Teaching turbulent flow through pipe fittings using computational fluid dynamics approach

Bhavesh D. Gajbhiye1  |  Harshawardhan A. Kulkarni1  |  Shashank S. Tiwari1  |  Channamallikarjun S. Mathpati1

1Department of Chemical Engineering, Institute of Chemical Technology, Mumbai, India

Correspondence
Channamallikarjun S. Mathpati, Department of Chemical Engineering, Institute of Chemical Technology, Nathalal Parekh Marg, Matunga East, Mumbai, Maharashtra 400019, India. Email: cs.mathpati@ictmumbai.edu.in

Funding information
Department of Atomic Energy, Government of India

The largest network of fluid transportation in the world is through pipelines. During the transportation of fluids through pipes, several “fittings” are used in the piping system such as elbows, T-junctions, reducers, expanders, bends, couplings, valves, etc. The flow complexities in pipe fittings are accounted for the pressure drop in piping network design. The pressure drop is estimated using the loss coefficient or equivalent length method using standard charts. Computational fluid dynamics (CFD) is a reliable tool to estimate pressure drop and understand nonidealities in pipe fittings. The use of CFD in the advanced level course in transport phenomena/fluid flow for piping network design can help students to implement modern mathematical tools as well as evaluate standard protocols followed in the industries. In this article, an interactive teaching methodology has been implemented to investigate the hydrodynamics in various pipe fittings (elbow, bend, Tee, and reducer) by actually visualizing the flow. The three-dimensional flow visualization is used to demonstrate the nonidealities such as separation, swirling, dead zones, etc. The CFD simulations of pipe fittings provided a new learning experience to the students that would help them to predict the pressure drops in industrial piping network systems. The outcome from the students’ survey showed that the proposed CFD methodology assisted them to gain a better understanding of conventional Chemical Engineering subjects of “Transport Phenomena” and “Fluid Dynamics” in an innovative way.

KEYWORDS
computational fluid dynamics, head loss coefficient, pipe fittings, turbulence models

1 | INTRODUCTION

The pipeline network systems are the most extensive and economical mode of fluid transportation in chemical and allied industries, all around the world. All Chemical Process Industries (CPIs) involve a complex network of pipelines and pipe...
fittings. These piping networks consist of various pipe fittings such as Tee junctions, elbows, bends, reducers, expanders, valves, etc. The turbulent flow of fluids in a piping system is accompanied by both skin and form frictions, resulting in pressure or energy loss. Skin friction occurs between the pipe/pipe fitting wall and the fluid flowing inside of it. In contrast to this, form friction occurs in pipes/pipe fittings as the fluid is subjected to sudden velocity and direction changes. Precise estimation of these frictional forces is necessary for the selection of optimal pumping systems. Generally, the frictional pressure drop is represented by Equation (1)

\[ \Delta P = \frac{2f L \rho V^2}{d} \]  

Here, \( f = \frac{\tau_w}{\left( \frac{1}{2} \rho V^2 \right)} \)

Here, the friction factor \( f \) is related to the wall shear stress, which in turn depends on the degree of turbulence in the system.

In general, the flow of liquid along a pipe can be determined approximately using the Bernoulli Equation or accurately solving the equation of motion. The former represents the conservation of mechanical energy, which is either pressure, potential, or kinetic energy; and the latter accounts for the momentum exchange between the fluid elements. However, losses due to friction between the moving liquid and the walls of pipe cause the pressure within the pipe to reduce with distance. This reduction in pressure due to friction between the flowing fluid and solid wall is commonly known as head loss. The engineering designs of such piping networks are mainly done using the head loss coefficient \( K \) also known as \( K \)-factor. The head loss coefficient \( K \) in various pipe fittings is predicted using Equation (2).

\[ K = \frac{\Delta P}{\frac{1}{2} \rho V^2} \]  

Despite the considerable amount of work in pipe bends, a large scatter exists in the reported experimental values. This disagreement amongst the reported values of head loss coefficients is mainly due to ignorance of one or more parameters, which may affect the pressure drop making these values unreliable. As previously pointed out by Crawford et al, the inconsistencies in the reported experimental values may also occur due to uncontrolled flow in the bend entry and exit sections. Moreover, inadequate accuracy in the measurement of pressure drop also results in incorrect predictions of the head loss coefficient \( K \). Conducting high fidelity experiments for accurate prediction of pressure drops in pipe fittings is often tricky and expensive. However, in recent times the advent of authoritative computational sources and efficient numerical algorithms, computational fluid dynamics (CFD) has become a potential candidate for pressure drop predictions and flow visualization through piping networks.

CFD has been used as a useful tool for hydrodynamic prediction, design, scale-up, and optimization of several chemical engineering equipments. Some of the commonly used chemical engineering equipments, that have investigated using CFD so as to understand the underlying flow physics involved are: pipes and pipe bends, particulate flows, packed bed reactors, fluidized bed reactors, bubbly flows, and heat exchangers. CFD has gained a lot of importance in the previous three decades owing to the wide application and promising results, which it offers at a nominal cost. CFD mainly facilitates in prior evaluation of chemical processes via following five steps: (i) estimation of engineering design parameters, (ii) understanding the transport phenomena of chemical engineering processes under consideration, (iii) development of models and design objectives of system under consideration, (iv) design optimization and scale-up of the system under consideration, and (v) safety assessment of postulated worst-case scenario, which could be involved while the equipment is in operation. The inclusion of basic and advanced courses on CFD in the Chemical Engineering curriculum is thus felt to be of utmost importance in recent times as also reported by Adair et al.

An elaborated view of the use of various CFD models for understanding the dynamics of the flow structures and relating them with flow parameter optimization for design and scale-up of multiphase reactors has been elaborated in Joshi et al. and Mathpati et al. Riffat and Gan used CFD to predict the pressure loss coefficients of rectangular and flat-oval duct elbows for a range of aspect ratios. Lin and Ferng predicted the hydrodynamic characteristics and corrosion rates using CFD in pressurized water reactor piping systems. The results showed that flow rate, static pressure, and wall shear stress had an effect on corrosion distribution due to secondary and separating flows. Röhrig et al. studied turbulent flow through a 90° pipe elbow in a range of Reynolds numbers between 14 000 and 34 000 by CFD using wall-resolved large-eddy simulation (LES) as well as various Reynolds-averaged Navier-Stokes (RANS) models aiming at a comparative assessment to illustrate
the benefits and drawbacks of different computational approaches. Dutta et al\textsuperscript{27} analyzed single-phase turbulent flow through pipe bends using the $k$-$\varepsilon$ turbulence model to find the flow separation and reattachment characteristics under high Reynolds number. The detailed account of previous studies on pressure drop prediction in pipe and pipe networks using CFD has been presented in Table A1. These studies indicate the potential of CFD in terms of quantification of pressure drop in pipe fittings.

Previous literature shows huge applications of computational software packages in enhanced teaching and learning process of the basic engineering courses. Pieritz et al\textsuperscript{36} introduced an educational freeware CFD Studio software package designed for teaching fluid mechanics and heat transfer processes. Hung et al\textsuperscript{37} introduced the concept of electronic learning technology for training and instructing in engineering courses. The aim of their research was to combine multimedia network and CFD methods as an additional instrument to provide an effective way of teaching in thermal-hydraulic areas. Vicéns and Zamora\textsuperscript{38} reported on a CFD-based teaching method supported by Matlab programming to improve learning procedures in the subject of “Hydropower and Ocean Power” for a Masters degree in renewable energy program at the Technical University of Cartagena (Spain). Improving teaching methods and learning processes for the course of “Hydrodynamic, Resistance, and Propulsion” in the degree of naval architecture by combining information and communication technologies and CFD was reported by Gutiérrez and Zamora.\textsuperscript{39} Panagiotopoulos and Manolis\textsuperscript{40} reported a web-based training package to teach structural dynamics in civil engineering, as well as to provide an easily accessible online interactive experimentation tool that is useful for both undergraduate and graduate courses. Implementing such a course on CFD will help the students to visualize the flow physics behind the Partial Differential Equations (PDEs), which they study in a basic “Transport Phenomena” and “Fluid Mechanics” course. Besides, it will also liberate the students to think beyond Bernoulli theorem thereby developing a rational understanding of the intricate flow patterns and providing a versatile tool for pressure drop predictions in any flow network.

The present investigation involves one such Chemical Engineering course evaluation study using CFD as a tool for flow visualization and hydrodynamic studies. In the present investigation, four different geometries: (i) reducer (ii) elbow (iii) bend, and (iv) Tee junction, have been simulated. The details of the simulation methodology along with the geometrical description have been explained in the Modeling and Simulation section below.

2 | COURSE MATERIAL

The content described in this article can be taught after the students are introduced with Bernoulli theorem followed by a few numerical examples on pressure drop calculations to determine the head loss coefficient for pipes and pipe fittings. The standard value of the head loss coefficient for pipes and pipe fittings can be found in the literature. However, all of these values are only valid for cases where we do not provide sufficient entry length. Thus, the head loss coefficients reported in the literature may not be applicable under several conditions. Under those circumstances, the best possible estimation of the head loss coefficients can be obtained using CFD.

Any CFD application involves three steps (a) preprocessing, (b) iteratively solving the governing equations, and (c) postprocessing of results. The preprocessing step involves geometry generation, meshing, identification of boundary faces, boundary conditions, and initialization of flow field. The geometry generation and meshing can be done using commercial tools such as design modeler and mesher, ICEM CFD from ANSYS Workbench (Ansys, Inc., Canonsburg, PA) or open source tools such as Salome, Gmesh, snappyHexMesh, Cfmesh, etc. To have an accurate estimation of wall shear stress, it is necessary to resolve the boundary layer region using sufficient grid points. The meshed geometry is then imported into any solver such as ANSYS Fluent/CFX, OpenFOAM, etc.; and appropriate boundary conditions, fluid properties, turbulence model, and discretization methods need to be specified. In order to find the adequate grid resolution, grid independency study needs to be carried out. In the present study, the students used a moderately coarse grid comprising of 0.3 to 0.4 million cells for all the fittings. More accurate results can be obtained by increasing the grid resolution at the cost of simulation time. To explain this trade-off between the grid size and accuracy of results, the students were asked to submit an assignment using finer grids.

The tutorial cases reported below are simulated on ANSYS Fluent, a CFD software package that is widely used in CPIs. In this tutorial, the diameter was taken as $25.4$ mm whereas the length to diameter ratio ($L/D$) of 10 was maintained for each branch of T-junction, elbow, bend, and reducer for generating fully developed flow profile. For reducer, the upstream inlet diameter was 25.4 mm and the downstream diameter was $12.7$ mm. The main objective of the tutorial was to make the students aware of the following concepts of CFD: (i) Students should understand the geometry building (computer-aided design [CAD] model) operations such as creating a reference plane, generating lines and curves out of
points followed by surface creation from curves, carving a solid wall along a specified arc, splitting a solid surface or plane, etc. (ii) Meshing operations such as specifying inflation layers, learning different types of meshes such as hexahedral and tetrahedral meshes. (iii) Spatial discretization schemes, convergence criteria, parameters to be monitored in the simulation settings. (iv) Postprocessing of results to visualize streamlines and contours. The governing equations, geometrical details, and the simulation methodology have been explained in detail in the next section.

The tutorial was taken in three parts. The first part comprised of the construction of CAD model and meshing operations. This was followed by the second part which encompassed setting up the simulation parameters and running the solver. The postprocessing of simulations was then taken up as the third part of this tutorial. It is worth mentioning that it is extremely essential for the students to be well versed with the fundamental concepts of CFD and turbulence before using a black-box model such as ANSYS Fluent. To ensure this, the students were introduced to the basics of transport phenomena and turbulence from the classic books of Bird et al.\cite{41} and Pope\cite{42} in their regular “Advanced Transport Phenomena” lectures. The basic derivations of the governing equations, turbulence models, and discretization schemes were covered from the books of Versteeg and Malalasekera\cite{43} and Anderson.\cite{44}

The tutorial sessions were undertaken by assigning a distinctive problem statement to a group of two students. The tutorial sessions can also be assigned individually to the students, based on the availability of computers and licenses. Each group can be assigned with a unique set of dimensions for the pipe fittings and Reynolds number ($Re$) so that at the end of the simulations, all the students can exchange the simulation results to appreciate the effect of the geometrical modifications, boundary conditions, and $Re$ after the postprocessing.

For the quantitative estimation of pressure drop, two cross-sectional planes were created across the fitting. The difference in the surface area-weighted average static pressure predicted from these two planes was determined and the head loss coefficient was estimated using Equation 2. In addition to quantitative prediction, the nonidealities in the flow through fittings were analyzed using streamlines and contours.

### 3 | COMPUTATIONAL FLUID DYNAMIC MODELING

Four different pipe fittings were used for performing the simulations. All the simulations have been carried out under turbulent conditions ($Re = 10000$). The detailed description of these geometries along with their respective schematic diagrams is given below.

#### 3.1 | Geometrical description

##### 3.1.1 | Reducer

Flow through a reducer was considered in this study to understand the losses, which occur due to rapid changes in flow velocity mainly due to change in flow cross-section areas. There are different types of reducers such as concentric and eccentric reducers, square and tapered reducers, etc. In this case, we have considered a square reducer with an inlet diameter of 25.4 mm and an outlet diameter of 19.05 mm. Figure 1A shows the geometry of reducer used for further simulations.

##### 3.1.2 | Elbow

In case of flow through the elbow, bend radius ratio $R_c/D$ (the ratio of the radius of curvature to the inside diameter of the pipe) and flow deflection angle “$\alpha$” are important geometric parameters to be considered.\cite{45} In this work, the simulations were carried out for the elbow of configuration $R_c/D = 1$ having a flow deflection angle of 90°. Figure 1B shows the sectional view of elbow used for simulations with $R_c$ and $D$ of 25.4 mm.

##### 3.1.3 | Bend

Bend is a long radius elbow with $R_c/D = 1.5$, where $R_c$ is the radius of curvature of the bend. Figure 1C shows the sectional view of bend used for simulations with $R_c = 38.1$ mm, $D = 25.4$ mm, and flow deflection angle of 90°.
3.1.4 | Tee junction

Tee junction is one of the most common yet complex parts of any piping network. Tee junction is mainly used to accumulate (converge) flows from many pipes to a single main pipe and to distribute (diverge) the flow from the main pipe to several branching pipes. Tee-junction simulations were carried out for four different cases at $Re = 10,000$. The internal diameter of the Tee junction was 25.4 mm for all the cases. Tee junction with different flow surfaces (A, B, and C) is shown in Figure 1D. Four different inlet-outlet combinations are possible for the flow through a Tee junction. These are, symmetrical combining (where surfaces A and B serve as inlets and C is the only outlet), nonsymmetrical combining (where surfaces B and C serve as inlets and A is the only outlet), symmetrical dividing (where surface C is the only inlet while A and B are the two outlet), and nonsymmetrical dividing (surface B is the only inlet whereas A and C are the outlet).

3.2 | Governing equations

Fundamentals of turbulent flows and modelling equations related to continuity, momentum, and Reynolds Stress Model (RSM) have been explained in great details in Section B1.

3.3 | Simulation methodology

All the simulations in this tutorial have been performed in ANSYS Fluent software package, which uses a Finite Volume Methodology-based spatial discretization scheme to linearize the PDEs. To minimize the diffusive errors, it was made sure that the mesh configurations used were capable of capturing the boundary layer adequately. Also, to ensure the independency of results, a priori mesh sensitivity analysis was performed for three different mesh configurations (for each of the four geometries) based on which the selection of final mesh configuration was done. The mesh independent configurations for the four geometries are (a) reducer: 0.413 million cells, (b) elbow: 0.302 million cells, (c) bend: 0.307 million cells, AND (d) Tee junction: 0.318 million cells. For better understanding, the mesh sensitivity analysis for the reducer is shown in Figure 2. To ensure the spatial accuracy of the simulations, the axial velocity was tested over three
different mesh configurations. Hexahedral grids were used for all the three configurations; and an increment of around
twice the initial grid count was provided for each of the subsequent mesh configuration used. As shown in Figure 2, the
mesh configurations M1 and M2 show a 7% deviation whereas the mesh configurations M2 and M3 are very much in
agreement with each other with a small deviation of around 2% to 3%. This shows that the solution becomes mesh in-
dependent after implementing the M2 configuration, that is, 0.413 million grids. Hence, we used the M2 configuration for
the reducer simulations carried out in this work. Similar mesh sensitivity analysis was carried out for elbow, bend, and
Tee junction.

The simulations have been carried out using water as the working fluid ($\rho = 1000 \text{ kg/m}^3$ and $\mu = 0.001 \text{ kg/m s}$). The
turbulence in the system has been modeled using the RSM as specified in Section 3.2. The pressure-velocity coupling
has been carried out using the SIMPLE algorithm. The entire flow field has been initialized based on the inlet velocity. For CFD of pipe fittings,
we need to provide the boundary conditions (i) Inlet: uniform velocity profile, based on the Reynolds number $[Re = 10000
(v = 0.39 \text{ m/s})]$ and turbulence parameters based on equivalent diameter and turbulence intensity (6%). A zero gradient
pressure is prescribed on the inlet plane. (ii) Outlet: an atmospheric pressure condition, that is, a gauge pressure of 0 Pa is
prescribed on the outlet plane. The velocity and turbulence parameters are treated as fully developed (ie, zero gradient) at
the outlet. (iii) Wall: a no-slip boundary condition has been imposed on the wall. The roughness of the pipe fitting com-
ponent under consideration is assumed to be similar to the roughness of the connecting pipe. The convergence criterion
was set to 0.0001 for all the governing equations. The simulation results have been analyzed using plots and streamlines
of velocity and pressure as well as contours wherever necessary.

4 | RESULTS

All the fittings were analyzed in terms of velocity profiles, the pressure drop across fittings, and the streamlines. Analysis
of velocity streamlines plotted using CFD-Post helped in a much better understanding of the fluid flow through different
pipe fittings. Color bars along with the streamlines helps in identifying the changes in fluid velocity due to the fittings.
Streamlines show the flow patterns and changes in flow directions due to the fittings. Flow entry and exits are shown by
the arrows at the inlet and outlet boundaries. All the streamlines inside the fittings are shown by making the wall 75%
transparent in ANSYS CFD-Post.

4.1 | Reducer

According to the continuity equation, the reduction in cross-section area causes an increase in average velocity. As per
the continuity equation, we know that

$$\frac{V_2}{V_1} = \frac{A_1}{A_2} = \left(\frac{D_1}{D_2}\right)^2.$$  (3)
Accordingly, we get $V_2 = 0.693 \text{ m/s}$. Also, by the Bernoulli equation and continuity equation, static pressure in the upstream of reducer was found to be proportionately higher in comparison to the static pressure obtained in the downstream. The loss coefficient of 2.87 was obtained for the reducer by CFD using the RSM model at $Re = 10\,000$. At the reducer inlet surface “A”, pipe diameter is 25.4 mm and inlet velocity of 0.39 m/s corresponding to the $Re = 10\,000$ can be observed with the help of velocity streamlines and color bar. At the reducer outlet surface “B”, the pipe diameter is 19.05 mm and outlet velocity was found to be approximately 0.693 m/s with corresponding $Re = 13\,200$, which is in accordance with the velocity obtained by solving the continuity Equation (3). Also, for a better understanding of flow through reducer, an enlarged image of reducer along with the velocity contours at entry and exit of reducer cross-section are shown in Figure 3.

4.2 Elbow

The prediction of head loss for turbulent single-phase flow through elbow is difficult because of the flow complexities arising due to frictional and flow separation effects. Miller predicted the pressure loss coefficient of bend for different Reynolds numbers above $10^5$ and found that Reynolds number had a negligible effect on pressure loss coefficients. For flow through elbow, there is a widely scattered existing data. This scattered data of different researchers might be due to the manufacturing variations of the fittings, pressure measurement locations, the accuracy of the measurement devices, etc. Hence, CFD might turn into a very useful tool for flow and pressure drop studies in various pipe fittings.

Figure 4A shows the velocity streamlines of the elbow for $Re = 10\,000$. Surface (A) is the inlet and (B) is the outlet of the elbow. The diameter of the elbow is 25.4 mm and $R_c/D = 1$. To maintain the $Re = 10\,000$, the inlet velocity of the 0.39 m/s was given in CFD simulation at the inlet boundary condition at the surface “A”. The inner wall and outer wall of the elbow (bends) are known as intrados and extrados, respectively. Since there is no change in the cross-section area of flow through the elbow, the expected outlet velocity of 0.39 m/s was observed from the CFD results. As observed from Figure 4A, as the flow enters in the elbow curvature from the straight pipe, velocity is found to be maximum at the intrados and minimum at the extrados of the elbow. Radial evolution of velocity for elbow has been plotted on different lines at angles $0^\circ$, $30^\circ$, $45^\circ$, $60^\circ$, and $90^\circ$ as shown in Figure 4B. Dimensionless velocities ($U/U_i$) were plotted against the dimensionless radial distance from intrados to extrados of the elbow. At the entrance of elbow due to the curvature effect, a high-velocity zone was observed at the intrados of the elbow from $0^\circ$ to $45^\circ$, which gradually shifts towards the extrados of the elbow as the flow proceeds towards the downstream pipeline of the elbow. The loss coefficient at $Re = 10\,000$ obtained was 0.27. Chaotic flow patterns are observed near the intrados section of the elbow from Figure 4A. This nature can be better visualized by the velocity contours drawn at $0^\circ$, $30^\circ$, $45^\circ$, $60^\circ$, and $90^\circ$ cross-section planes as shown in Figure 4A. A high-velocity region at the intrados was observed starting from $0^\circ$ plane to $60^\circ$ plane; and a dead zone or flow separation due to the effect of the curvature of the elbow was observed at the exit of the intrados, that is, at $90^\circ$ plane. This high velocity at the entry of intrados is due to the sudden change in the direction of the flow. The high-velocity zone then shifts towards the extrados at the exit of the elbow.
4.3 Bend

Ninety-degree bend is structurally similar to 90° elbow mentioned above. The only difference is the radius of curvature of the bend is 1.5 times the pipe diameter, and in the case of the elbow, the radius of curvature is equal to the pipe diameter. To maintain the $Re = 10000$, the inlet velocity of 0.39 m/s was given in CFD simulation at the inlet boundary condition at the surface “A.” Since there is no change in the cross-section area of flow through the bend, therefore following the continuity equation, the inlet and exit velocities are found to be the same, that is, 0.39 m/s. From the velocity contours (Figure 5A, enlarged section) at 0°, 30°, 45°, 60°, and 90°, a shift of high-velocity zone from intrados to extrados due to the curvature effect of the bend can be observed. At 90° cross-section due to the sudden change in the flow direction, high-velocity flow separation from the intrados towards the extrados was observed. In the case of bend, the flow velocity near the intrados increased from 0° up to 30° and later decreased gradually to 45°, 60°, and 90°. At $Re = 10000$, the loss coefficient obtained in the case of the bend was 0.31. Radial evolution of velocity for bend has been plotted on different lines at angles 0°, 30°, 45°, 60°, and 90° as shown in Figure 5B. Dimensionless velocities ($U/U_b$) was plotted against the dimensionless radial distance from intrados to extrados of the bend. The shift of
the high-velocity zone along the bend curvature from 0° to 90° from intrados to extrados can be clearly observed from the figure.

### 4.4 | Tee junction

Velocity streamline distributions and contours at different planes for different flow cases at inlet Reynolds number of 10,000 for flow through Tee junction are shown in Figure 6 below. For case-1 where surfaces A and B are inlets and C is the outlet, symmetrical combining flow can be observed from the velocity contours. For case-2 where surfaces B and C are inlets and A is the outlet, nonsymmetrical combining flow behavior can be observed in a Tee junction. For case-3 where surface C is an inlet and A and B are outlets, symmetrical dividing flow behavior can be observed in a Tee junction. For case-4 where surface A is inlet and B and C are outlets, nonsymmetrical dividing flow behavior can be seen in a Tee junction. Velocity streamlines and contours help in visualizing all the different possible flow patterns in a Tee junction. Three different planes were created in all the three respective legs of the Tee junction near the central zone, where static pressure over that plane was determined. Using this static pressure difference, the head loss coefficient was determined between the different surfaces of the Tee junction. The loss coefficients for all possible cases of Tee junctions at inlet $Re = 10,000$ were determined using RSM and are reported in Table 1. The set of cross-sectional planes on all the three surfaces viz., A, B, and C at 25.4 mm from the center of Tee junction were plotted to get the pressure drop between them for calculating the loss coefficients.

Case-1 shows symmetrical combining fluid flow through Tee junction. As shown in Figure 6A, surfaces A and B are the inlets and surface C is the outlet. The velocities at both the inlets are 0.39 m/s and the outlet velocity as estimated from CFD was found to be 0.7824 m/s. Dead zones were observed on the periphery of the outlet branch just at the beginning of the vertical section of the Tee junction. Also, a small vortex and a dead zone formation were observed at the top of the center of the horizontal section of the Tee junction. This phenomenon might be due to the flow impingement, mixing, and sudden change in flow direction at the center of Tee junction. This can be better visualized in the magnified image of Figure 6A case-1.

Figure 6B case-2 describes the fluid flow pattern of nonsymmetrical combining flow through Tee junction. Surfaces B and C are the inlets and surface A is the outlet in this case. The fluid velocity at both the inlets, B and C is 0.39 m/s

---

**FIGURE 6** Velocity streamlines and contours (in insight) of Tee junction at $Re = 10,000$. (A) Case-1, (B) Case-2, (C) Case-3, (D) Case-4
and that obtained at the outlet A is 0.7802 m/s. A small dead zone was also observed at the bottom of the outlet branch of the case-2, just at the beginning of the combined flow. This is due to vertical upward flow through the surface C, which pushes the horizontal fluid flowing through the surface B. Due to the combining of flow from two inlets as shown in Figure 6A,B, a comparatively higher velocity zone was observed in the outlet branch of the Tee junction.

Symmetrical dividing flow through Tee junction is shown in Figure 6C case-3. At surface C the inlet velocity of 0.39 m/s is observed, whereas at the outlet surfaces A and B the velocity observed is 0.1953 m/s at both the ends. A large number of swirls were observed at both the exits near the Tee junction, which later stabilizes over the pipe length of 254 mm ($L/D = 10$).

Figure 6D case-4 shows the nonsymmetric dividing flow pattern through the Tee junction. The inlet velocity of 0.39 m/s was given at the inlet surface A. The outlet velocity at the straight horizontal branch surface B was found to be 0.3259 m/s and the outlet velocity at the branch section on the surface (C) was reported 0.0641 m/s by CFD. Very little fluid deviated through the branched section of the Tee junction. Maximum fluid moved through the straight horizontal section of the Tee junction due to the inertial forces.

5 | TEACHING AND LEARNING EVALUATION

This approach has been taught in the second semester of the Master of Chemical Engineering course on Advanced Transport Phenomena with a number of students in the group being 50 and none of which had any prior exposure to CFD modeling. The personal characteristics of the participants are summarized in Table 2.

The workshop was conducted in three sessions. Theoretical explanation and introduction to CFD and its applications from a chemical engineering perspective were conducted in one session. In the second session, the students were well
acquainted with the interface of ANSYS Design Modeler and Fluent through hands-on training and simulations have been carried out. All the participants were provided with an instruction manual, which consisted of the dimensions of the geometry to be modeled and the details of mesh to be generated. Participants were then instructed about the simulation settings (discretization schemes, boundary conditions, convergence criteria, etc.) to be used. After the simulations, participants were asked to carry out the postprocessing in the third session to estimate the loss coefficient as well as to analyze the path lines and contours. Feedback was taken at the end of the third session. Participants found that CFD helped them to understand and visualize the flow through different pipe fittings, thereby bringing clarity to the concept of frictional losses, which need to be accounted in the Bernoulli equation. The questionnaire of the feedback from participants is summarized in Table 3 and the survey result of those questionnaires is shown in Figure 7.

The students rated each of the questions from 1 to 5, 5 being the most relevant. Figure 7 shows the pie chart distribution with student's rating 1 shown by red color, 2 by green color, 3 by grey color, 4 by yellow color, and 5 by blue color to each of the survey questions. Response to question 1 (Q1 Responses) shows that 92% of students found CFD to be useful in improving their knowledge of fluid mechanics and 8% of students took a positive stance on the question. Q2 responses show that all the students found CFD as a favorable tool to avoid difficult and time-consuming experiments. It can be observed from Q3 responses, that 84% of students think assisting experiments with numerical modelling helps in a better understanding of the physics of the fluid flow and 16% of students gave affirmative feedback. From the responses to question 4 (Q4 Responses), it can be observed that although 60% of students were able to relate the use of numerical techniques for solving PDEs using fluid flow problems, there was still a considerable amount of students who did not strongly support to this comment. An in-depth theoretical explanation on the use of numerical techniques for solving PDEs using fluid flow problems can help in conceptual

| Sr. no. | Questions                                                                 |
|---------|---------------------------------------------------------------------------|
| Q1.     | CFD enhances my understanding of the fluid mechanics course.              |
| Q2.     | CFD is a favorable tool to avoid difficult and time-consuming experiments. |
| Q3.     | Do you think assisting experiments with numerical modelling help in better understanding of the physics of fluid flow? |
| Q4.     | Are you able to relate the use of numerical techniques for solving PDEs using fluid flow problems? |
| Q5.     | CFD should be incorporated as a formal component of advanced transport phenomena course. |

Abbreviations: 0, not relevant; 5, most relevant; CFD, computational fluid dynamics; PDE, partial differential equation.

**TABLE 3** Survey questions regarding compatibility with CFD (rate the following statements on the scale of 0 to 5)

![Figure 7](image_url)  
*Survey result regarding compatibility with CFD*
understanding of all the students. It can be seen from Q5 responses that the participants highly recommended that CFD should be incorporated as a formal component of advanced transport phenomena course in the postgraduate course curriculum.

6 | CONCLUSION

With the help of numerical simulations, we can find out loss coefficients of different pipe fittings such as reducers, elbows, bends, Tee junction, etc., which are useful in designing piping networks for any chemical plant. A systematic CFD-based approach to estimate the loss coefficient has been presented. The CFD simulations help to understand the nonidealities and complex flow patterns in the system for different pipe fittings. In the case of flow through reducer, due to different cross-section areas across the fitting, change in the inlet and outlet flow velocity was observed. For flow through elbow and bend, as the cross-section area across the fitting remains constant, the inlet and outlet flow velocities remain constant. In the case of flow through Tee junction, although the cross-section area across the fitting remains constant, variations in inlet and outlet flow velocities are observed due to the converging and diverging effects. The flow velocities across all these pipe fittings were in accordance with the continuity equation of the fluid dynamics. The participants found that CFD helps in understanding the complex flow patterns in the pipe fittings as well as the quantification of design parameters using computational tools. Due to the importance of CFD in understanding the fluid mechanics and physics of fluid flow, students who participated in the CFD workshop strongly recommended CFD to be taught as an integral part of the advanced transport phenomena course in the postgraduate course curriculum. As per the current industrial requirements and standards, hands-on experience of CFD was also found to improve the employability of the students in Engineering, Procurement, and Construction (EPC) firms.

ACKNOWLEDGMENTS

Authors acknowledge the Department of Atomic Energy, Government of India for the financial support for the project.

ETHICS STATEMENT

The authors declare that informed consent has been obtained from the participants of the questionnaire.

CONFLICT OF INTEREST

The authors have no conflict of interest relevant to this article.

NOMENCLATURE

- \( D \) diameter of pipe/elbow/bend (m)
- \( D_o \) outlet diameter of reducer (m)
- \( D_i \) inlet diameter of reducer (m)
- \( f \) friction factor (dimensionless)
- \( K \) head loss coefficient (dimensionless)
- \( k \) turbulent kinetic energy (m\(^2\)/s\(^2\))
- \( \Delta P \) pressure drop (Pa)
- \( R_c \) radius of curvature of elbow/bend (m)
- \( Re \) Reynold number (dimensionless)
- \( \bar{u}_i \) mean component of velocity vector (m/s)
- \( u'_i \) fluctuating (turbulent) component of velocity vector (m/s)
- \( V \) average velocity (m/s)
- \( \alpha \) flow deflection angle (°)
- \( \epsilon \) turbulent energy dissipation rate (m\(^2\)/s\(^3\))
- \( \mu \) dynamic viscosity (kg/m s)
- \( \rho \) fluid density (kg/m\(^3\))
- \( \tau_s \) wall shear stress (Pa)
- \( F_i \) body forces (N)
REFERENCES

1. Bullen PR, Cheeseman DJ, Hussain LA, Ruffelt AE. The determination of pipe contraction pressure loss coefficients for incompressible turbulent flow. *Int J Heat Fluid Flow*. 1987;8:111-118. https://doi.org/10.1016/0142-727X(87)90008-7.

2. Devenport WJ, Sutton EP. An experimental study of two flows through an axisymmetric sudden expansion. *Exp Fluids*. 1993;14:423-432. https://doi.org/10.1007/BF00190197.

3. Spalart PR, Coleman GN, Johnstone R. Direct numerical simulation of the Ekman layer: a step in Reynolds number, and cautious support for a log law with a shifted origin. *Phys Fluids*. 2008;20:101507. https://doi.org/10.1063/1.3005858.

4. Perumal K, Ganesan R. CFD modeling for the estimation of pressure loss coefficients of pipe fittings: an undergraduate project. *Comput Appl Eng Educ*. 2016;24:180-185. https://doi.org/10.1002/cae.21695.

5. Xing D, Yan C, Sun L, Xu C. Effects of rolling on characteristics of single-phase water flow in narrow rectangular ducts. *Nucl Eng Des*. 2012;247:221-229. https://doi.org/10.1016/j.nucleng.2012.03.010.

6. Gan G, Riffat SB. k-Factors for HVAC ducts: numerical and experimental determination. *Build Serv Eng Res Technol*. 1995;16:133-139. https://doi.org/10.1177/014362449501600303.

7. Smith AJW. The flow and pressure losses in smooth pipe bends of constant cross section. *Aeronaut J*. 1963;67:437-447. https://doi.org/10.1017/s0368393910078846.

8. Crawford NM, Cunningham G, Spence SWT. An experimental investigation into the pressure drop for turbulent flow in 90° elbow bends. *Proc Inst Mech Eng E—J Process Mech Eng*. 2007;221:77-88. https://doi.org/10.1243/09544088JIPME84.

9. Khanwale MA, Sona CS, Mathpati CS, Borgohain A, Maheshwari NK. Investigation of heat transfer characteristics and energy balance analysis of FLiNaK in turbulent boundary layers of pipe flow. *Appl Therm Eng*. 2015;75:1022-1033. https://doi.org/10.1016/j.applthermeng.2014.10.049.

10. Rani HP, Divya T, Sahaya RR, Kain V, Barua DK. CFD study of flow accelerated corrosion in 3D elbows. *Ann Nucl Energy*. 2014;69:344-351. https://doi.org/10.1016/j.anucene.2014.01.031.

11. Rodriguez I, Borell R, Lehmkuhl O, Perez Segarra CD, Oliva A. Direct numerical simulation of the flow over a sphere at Re = 3700. *J Fluid Mech*. 2011;679:263-287. https://doi.org/10.1017/jfm.2011.136.

12. Tiwari SS, Pal E, Bale S, et al. Flow past a single stationary sphere, 1. Experimental and numerical techniques. *Powder Technol*. 2019. https://doi.org/10.1016/j.powtec.2019.01.037.

13. Bale S, Tiwari SS, Sathe M, Berrouk AS, Nandakumar K, Joshi JB. Direct numerical simulation study of end effects and D/d ratio on mass transfer in packed beds. *Int J Heat Mass Transf*. 2018;127:234-244. https://doi.org/10.1016/j.ijheatmasstransfer.2018.07.100.

14. Karthik GM, Buwa VV. Particle-resolved simulations of methane steam reforming in multilayered packed beds. *AIChE J*. 2018;64:4162-4176. https://doi.org/10.1002/aic.16386.

15. Kalaga DV, Reddy RK, Joshi JB, Dalvi SV, Nandakumar K. Liquid phase axial mixing in solid–liquid circulating multistage fluidized bed: CFD modeling and RTD measurements. *Chem Eng J*. 2012;191:475-490. https://doi.org/10.1016/j.cej.2012.02.091.

16. Khan MS, Mitra S, Ghatage SV, Dorooodchi E, Joshi JB, Evans GM. Segregation and dispersion studies in binary solid-liquid fluidised beds: a theoretical and computational study. *Powder Technol*. 2017;314:400-411. https://doi.org/10.1016/j.powtec.2016.12.070.

17. Joshi JB, Nandakumar K, Evans GM, et al. Bubble generated turbulence and direct numerical simulations. *Chem Eng Sci*. 2017;157:26-75. https://doi.org/10.1016/j.ces.2016.03.041.

18. Khan Z, Bhusare VH, Joshi JB. Comparison of turbulence models for bubble column reactors. *Chem Eng Sci*. 2017;164:34-52. https://doi.org/10.1016/j.ces.2017.01.023.

19. Aslam Bhutta MM, Hayat N, Bashir MH, Khan AR, Ahmad KN, Khan S. CFD applications in various heat exchangers design: a review. *Appl Therm Eng*. 2012;32:1-12. https://doi.org/10.1016/j.applthermaleng.2011.09.001.

20. Pal E, Kumar I, Joshi JB, Maheshwari NK. CFD simulations of shell-side flow in a shell-and-tube type heat exchanger with and without baffles. *Chem Eng Sci*. 2016;143:314-340. https://doi.org/10.1016/j.cej.2016.01.011.

21. Adair D, Bakenov Z, Jaeger M. Building on a traditional chemical engineering curriculum using computational fluid dynamics. *Educ Chem Eng*. 2014;4:4:e85-493. https://doi.org/10.1016/j.ece.2014.06.001.

22. Joshi JB, Tabib MV, Deshpande SS, Mathpati CS. Dynamics of flow structures and transport phenomena, 1. Experimental and numerical techniques for identification and energy content of flow structures. *Ind Eng Chem Res*. 2009;48:8244-8284. https://doi.org/10.1021/ie9012506.

23. Mathpati CS, Tabib MV, Deshpande SS, Joshi JB. Dynamics of flow structures and transport phenomena, 2. Relationship with design objectives and design optimization. *Ind Eng Chem Res*. 2009;48:8285-8311. https://doi.org/10.1021/ie900396k.

24. Riffat SB, Gan G. CFD prediction of k-factors of duct elbows. *Int J Energy Res*. 1997;21:675-681.

25. Lin CH, Fering YM. Predictions of hydrodynamic characteristics and corrosion rates using CFD in the piping systems of pressurized-water reactor power plant. *Ann Nucl Energy*. 2014;65:214-222.

26. Röhrl R, Jakirić S, Tropea C. Comparative computational study of turbulent flow in a 90° pipe elbow. *Int J Heat Fluid Flow*. 2015;55:120-131.
27. Dutta P, Saha SK, Nandi N, Pal N. Numerical study on flow separation in 90° pipe bend under high Reynolds number by k-ε modelling. Eng Sci Technol. 2016;19:904-910.
28. Pal R, Hwang CYJ. Loss coefficients for flow of surfactant-stabilized emulsions through pipe components. Chem Eng Res Des. 1999;77:685-691. https://doi.org/10.1205/026387699526818.
29. Gan G, Riffat SB. Numerical determination of energy losses at duct junctions. Appl Energy. 2000;67:331-340. https://doi.org/10.1016/S0306-2619(00)00026-X.
30. Moujaes SF, Deshmukh S. Three-dimensional CFD predictions and experimental comparison of pressure drop of some common pipe fittings in turbulent flow. J Energy Eng. 2006;132:61-66. https://doi.org/10.1061/(ASCE)0733-9402.
31. Ma ZW, Zhang P. Pressure drops and loss coefficients of a phase change material slurry in pipe fittings. Int J Refrig. 2012;4:992-1002. https://doi.org/10.1016/j.ijrefrig.2012.01.010.
32. Abdulwahhab M, Niranjan IK, Sadoun FD. Numerical prediction of pressure loss of fluid in a T-junction. Int J Energy Environ. 2013;4:253-264.
33. Tian X, Sean Xue-Yong Z, Valade M, Bradshaw P. Head loss through pipe fittings for laminar flows. Pipelines Proceedings. Paper presented at: Pipelines 2013; Fort Worth, TX; 2013:301–308. https://doi.org/10.1061/9780784413012.028.
34. Lukiyanto YB, Wardana ING, Wijayanti W, Choirom MA. Flow visualization pattern on sharp edge T-junction through dividing flow channel. Appl Mech Mater. 2014;493:62-67. https://doi.org/10.4028/www.scientific.net/AMM.493.62.
35. Santos APP, de Andrade CR, Zaparoli EL. CFD prediction of the round elbow fitting loss coefficient. Int J Mech Mechatronics Eng. 2014;8:743-747.
36. Pieritz RA, Mendes R, Da Silva RFAF, Maliska CR. CFD studio: an educational software package for CFD analysis and design. Comput Appl Eng Educ. 2004;12:20-30. https://doi.org/10.1002/cae.10055.
37. Hung TC, Wang SK, Tai SW, Hung CT. An innovative improvement of engineering learning system using computational fluid dynamics concept. Comput Appl Eng Educ. 2005;13:306-315. https://doi.org/10.1002/cae.20056.
38. Vicéns JL, Zamora B. A teaching-learning method based on CFD assisted with MATLAB programming for hydraulic machinery courses. Comput Appl Eng Educ. 2014;22:630-638. https://doi.org/10.1002/cae.21554.
39. Gutiérrez JE, Zamora B. Improving teaching-learning process through ICT methods assisted with CFD techniques for marine engineering courses. Comput Appl Eng Educ. 2013;23:239-249. https://doi.org/10.1002/cae.21592.
40. Panagiotopoulos CG, Manolis GD. A web-based educational software for structural dynamics. Comput Appl Eng Educ. 2016;24(4):599-614. https://doi.org/10.1002/cae.21735.
41. Bird RB, Stewart WL, Lightfoot EN. Transport Phenomena. New York, NY: John Wiley & Sons; 2006.
42. Pope SB. Turbulent Flows. Cambridge, UK: Cambridge University Press; 2000.
43. Versteeg HK, Malalasekera W. An Introduction to Computational Fluid Dynamics—The Finite Volume Method. New York, NY: Pearson Education Limited; 1995.
44. Anderson J. Computational Fluid Dynamics. New York, NY: McGraw-Hill Education; 1995.
45. Rennels DC, Hudson HM. Pipe Flow: A Practical and Comprehensive Guide. Hoboken, NJ: John Wiley & Sons; 2012.
46. Miller DS. Internal Flow: A Guide to Losses in Pipe and Duct Systems. Cranfield, England: British Hydromechanics Research Association; 1971.
47. Hirai S, Takagi T, Matsumoto M. Predictions of the laminarization phenomena in an axially rotating pipe flow. J Fluids Eng. 1988;110:424-430. https://doi.org/10.1115/1.3243573.
48. Kitoh O. Experimental study of turbulent swirling flow in a straight pipe. J Fluid Mech. 1991;225:445-479. https://doi.org/10.1017/S0022112091002124.
49. Cočić AS, Lečić MR, Cantrak SM. Numerical analysis of axisymmetric turbulent swirling flow in circular pipe. Therm Sci. 2014;18:493-505. https://doi.org/10.2298TSCI130315064C.
50. Biswas G, Eswaran V. Turbulent Flows, Fundamentals, Experiments and Modeling. IIT Kanpur Series of Advanced Texts. New Delhi, India: Narosa Publishing House; 2002.
51. Fluent A. ANSYS Fluent Theory Guide. Canonsburg, PA: ANSYS Inc.; 2011:724-746.
# Appendix A

## A1 Introduction

### Table A1: Previous work

| References          | Pipe fittings                  | Fluid                                | Geometric parameters | Operating parameters | Remarks                                                                 |
|---------------------|--------------------------------|--------------------------------------|----------------------|----------------------|--------------------------------------------------------------------------|
| Riffat and Gan 24   | Rectangular and flat-oval duct elbows | Air $\rho = 1.225 \text{ kg/m}^3$
$\mu = 1.983 \times 10^{-5} \text{ Pa s}$ | $d = 0.3 \text{ m}$
$R_e = 1.5d$
Inlet duct $= 9 \text{ m}$
Outlet duct $= 15 \text{ m}$ | $v = 10 \text{ m/s}$ | Numerical modeling has shown that CFD can be used to predict $k$-factors for duct elbows of round, square, rectangular, and flat-oval cross-section. |

| Pal and Hwang 28    | Expansion, contraction, and globe valve | Oil in water emulsion and surfactant: Oil - Bayol-35
$\rho = 780 \text{ kg/m}^3$
$\mu = 2.72 \text{ mPa s}$ Emulsifier Triton X-100 dissolved in water | Expansion/Contraction
$d_{in} = 41.24 \text{ mm}$
$d_{out} = 20.37 \text{ mm}$; Globe valve (Fully open and half open) $d = 27.18 \text{ mm}$ | Expansion: $v = 0.9$ to $6.9 \text{ m/s}$;
Contraction: $v = 3.1$ to $6.5 \text{ m/s}$;
Globe valve (fully open):
$v = 0.1$ to $3.7 \text{ m/s}$;
Globe valve (half open): $v = 0.1$ to $3.2 \text{ m/s}$ | Loss coefficients obtained experimentally vs different $Re$ were correlated and presented. For expansion, loss coefficient is independent of the Reynolds number and has an average value of 0.49. For contraction, the loss coefficient is independent of the Reynolds number and has an average value of 0.43. The average loss coefficient for fully open and half open globe valve was 7.8 and 14.7, respectively. |

| Gan and Riffat 29   | Duct T-junction of square cross-section | Air $\rho = 1.225 \text{ kg/m}^3$
$\mu = 1.983 \times 10^{-5} \text{ Pa s}$ | Square duct of side $= 0.3 \text{ m}$ | $v = 10 \text{ m/s}$
$Re = 200000$
Model: $k-\varepsilon$
Scheme: QUICK | CFD predictions were carried for combining & dividing flows |

| Moujaes and Deshmukh 30 | Short elbow, Tee | Water | Elbow $d = 5 \text{ cm}$,
$L = 50 \text{ cm}$; Tee $d = 2.54 \text{ cm}$, $L = 50 \text{ cm}$ | $Re = 78000, 94000, 110000$, $122000, 125000, 141000$, $156000$
Model: $k-\varepsilon$, $k-\varepsilon$ Chen | Simulations were done with $k-\varepsilon$ models high $Re$ model and $k-\varepsilon$ Chen model. $K$ value results were compared with ASHRAE handbook values. |

(Continues)
| References          | Pipe fittings                          | Fluid                        | Geometric parameters                                                                 | Operating parameters                                                                 | Remarks                                                                                   |
|---------------------|----------------------------------------|------------------------------|--------------------------------------------------------------------------------------|----------------------------------------------------------------------------------------|-------------------------------------------------------------------------------------------|
| Ma and Zhang        | Straight tubes, sudden contractions and expansions, and elbows and tee | Tetrabutylammonium bromide (TBAB) and clathrate hydrate slurry (CHS) | Straight tubes: $d = 6$ and $14$ mm; Elbow = $14$ mm; Contraction from 6 to 14 mm; Expansion from 6 to 14 mm; Tee $d = 14$ mm | Coriolis mass flowmeter was used to determine the mass flow rate. Straight tubes: For 6 mm tubes $v = 0.8$ to $7.5$ m/s; Elbow: $v = 0.5$ to $2.4$ m/s; Reducer: $v = 1.8$ to $7.5$ m/s; Expander: $v = 1$ to $3$ m/s | $k$ values for multiphase system were experimentally calculated. |
| Abdulwahhab et al   | Tee junction                           | Water $\rho = 998.2$ kg/m$^3$ $\mu = 0.001$ Pa s | 2012: $D = 0.0254$ m, $L = 10$D; 2013: $D = 0.0254$ m, $L = 2$D. | 2012: $Re = 36,000$; 2013: $Re = 50,718$, 50,686, 50,676, 12,670. | ANSYS CFX was used for simulations. The $K$ values given by the numerical results is higher than those obtained from theoretical and experimental results. |
| Tian et al          | 90° Elbow, globe valve, Tee junction   | Water: $\mu = 1$ cP; Glycerol: $\mu = 219$ cP | $Re = 110,000$ (Water); $Re = 480$ (Glycerol); 10 gpm, 1 cP and 100 cP; 2.5 mgd, 25 mgd, 1 cP | This paper reviews Hooper 2-K method and Darby 3-K method for determining $K$ factors. CFD of 90° Elbow, Globe valve was done for different $Re$. |
| Lukiyanto et al     | Dividing T-junction duct               | Salt Water $\rho_f = 1080.27$ kg/m$^3$ $\rho_p = 1270$ kg/m$^3$ $\mu = 0.142007$ Pas | Rectangular Tee Straight Tee: $2$ mm $\times$ $20$ mm; Branch Tee $5$ mm $\times$ $20$ mm; | $v = 1.08$, 0.57, 0.17, 1.1, 0.6, 0.1 m/s | Computational and experiment study of dividing T-junction had been carried out to determine head losses and flow pattern. |
| Santos et al        | 90° Round elbow                       | Air                          | $d = 76$ mm $R_c = 1.5d$ $L = 10$d | $Re = 10^4$ to $10^6$ | A conventional 90° round elbow under turbulent incompressible airflow was numerically studied. |
| Perumal and Ganesan  | 90° Bend                               | Water $\rho = 998.2$ kg/m$^3$ $\mu = 0.001$ Pa s | Horizontal $L = 150$ in; Vertical $L = 50$ in | $Re = 10,000$ | CFD results has good agreement with the literature experimental results. |
| Present study       | Elbows, bends, Tee, reducer, expander | Water $\rho = 998.2$ kg/m$^3$ $\mu = 0.001$ Pa s | $d = 0.0254$ m $R_c = 1.5d$, $L \geq 10$d | $Re = 10,000$ | Head loss coefficient at different $Re$ was predicted with the help of CFD and compared with the previous literature. |
APPENDIX B

B.1 Governing equation

In CFD, the equations of motion and continuity are solved iteratively for the domain of interest using appropriate boundary conditions. These equations of continuity and momentum in their instantaneous form are shown below in Equations (B.1.1) and (B.1.2), respectively. These equations are for incompressible flow with constant properties.

\[ \frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0, \]  
\[ \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + F_i. \]

In order to solve these equations in their crude form, we need to perform DNSs using very fine grid sizes and a correspondingly small time-step size. This makes DNS computationally intensive. In order to achieve robust, reliable, and accurate solutions, we use the concept of spatial and temporal averaging to describe the turbulence effects. The most commonly used averaging concept is the time-averaging approach, that is, Reynold averaging approach. This involves the decomposition of any instantaneous flow parameter, for example, velocity \((u_i)\) into a mean \((\overline{u_i})\) and a fluctuating component \((u'_i)\) as shown in Equation (B.1.3) below.

\[ u_i = \overline{u_i} + u'_i. \]

Introducing the above-stated concept in Equations (B.1.1) and (B.1.2) results in the time-averaged form of the continuity and momentum equations as shown in Equations (B.1.4) and (B.1.5), respectively.

\[ \frac{\partial \overline{u_i}}{\partial x_i} = 0, \]  
\[ \frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \nu \frac{\partial \overline{u_i}}{\partial x_j} - \overline{u_i' u_j'} \right) + F_i. \]

As shown in Equation (B.1.5), the mean of product of two fluctuating terms, that is, Reynolds stress terms \(\overline{u'_i u'_j}\) leads to the closure problem. This closure problem is more commonly tackled by introducing the concept of eddy viscosity \((\mu_t)\) and using additional constitutive equations for the turbulent kinetic energy \((k)\) and rate of dissipation of turbulent kinetic energy \((\epsilon)\) or specific rate of dissipation of turbulent kinetic energy \((\omega)\). The \(k-\epsilon\) model, \(k-\omega\) model, shear stress transport (SST)-\(k-\omega\) model, and realizable \(k-\epsilon\) model are some of the commonly used RANS models, which use this concept of eddy viscosity. However, it has been observed previously that for the case of turbulent flow in pipe fittings using an eddy viscosity-based RANS model fail to predict the experimental data. This is mainly because the turbulent flow in pipes is anisotropic and the eddy viscosity RANS models are based on the assumption of isotropy. In contrast to this, the Reynolds stress approach is based on solving each of the six stress terms separately instead of modeling them using the eddy viscosity terms. Thus, RSM, in theory, can circumvent all the above-mentioned deficiencies of eddy viscosity models thereby predicting each of the individual stress terms more accurately. The six equations in RSM contain the following terms (a) accumulation, (b) convection, (c) production, (d) turbulent transport, (e) viscous transport, (f) dissipation, and (g) pressure strain, as shown in Equation (B.1.6).

Besides, these six equations, the equation for the rate of dissipation of turbulent kinetic energy is also solved in the modeled form as shown in Equation (B.1.7). The advantage of using the RSM model is that, unlike the eddy viscosity-based models, RSM does not assume turbulence to be isotropic. Thus, phenomena such as swirling effects, streamline curvature, rotational strains, and other body-force effects. Looking at the modeling capabilities of the RSM, we use the Launder-Gibson RSM (Equations [B.1.6] and [B.1.7]) in the present case. These models have been extensively corroborated and verified against experimental results for flow through pipe fittings,\(^{47-49}\) thus making them a reliable model to be used for the present case. Fundamentals of turbulent flows and modelling equations have been explained in great detail in Biswas and Eswaran.\(^{50}\)
\[
\begin{align*}
\{ \rho \frac{\partial \tau_{ij}}{\partial t} \} + \{ \rho \bar{u}_k \frac{\partial \tau_{ij}}{\partial x_k} \} &= \begin{bmatrix} -\rho \left( \tau_{ik} \frac{\partial \bar{u}_j}{\partial x_k} + \tau_{jk} \frac{\partial \bar{u}_i}{\partial x_k} \right) \end{bmatrix} + \begin{bmatrix} \frac{\partial (\mu \tau_{ij})}{\partial x_k} \end{bmatrix} + \begin{bmatrix} \frac{\partial \tau_{ij}}{\partial x_k} \end{bmatrix} \\
+ \begin{bmatrix} -\frac{2}{3} \varepsilon \delta_{ij} \end{bmatrix} + \{ \phi_{ij} \}. \\
\end{align*}
\] (B.1.6)

\[
\begin{align*}
\{ \rho \frac{\partial \varepsilon}{\partial t} \} + \{ \rho \bar{u}_j \frac{\partial \varepsilon}{\partial x_j} \} &= \begin{bmatrix} \rho C_{\varepsilon 1} \frac{\varepsilon}{k^2} \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} \end{bmatrix} + \begin{bmatrix} \frac{\partial (\mu \varepsilon)}{\partial x_j} \left( \frac{\mu}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) \end{bmatrix} + \begin{bmatrix} \frac{\partial \varepsilon}{\partial x_j} \left( \frac{\mu}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} \right) \end{bmatrix} + \begin{bmatrix} -C_{\varepsilon 2} \frac{\varepsilon^2}{k^2} \end{bmatrix}. \\
\end{align*}
\] (B.1.7)

In the above equations, the turbulent kinetic energy as shown in Equation (B.1.8) is essentially used for obtaining the boundary conditions of the Reynolds stresses.

\[
\begin{align*}
\{ \rho \frac{\partial k}{\partial t} \} + \{ \rho \bar{u}_j \frac{\partial k}{\partial x_j} \} &= \begin{bmatrix} \tau_{ij} \frac{\partial \bar{u}_i}{\partial x_j} \end{bmatrix} + \begin{bmatrix} \frac{\partial (\mu \tau_{ij})}{\partial x_k} \end{bmatrix} + \begin{bmatrix} \frac{\partial \tau_{ij}}{\partial x_k} \end{bmatrix} + \{ -\rho \varepsilon \}. \\
\end{align*}
\] (B.1.8)

Here, the eddy viscosity (\( \mu_t \)) term is modeled using the Boussinesq approximation (see Equation [B.1.9]) as done in the case of the \( k-\varepsilon \) equation.

\[
\mu_t = C_\mu \rho \frac{k^2}{\varepsilon}. \] (B.1.9)

The terms \( \sigma_k, \sigma_\varepsilon, C_\mu, C_{\varepsilon 1}, \) and \( C_{\varepsilon 2} \) in the above equation are set to their default values that is, \( \sigma_k = 0.82, \sigma_\varepsilon = 1.0, \)
\( C_\mu = 0.09, C_{\varepsilon 1} = 1.44, \) and \( C_{\varepsilon 2} = 1.92. \)