ISSN 2347 - 3983
Volume 11. No.8, August 2023
International Journal of Emerging Trends in Engineering Research
Available Online at http://www.warse.org/IJETER/static/pdf/file/ijeter031182023.pdf
https://doi.org/10.30534/ijeter/2023/031182023

# Numerical Study of Fluid Mixture Characteristics at 90° Angles with Variations of Standard Wall Functions on the k-ε Turbulence Model

**Agus Feriyanto[1], Muh. Nurkoyim Kustanto[1]*, Agus Triono[1], Andi Sanata[1], Salahuddin Junus[1], Iis Rohmawati[2]**
[1]University of Jember, Indonesia, nurkoyin@unej.ac.id
[2] Meiji University, Japan

Received Date: July 01, 2023 Accepted Date: July 19, 2023 Published Date : August 07, 2023

**ABSTRACT**

A numerical study has been carried out to determine the characteristics of fluid mixtures with different temperatures at 90° angles using ANSYS Fluent Computational Fluid Dynamics (CFD). The variation of the turbulence model used is k-ε, which is standard, RNG, and Realizable with the Near Wall Treatment method of Standard Wall Functions. The simulation domain is 90° angled with 2 perpendicular inputs and one output. The first step is to do a grid independence analysis with different mesh variations to get a proportional mesh. The research object is focused on section 6 with a distance of 35 cm before the flow output or located at 105 cm from the coordinate center. Validation was carried out by comparing temperature and velocity magnitude in research from S.N. Sridhara in section 6. It was found that the standard k-ε turbulence model was the best compared to the other variations. This gives a good idea of the distribution and flow behavior of the fluid which can be used for efficient elbow design.

**Key words :** Fluid mixing simulation, standard k-ε turbulence model, 90° angle, computational fluid dynamics (CFD)

## 1. INTRODUCTION

Mixing elbow is a device used in piping systems to transfer and mix fluids of different temperatures whether in a single phase or not. This type of mixing has many applications in industries such as polymers, food processing and even warm water needs. The mixing geometry usually consists of a larger diameter 90° elbow with smaller tubing attached tangentially [1][2]. Mixing characteristics are usually studied by means of velocity and temperature contours in the flow plane inside a mixing elbow. Both computation and experimentation are used to predict and observe fluid flow patterns and temperature profiles across flow paths [3]. In the process of mixing fluids with different temperatures flow into the mixing chamber per second, this indicates that a comparison of mass flow rates is needed in this case [4][5].

In this case the temperature changes that occur in the mixing of fluid flows with different temperatures using the k-ε turbulence model (Standard, RNG, Realizable) with the Near Wall Treatment standard wall function method are used to study the behavior of mixing fluid flows in section 6 with a distance of 35 cm is before the flow output or is located at 105 cm from the coordinate center [1]. This simulation also compares the mesh quality of the three mesh variations as shown in Table 1-3 Grid Independence Analysis on Check Mesh Quality, the results are velocity and flow contour graphs [6][7][8].

CFD is used as a tool for simulating fluid flow, process evaluation and simulating component designs to solve more complicated problems, with more detail and more precise results [9]. The focus of this study is to numerically analyze fluid dynamics to determine the characteristics of heat transfer and fluid flow in a 90° elbow pipe using CFD ANSYS Fluent. The discretization equation uses a semi-implicit volume approach with the SIMPLE algorithm [10]. This study uses the k-ε turbulence model where this model is quite complete with two equations, namely turbulent velocity and length scales. This turbulence model is quite economical from a computational point of view and sufficient accuracy for various types of turbulent flow makes the k-epsilon model often used in fluid and heat transfer simulations [11][12][13].

Other researchers conducted similar research on simulations that focused on performance near-wall treatment methods in Computational Fluid Dynamics, with the turbulence models used being k-ε Standard (SKE), k-ε RNG (RNG) and k-ε realizable (RKE). . The existence of wall shear stress on the channel in the pipe greatly influences the optimization of the pipe design. Therefore a more detailed analysis is needed in the area near the wall using CFD [6][9].

However, in previous studies, modeling of the 90° elbow had not been carried out effectively by selecting the best mesh to determine the characteristics of the fluid flow so as to obtain an optimal and efficient 90° elbow design [14][15][16]. One of the advantages obtained by conducting this research is that the optimal 90° elbow design is obtained only through CFD simulations so that it is more economical and efficient. The parameters observed were velocity magnitude and temperature distribution which were validated by the research of S.N. Sridhara [1].

## 2. RESEARCH METHODS

This research was conducted numerically using CFD ANSYS Fluent. In general, the procedures performed for CFD simulation are geometric modeling, defining boundary conditions, meshing, then running the simulation and analyzing the results (post-processing). This modeling uses the Reynolds Averaged Navier Stokes (RANS) equation with the k-ε turbulence model. This modeling is economical and able to analyze wall shear stress at 90° angles [17][18][19].

In the 90° angled fluid mixing simulation, we use Fluent CFD software. This research uses a 2D solver, steady-state, with turbulent flow k-ε in the form of a near wall treatment of the standard wall functions method that already exists in the k-ε model [9].

### 2.1 Geometry of Fluid Mixing

Figure 1 shows the geometry is a simulated geometry in the form of a 90° angle with two mutually perpendicular inlets and one outlet. The center of the x-y coordinate is placed on the side of the inlet 1 with the x-axis horizontally and the y-axis vertically to the elbow. Inlet 1 is placed on the x-axis, inlet 2 is perpendicular to the x-axis with a distance of 50 cm from the coordinate center. While the outlet is 193 cm from inlet 2. The flow through the inlet is uniform with different speeds and temperatures as shown in Figure 1. Meanwhile Figure 2 shows the location of the objectivity of this study, namely in section 6 which is located at 35 cm from the outlet.

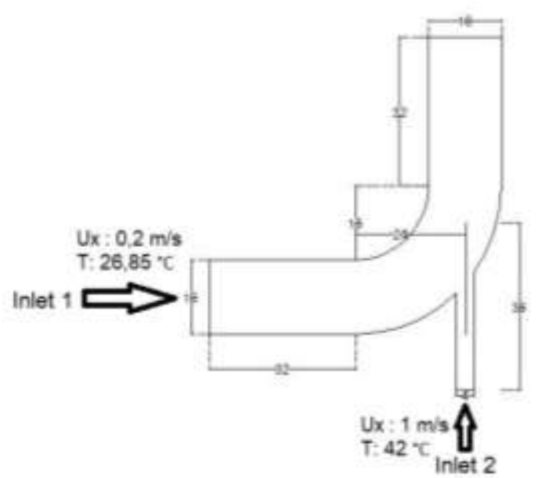



**Figure 1:** Geometry of the 90° Elbow

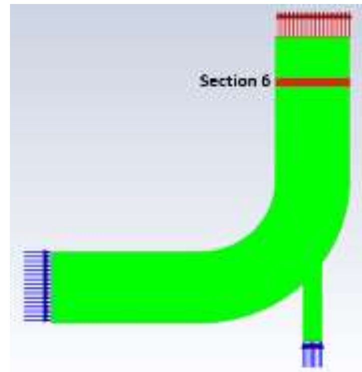
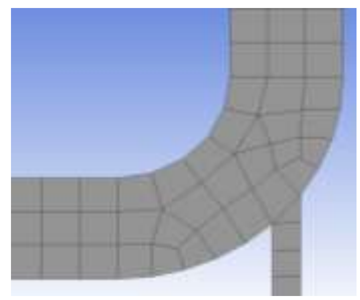


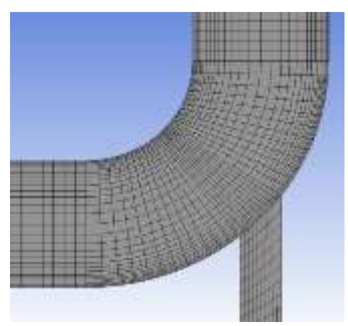

**Figure 2:** The Objectivity Focus of the 90° Elbow Research

The inlet boundary conditions used are velocity inlets. While the outlet boundary conditions used are pressure outlets. Wall boundary conditions are assumed to be no-slip conditions and stationary wall conditions. Flow is assumed to be incompressible [9][20].

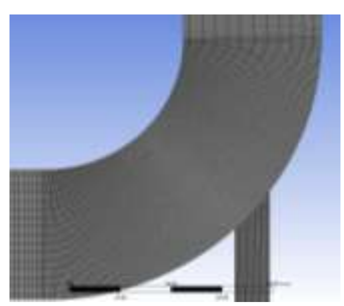
The meshing used is shown in Figure 3. The following variations of the 90° angled mesh.



(a)



(b)



(c)

**Figure 3:** Variation of 90° Elbow Mesh namely: (a) Coarse, (b) Medium, and (c) Fine

## 3. RESULTS AND DISCUSSION

### 3.1 Grid Independence Analysis

Grid Independence Analysis is used to determine the best meshing model by considering the quality of the resulting mesh, both element quality and number of elements [6][7].

**Table 1:** Coarse mesh grid independence analysis

| Mesh Type | Number of Elements | Mesh Metrics | Skewness |
|---|---|---|---|
| Coarse | 78 | Min | 3.626 |
| | | Max | 0.378 |
| | | Average | 9.029 |
| | | Standard Deviation | 0.125 |

**Table 2:** Medium mesh grid independence analysis

| Mesh Type | Number of Elements | Mesh Metrics | Skewness |
|---|---|---|---|
| Medium | 3890 | Min | 1.305 |
| | | Max | 0.589 |
| | | Average | 3.260 |
| | | Standard Deviation | 7.707 |

**Table 3:** Fine mesh grid independence analysis

| Mesh Type | Number of Elements | Mesh Metrics | Skewness |
|---|---|---|---|
| Fine | 24806 | Min | 1.305 |
| | | Max | 0.601 |
| | | Average | 2.430 |
| | | Standard Deviation | 8.963 |

Taking into account the quality of the mesh in the three mesh variations as in Table 1-3 above, the fine mesh has the smallest skewness value so that the fine mesh is used for further simulations with variations of the k-ε turbulence model.

### 3.2 Standard Wall Functions

standard wall function is a near wall treatment that is carried out on a 90° elbow simulation of the k-ε turbulence model with a smooth elbow mesh variation for further testing.

**Table 4:** Results of simulated 90° elbows on fine mesh

| Turbulence models k-ε | Wall Treatment Method | Convergence | y+ |
|---|---|---|---|
| **Standard** | **Standard Wall Functions** | 55 | 51.84744 |
| **RNG** | | 82 | 50.57237 |
| **Realizable** | | 138 | 50.47751 |

From table 4 above it can be concluded that y+ and convergence have the least value in the standard turbulence model. This indicates that the best simulation results occur in the geometric mesh.

Figure 4 shows a comparison of the velocity magnitude profile graphs between the turbulence models in section 6. If you look at it and make observations that the standard turbulence model has graphic results close to that of S.N. Sridhara [1], compared to the RNG and Realizable turbulence models. Section 6 is located 35 cm before the flow output or 105 cm from the coordinate center as shown in Figure 2.

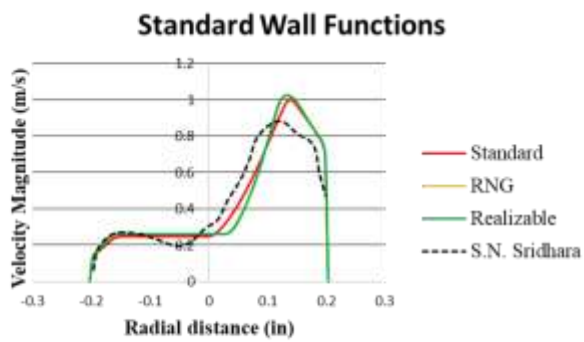



**Figure 4:** Graph of the Line Velocity Magnitude of the 90° Elbow in Section 6

Figure 5 shows a graphical comparison of the temperature profile between the turbulence model and the reference in section 6. If one looks at it and observes that the standard turbulence model has graphic results close to that of S.N. Sridhara [1], compared to the RNG and Realizable turbulence models.

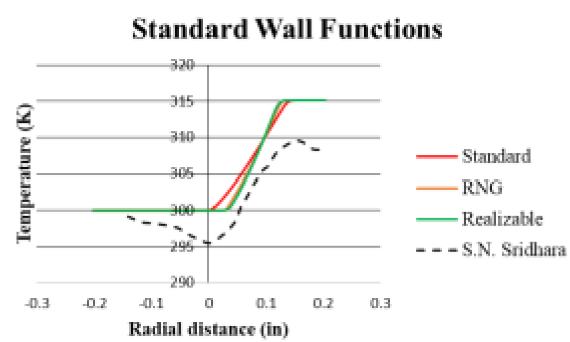



**Figure 5:** Graph of Line Temperature with a 90° Elbow in Section 6

### 3.3 Velocity Profile Contours

The velocity profile of the mixer elbow at inlet-1 speed is 0.2 m/s and 2-inlet speed is 1 m/s as shown in Figure 6. For inlet-1 speed at 0.2 m/s, the simulation is carried out using the k-turbulence model ε (Standard, RNG, Realizable) with Near Wall Treatment standard wall function method. With reference to the flow diagram, due to the venturi effect near the inlet-2, the velocity fluctuates slightly and shifts upward due to the faster movement of the fluid. This indicates the proper need of mixing hot and cold water. In the fluid velocity layer at the center of the elbow, the velocity increases to compensate for the reduced velocity of the fluid layer near the wall surface. This gives rise to a velocity gradient across the angled section.

The area closest to the wall shows a nearly linear velocity profile, [1][11][21].

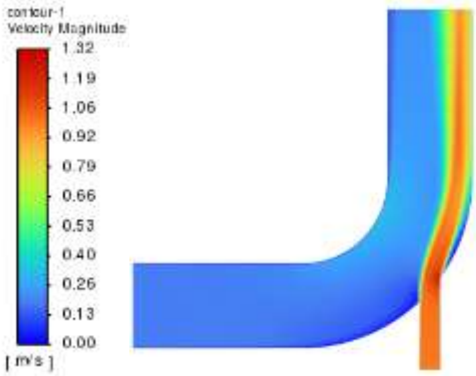



**Figure 6:** Contour Velocity Magnitude Elbow 90° with k-ε Standard

From Figure 6 above it can be seen that the fluid coming from inlet 1 and inlet 2 is mixed after going through a 90° elbow. This is clearly visible with the green contour. This shows sufficient accuracy in the standard k-ε turbulence model.

### 3.4 Temperature Profile Contours

The temperature profile is the second important profile to consider apart from the velocity profile. The temperature contour of the standard k-ε turbulence model at a 90° angle is shown in Figure 7. It can be seen that the proper mixing flow has not occurred perfectly, perhaps because the flow velocity at inlet-2 is not high so that at the upstream elbow section 6 near the outlet it is still not perfectly mixed [1][4].

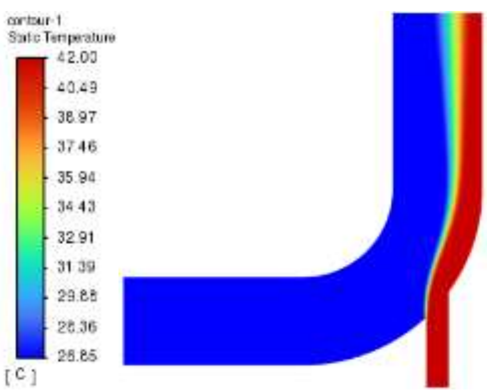



**Figure 7:** Temperature contour of 90° elbow with k-ε standard

From the visualization shown in Figure 7, the fluid starts to mix shortly before the outlet, but this mixing is not perfect.

### 3.5 Vectors Velocity Contours

In choosing the turbulence model, it greatly influences the condition of the flow velocity in the velocity vectors, with reference to the flow diagram, due to the venturi effect near inlet-2, the velocity fluctuates slightly and shifts upward due to faster fluid movement. This indicates the proper need of mixing hot and cold water. The velocity layer in the fluid at the center of the 90° elbow, the velocity increases to compensate for the reduced velocity of the fluid layer near the wall surface this gives rise to a velocity gradient across the angled section. Areas that are very close to the wall show a nearly linear velocity profile [9][21].

With the velocity vectors data, we can see that if there are two fluid supplies that have different temperatures, there will be a mixing of temperatures, where the one that was previously hotter will go down, but the one previously below will be up so that the temperature will be the average temperature [1] [4].

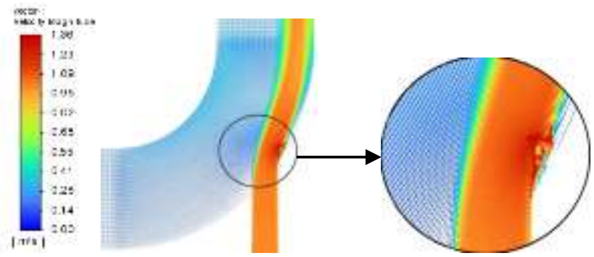


**Figure 8:** Contour velocity vectors angled 90° with k-ε standard

From the velocity vector distribution in Figure 8 above, it can be seen that the direction of the vector turns towards the outlet. This indicates that the beginning of the separation on the wall. Therefore, it is important to do further analysis to get the optimization of fluid mixing with different temperatures.

## 4. CONCLUSION

This simulation is used to determine the flow characteristics in mixing fluids with different temperatures. Investigate proper mix output with different flow rates and flow temperatures. The best mesh results are selected as a simulation model (CFD).

To find out the accuracy of this numerical method, validation has been carried out with the research of S.N. Sridhara and the simulation results are almost the same.

From the results after the simulation, the temperature profile contour and velocity magnitude that occur at the upstream elbow in section 6 before the outlet have started to mix perfectly which indicates sufficient accuracy in the standard k-ε turbulence model

## REFERENCES

[1] S. N. Sridhara, "Computational Fluid Dynamics Analysis of Heat Transfer and Fluid Flow in Mixing Elbow," *Proc. AMIET*, 2020, ISBN – 978-93-5407-803-3.

[2] M. Dhanenthiran, M. U. Samy, and G. R. Rani, "CFD Analysis of Hot and Cold Steam Flow In An Elbow," *Int. Res. J. Eng. Technol.*, 2020, ISSN: 2395-0056.

[3] A. E. Liemena, "Comparison Of Hot Water Mass Flow Rate Against Cold Water Entering The Mixing Room," *J. Tek. Ind. ARIKA*, vol. 05, No. 1, 2011, ISSN:1978-1105.

[4] A. S. Nugroho & H. Kustanto, "Computation Of Fluid Dynamics Of Water Filling In Elbow Economiser,"

2013, doi: 10.1016/j.egypro.2015.01.064.

[5] J. Zhang, H. Huang, K. Hu, and P. Zhang, "Analysis of elbow effect on the high pressure gas pipe," in *Journal of Physics: Conference Series*, 2020, no. 1. doi: 10.1088/1742-6596/1600/1/012087.

[6] R. Mahmud and I. Rohmawati, "Numerical Simulation of Backward-Facing Step Flow Using Near Wall Treatment Methods and the Viscous k-ε Model," *Semin. Nas. Teknol. Ind. Sustainable I (SENASTITAN I)*, 2021, ISSN 2775-5630.

[7] C. G. Armstrong, D. J. Robinson, R. M. McKeag, T. S. Li, S. J. Bridgett, R. J. Donaghy & C. A. McGleenan, "Medials For Meshing and More," *Queen's Univ. Belfast Belfast BT9 5AH N Irel.*, pp. 277–288, 1995, http://sog1.me.qub.ac.uk/femgroup. html.

[8] W. Versteeg, H. K., and Malalasekera, An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education Limited. 2007. ISBN: 978-0-13-127498-3.

[9] F. Tuakia, "CFD Basics Using Fluent CFD: Computational Fluid Dynamics." Bandung: Bandung Informatika, 2008. ISBN/ISSN : 978-979-1153-21-8.

[10] Q. H. Mazumder, "CFD Analysis of Single and Multiphase Flow Characteristics in Elbow," *Engineering*, vol. 04, no. 04, 2012, doi: 10.4236/eng.2012.44028.

[11] A. S. Sulistiya, S., & Kasman, "Validation of Turbulence Models in Numerical Simulation Using Fluent Software with Wing Onera M6," *J. Aero Technol.*, 2019, https://doi.org/10.29122/joat.v2i1. 3817.

[12] S. Hariyadi, "Turbulent Flow Dynamics Simulation on Flat Plate Boundary Layer 1 - 11 Using the k – ε Turbulent Model (Standard, Realizable, RNG)," *Aviat. Technol. J.*, pp. 1–11, 2017, ISSN : 2548-8090 e-ISSN : 2548-810X.

[13] R. Daneshfaraz, A. Rezazadehjoudi, and J. Abraham, "Numerical investigation on the effect of sudden contraction on flow behavior in a 90-degree bend," *KSCE J. Civ. Eng.*, vol. 22, no. 2, 2018, doi: 10.1007/s12205-017-1313-3.

[14] V. S. N. Nimbalkar, A. W. Patwardhan, I. Banerjee, G. Padmakumar, and G. Vaidyanathan, "Thermal mixing in T-junctions," *Chem. Eng. Sci.*, vol. 65, no. 22, 2010, doi: 10.1016/j.ces.2010.08.017.

[15] D. Sharp, M. Zhang, Z. Xu, J. Ryan, S. Wanke, and A. Afacan, "Mixing hot and cold water streams at a T-junction," *Chem. Eng. Educ.*, vol. 42, no. 3, 2008, ISSN : 00092479.

[16] T. A. Sedrez, S. A. Shirazia, Y. R. Rajkumar, K. Sambath, H. J. Subramani, "Experiments and CFD simulations of erosion of a 90° elbow in liquid-dominated liquid-solid and dispersed-bubble-solid flows," *elsevier journals*, pp. 1–11, Wear 426–427 (2019) 570–580, https://doi.org/10.1016/j.wear.2019.01.015.

[17] J. Kim, M. Yadav, and S. Kim, "Characteristics of secondary flow induced by 90-degree elbow in turbulent pipe flow," *Eng. Appl. Comput. Fluid Mech.*, 2014, doi: 10.1080/19942060.2014.11015509.

[18] P. Dutta and N. Nandi, "Study on pressure drop characteristics of single phase turbulent flow in pipe bend for high reynolds number," *ARPN J. Eng. Appl. Sci.*, vol. 10, no. 5, 2015. ISSN 1819-6608

[19] S. Vijiapurapu & J. Cui, "Performance of turbulence models for flows through rough pipes," *elsevier journals*, pp. 1–9, ISSN: 1458–1466, 2009, https://dx.doi.org/10. 1016/j.apm.2009.08.029.

[20] Q. H. Mazumder, K. Hassn, A. Kamble, VT Nallamothu, "Characterization of particulated flow induced erosion in elbow geometry," *springer J.*, pp. 1–8, 2020, https://doi.org/10.1007/s42757-020-0066-2.

[21] T. Lu, D. Attinger, and S. M. Liu, "Large-eddy simulations of velocity and temperature fluctuations in hot and cold fluids mixing in a tee junction with an upstream straight or elbow main pipe," *Nucl. Eng. Des.*, vol. 263, 2013, doi: 10.1016/j.nucengdes.2013. 04.002.