 |
|           |                      | 6   |  7  |  8  |  9  | 10  | 11  | 12  | 13  |
| 30 CW     |                      | 13.636 | 13.094 | 13.128 | 12.955 | 12.806 | 12.535 | 12.326 | 12.202 |
| 30 CCW    |                      | 14.116 | 13.651 | 13.173 | 12.861 | 12.666 | 12.453 | 12.324 | 12.016 |

Figure 3(a) is shown that double damper at $30^\circ$ CCW. Fluid flow is divided into three parts, the inner elbow, centerline, and outer elbow sides. There is a unification of flow from the centerline and
outer elbow sides that point to the outer elbow side, so that the flow becomes faster on the outer elbow side due to the double damper installed. In contrast to the double damper installed 30° CW in Figure 3 (b). Whereas in the inner elbow side there is a secondary flow right behind the damper which results in a change in the direction of flow velocity leading to the outer elbow. The speed on the inner elbow side is lower than that of the outer side and the velocity on the channel wall is zero, according to Musa and Mukhtar [19]. Double dampers are installed after the elbow outlet has an impact on the flow rate along the channel. The maximum velocity is shown in Table 3.

3.3. Velocity vector
Visual velocity vectors in each cross section have been shown in Figure 4(a). From the visualization it is specified only in the position in some parts as shown in Figure 4(b). Secondary flow pattern can clearly be observed in this section, where the flow pattern was extremely unsteady and complex.

**Figure 4.** velocity vector of damper opening angle 30 CCW

Figure 4 shows Secondary flow on the inner side that was formed before due to the radius of the elbow curvature then split by a double damper towards the inner side. However, the secondary flow caused by the damper impacts the flow rate to the end of the channel. After passing through the channel up to $x/D_h = 11$, the flow gradually returns to its main flow, marked by a velocity vector in the form of a point, the velocity vector of that flow direction to the x-axis.

4. Conclusion
In this study, numerical work on a turbulent flow in a duct with a 90-degree elbow was performed. Visualisai aliran dapat digambarkan melalui kontur kecepatan secara melintang terhadap double damper 90°. Visualisation of fluid flow can be described through the contour velocity transversely to the double damper 90°. Through this visualization, the installation of double dampers has an effect on the flow rate to the end of the channel $x/D_h = 16$. Secondary flow occurs behind each damper. However, it is more dominant in the dampers that are located near the inner elbow side. In addition, from the simulation results the maximum speed value is 26.143 m/s, which is at position $x/D_h = 0$ (right at the damper position) for the opening angle 30° CW and 23. 400 m/s at the opening angle 30° CCW.
5. Acknowledgments
We would be great appreciation to the DRPM DIKTI for giving us a mono year research grant (PDP). With contract numbers between researchers and LP4MP UNIM Number: 01 / LP4MP / UNIM / K / IV / 2019. And give us to disseminate of our research output to this paper.

6. References
[1] Rup K and Sarna P 2011 Analysis of turbulent flow through a square-sectioned duct with installed 90-degree elbow Flow Meas. Instrum. 22 383.
[2] Sudo K, Sumida M and Hibara H 2001 Experimental investigation on turbulent flow through a circular-sectioned 180° bend Exp. Fluids 28 51.
[3] Sutardi S, Wawan A W, Affan I, Iswati and Surisdno M D 2010 Experimental study of the effect of guide vane insertion and reynolds numbers on the flow pressure drop in a 90° rectangular elbow Reg. Conf. Mech. Aerosp. Technol.
[4] Miller D S 1990 A Guide to Losses in Pipe and Duct Systems p. 390.
[5] Dutta P and Nandi N 2015 Effect of Reynolds number and curvature ratio on single phase turbulent flow in pipe bends Mech. Mech. Eng. 19 5.
[6] Lai Y G, So R M C, Zhang H S and Hussaini M Y 1991 Theoretical and Computational Fluid Dynamics Turbulence-Driven Secondary Flows in a Curved Pipe 1 Theor. Comput. Fluid Dyn. 3 163.
[7] Jeon S Y, Yoon J Y and Shin M S 2010 Flow characteristics and performance evaluation of butterfly valves using numerical analysis IOP Conf. Ser. Earth Environ. Sci. 12 012099.
[8] Danbon F and Sollicie C 2002 Aerodynamic Torque of a Butterfly Valve—Influence of an Elbow on the Time-Mean and Instantaneous Aerodynamic Torque J. Fludics Eng. 122 337.
[9] Matyka M 2004 Solution to two-dimensional Incompressible Navier-Stokes Equations with SIMPLE, SIMPLER and Vorticity-Stream Function Approaches. Driven-Lid Cavity Problem: Solution and Visualization Online 39 13.
[10] Wagner C SA NE M SC PL O E – C EO AP LS TE S PL O E –”, vol. II.
[11] Herbst I and Skibsted E 2011 Analyticity estimates for the Navier-Stokes equations Adv. Math. (N. Y). 228 1990.
[12] Zhang R, Shan X and Chen H 2006 Efficient kinetic method for fluid simulation beyond the Navier-Stokes equation Phys. Rev. E - Stat. Nonlinear, Soft Matter Phys. 74 1.
[13] Bhattacharyya S, Minwalla S and Wadia S R 2009 The incompressible non-relativistic Navier-Stokes equation from gravity J. High Energy Phys. 2009.
[14] Rieschel E and Brandt L 2016 A Study on the Flow of Viscous Fluids in a Square Duct p. 1–30.
[15] Kim J, Yadav M and Kim S 2014 Characteristics of secondary flow induced by 90-degree elbow in turbulent pipe flow Eng. Appl. Comput. Fluid Mech. 8 229.
[16] Homizer G F 2004 Computational Fluid Dynamic Simulations of Pipe Elbow Flow Sand Rep. .
[17] Rahimzadeh H, Maghsoodi R, Sarkardeh H and Tavakkol S 2012 Simulating flow over circular spillways by using different turbulence models Eng. Appl. Comput. Fluid Mech. 6 100.
[18] Towne C E 2019 Computation of Viscous Flow in Cwved Ducts and Comparison With.
[19] Musa M N and Mukhtar M N H 2014 Air Flow Analysis in Square Duct Bend Appl. Mech. Mater. 695 622.