AVE **Trends in Intelligent Applied Sciences**

Computational Fluid Dynamics Analysis of Pressure and Flow in 90-Degree U-Shaped Pipe Bends for Power Plants

Mohammad Ayaz Ahmad[1], P. Banupriya[2,*], S. Silvia Priscila[3]

[1]Department of Mathematics, Physics and Statistics, University of Guyana, Georgetown, Guyana, South America.
[2]Department of Chemistry, Dhaanish Ahmed College of Engineering, Chennai, Tamil Nadu, India.
[3]Department of Computer Science, Bharath Institute of Higher Education and Research, Chennai, Tamil Nadu, India.
mohammad.ahmad@uog.edu.gy[1], banupriya@dhaanishcollege.in[2], silviaprisila.cbcs.cs@bharathuniv.ac.in[3]

*Corresponding author

Abstract: In a power plant, pipes are essential flow paths where fluids or steam are transported from one point to another. The fact that the plant has clear procedures in place makes the exact analysis of pressure changes across a 90-degree U-shaped pipe important through Computational Fluid Dynamics, towards understanding and optimizing fluid flow. Such analyses are of considerable significance across various industries, including chemical processing, petroleum, and heating, ventilation, and air conditioning systems. Fluid flow through pipes induces frictional losses, which can, therefore, have an impact on both hydrodynamic and thermohydrodynamic performance. Often, other complications such as elbows, junctions, T-joints, contractions, expansions, valves, meters, pumps, and turbines accompany the bent pipes and consequently influence the fluid dynamics. The number of bends along the length of the pipe considerably affects the structure of the flow and the behaviour of the fluid. The pressure and stress distribution in bends are applied for CFD analysis, providing valuable insight into the hydrodynamic properties of the fluid under consideration, which in turn helps improve the system's efficiency and reliability. In this respect, understanding the dynamics is important to optimise pipe design and ensure that fluids are moved effectively in power plants and industrial systems.

Keywords: Fluent Analysis; CFD Ent pipe and Pressure; Fluid Pipe; Mesh Mode; Turbulent Kinetic Energy; Darcy Weisbach Formula; Pressure and Stress Distribution; T-Joint and Thermo-Hydrodynamics.

Cite as: M. A. Ahmad, P. Banupriya, and S. S. Priscila, “Computational Fluid Dynamics Analysis of Pressure and Flow in 90-Degree U-Shaped Pipe Bends for Power Plants,” *AVE Trends in Intelligent Applied Sciences.*, vol. 1, no. 1, pp. 27–36, 2025.

Journal Homepage: https://www.avepubs.com/user/journals/details/ATIAS

Received on: 25/04/2024, **Revised on:** 04/06/2024, **Accepted on:** 13/08/2024, **Published on:** 07/03/2025

DOI: https://doi.org/10.64091/ATIAS.2025.000113

1. Introduction

Researchers have identified a crucial tool in the field of computational fluid dynamics for studying fluid flow through bent pipes and other piping configurations commonly used in industrial practices [12]. These configurations typically comprise geothermal power plants, chemical processing facilities, and HVAC units, each relying on an efficient method to transport fluids in the form of steam, brine, or gas [3]. The curved pipes and elbows of the system shall be safe, reliable, efficient, with reduced pressure loss, and energy dissipation [7]. For example, geothermal piping has issues with thermal expansion, seismic activity, and exposure to environmental conditions such as rain, wind, and landslides [14]. Piping infrastructure covers the production well, through the separation stations, and up to reinjection wells and often needs to cross over uneven terrain and steeper slopes [2]. Ideally, design requirements include that it must be both flexible and stiff, such that neither thermal nor operational loads compromise system integrity [9].

The piping system of a geothermal setup has several critical components, such as:. Two-phase flow piping collects fluids from various wellheads and directs them to a separator vessel. The vessel then separates the steam from the brine [5]. The steam is sent through separate pipelines to the power plant, while the separated brine is transferred through brine pipelines to be re-injected into suitable wells [8]. Other miscellaneous piping, such as instrumental airlines and water supply lines, adds further complexity to the system [10]. In such cases, analysis by CFD is inevitable for predicting pressure differences, flow behavior, and stress distribution in components to ensure optimized designs enhance performance and prolong longevity [1].

The primary reasons behind the bent pipe systems in CFD analysis are the revelation of pressure differences at the pipe bends, an increase in fluid flow rate, and structural weaknesses such as cracks resulting from pressure differences. Furthermore, the overall flow efficiency needs to be optimized [13]. A review also incorporates the performance of a pipe elbow. It is necessary to change the direction of the flow and meet space requirements [4]. Pipe elbows are applied in several industrial processes, including chemical and petroleum operations, as well as electronic and pulp and paper manufacturing [15]. They are produced from materials chosen to suit specific service conditions, such as thermal resistance and pressure resistance [6]. According to various design requirements, the range of pipe elbows includes both short-radius and long-radius elbows [11]. A short-radius elbow is the most preferred for use in very confined, tight spaces. In less restrained areas, elbows that offer the least resistance are usually preferred when using a long radius [3]. Further enhancing the flexibility of piping systems are elbows that reduce the number of elbows, including male pipe elbows and female pipe elbows. These can facilitate arrangements between two pipes of any size and configuration, making them more adaptable [14].

The energy loss that a bent pipe experiences in fluid flow is an important consideration in the design and analysis of such a pipe [9]. Such losses can be generally categorized into major losses and minor losses. These are primarily caused by friction and changes in velocity or direction [10]. The friction losses are generally estimated using Darcy-Weisbach and Chezy's formulae and occur mainly due to the loss of fluids to the pipe walls [8]. The influence of wall roughness is appreciable at the turbulent flow regimes; however, for laminar flow, wall roughness is of little significance [4]. The fluid's physical properties determine frictional losses, as well as the interaction of fluid particles with bends, wrinkles, or any pipe obstruction [6]. Minor losses occur due to abrupt changes in the pipe geometry, such as contractions, expansions, fittings, and bends [13]. It is the disruption on the local level by such means that creates turbulence and wastes energy; hence, design considerations must be careful not to permit it from taking place [1]. For example, sudden enlargement in the pipe's Diameter will produce turbulent eddies, causing flow separation leading to an energy loss in [11]. The magnitude of the energy loss depends on the difference in diameters, as well as the associated turbulence intensity [2].

On the other hand, sudden contractions increase the downstream velocity, creating a vena contracta where the flow converges before expanding to fill the smaller pipe [7]. Such flow characteristics can be quantitatively accounted for through CFD analysis, which also accounts for energy loss [3]. Local eddies appear due to the centripetal forces, along with the pressure gradients formed in the flow, because of the secondary pattern due to fluid flowing through a bend. Pipelines involving bends add complications. Piping systems involving bends typically add complexity to these systems [12]. Bend analysis and design would require consideration of factors for efficient fluid transport within pipes [15]. The CFD analysis may give a more in-depth understanding of such events, which helps engineers foresee and compensate for such energy losses [5]. Frictional head loss is determined through the Darcy-Weisbach equation. The Darcy-Weisbach equation consists of parameters such as the Diameter of the pipe, velocity of flow, and friction factor [14]. Chezy's formula relates the mean velocity of flow to hydraulic parameters [9].

Minor losses, as defined by pipe geometries and determined through specific coefficients, aid in establishing zones of potential design optimisation [13]. By modeling the flow conditions of bent pipes, CFD analysis can help identify pressure distributions, velocity profiles, and characteristics of turbulence; thus, design optimization would improve the system's performance [4]. Additionally, CFD analysis is useful for identifying structural flaws in bent pipes [10]. Pressure differences developed within the bends of a pipe are regions of stress in which the existence of stress concentrations may intensify the potential for cracking and failure [6]. Visualisation of the stress distribution can be useful in enabling the reinforcement of stress-rich areas, selecting suitable materials, and changing the design to reduce the risks [8]. Furthermore, optimising the flow efficiency in bent pipes minimises operating costs and maximises system reliability [2].

This is very critical in applications such as geothermal power, where fluid transport efficiency directly impacts energy output and economic viability [7]. CF application in the optimisation and design process of bent pipes in industrial operation is very fundamental [1]. CFD ensures that any piping system efficiently and reliably meets the demands based on the computation of problems related to pressure loss, energy dissipation, or structural integrity when working on such industrial applications [15]. From power plants to industries involving chemicals or petroleum, insights from CFD simulations drive innovations in design, allowing pipelines to meet these advanced demands of contemporary engineering and environmentally sustainable systems [12].

2. Literature survey

Based on its sensitivity, CFD can be broadly applied to model and predict fluid flow behaviour in bent pipes and complex geometries [5]. Researchers have found that U-shaped and bent pipes are exposed to significant pressure and stress variations due to their geometry, which affects heat transfer and fluid dynamics [3]. This makes the secondary flows and flow separation especially crucial for turbulent conditions [8]. These have therefore led to the provision of empirical corrections that are of value for use in engineering applications. The following examples of investigations, including measurements of the velocity distribution at several distances from the outlet of a bend, have shown that the pressure drop in two-phase flow depends on the bend angle [12, 6]. T-junctions in pipelines are very important as they are responsible for the divergence and convergence of flows in plants [9]. These components affect the thermal efficiency and fluid flow rates; it is found that velocity decreases and pressure generally decreases after passing through a T-junction [11]. This is as a result of energy losses due to the bending angle and the type of fluid flow [14]. An example is the bends with angles in the range of 60° to 125°, which have been studied for their effects on heat transfer rates and turbulence [7]. However, since this results in a loss of turbulence, which is undesirable from a fluid velocity and pressure perspective, it increases heat transfer, which becomes necessary when balancing these factors during design [4].

The two-phase flow in 90-degree elbows was studied at various velocities; pressure drop occurs when the fluid leaves the bend [13]. The pressure drop is proportional to air velocity, and the normalized pressure drops depend on the velocity of the fluid phases [2]. Further studies establish the interaction of fluids inside bent pipes and their effects on efficiency during operations [10]. Bent pipe design and other fittings are highly demanding [1]. Ovality is a condition resulting from deformation in the geometric form of a pipe, and flow rates may decrease significantly [15]. The angled branch design, vane elbow, and their inclusion pose significant complexities due to pressure drops associated with pipe branch connections [6]. Such complexities require more sophisticated modelling techniques that can predict fluid dynamics and structural behavior to an acceptable accuracy [8]. CFD analysis provides a comprehensive method for addressing the problem, as it can model various pipe geometries, including elbows, sudden expansions, and contractions [9]. This would allow for minor losses and identify the pressure differentials that could lead to structural weaknesses, such as cracks [14].

It emerges in the systematic modeling and Simulation of the phenomena in such a manner as to obtain precise values of the loss coefficients and validate CFD results by comparison with experimentally obtained observations [12]. Latest advanced CFD tools, such as ANSYS, facilitate detailed fluid flow modeling within diverse geometries, including mass flow rates, velocity, and density, across various pipe diameters [5]. Hence, the results from the Simulation can then be compared to empirical data to enhance the design methodology and predictive accuracy [7]. The design process for CFD-based research entails building the desired geometry, meshing the domain into small volumes of discrete elements, and specifying boundary and initial conditions [2]. This enables one to model the fluid dynamics within the system and analyze the study's results [13]. Other significant parameters to be modelled include outer and inner diameters of the pipe, thickness, bend radius, and nominal Diameter [3]. Analysis of factors such as mass flow rate, velocity, and density, among others, is taken into account when predicting fluid behavior under diverse operating conditions [11].

The ASME/ANSI B36.10 and B36.19 standards provide vital information on the design of steel pipes, including dimensions, wall thicknesses, and working pressures [10]. They ensure uniformity in the manufacturing process and compatibility in applications [1]. As the schedule numbers increase, so does the wall thickness, resulting in a smaller actual bore, which affects fluid dynamics [15]. CFD analysis is highly useful in addressing some of the challenges associated with fluid flow through curved pipes and other analogous geometries [4]. High-order modelling approaches can be utilized for the optimization of a pipe design, ensuring minimal energy losses, maximum heat transfer, and overall system efficiency [8]. Empirical data, combined with simulation results, have further deepened our understanding of fluid mechanics. Industrial piping systems, in their complex forms, pose innovative solutions to these issues [6].

3. Methodology

The methodology used in performing CFD analysis with bent pipes employs a rigorous step-by-step approach to achieve proper modeling and analysis of the fluid flow behavior. In that respect, an expansive literature review at the outset would allow for the proper setting up of the prior study and would therefore ensure the creation of a solid base of knowledge. The knowledge of prior work on pipe geometries, flow characteristics, and computational techniques provides a solid foundation for understanding the original paper's objectives. It then gathers data relevant to the task at hand, such as pipe dimensions, flow velocities, and the physical properties of the fluids involved. These provide the foundation upon which modelling and Simulation are done. The physical test rig is mounted. Open the bench valve, gate valve, and flow control valve; switch on the pump; fill up the rig with water. Then, bleed air carefully through the pressure tap points and manometers to achieve the desired reading accuracy. Adjust the water level using an air bleed screw and obtain a steady-state reading at the chosen flow rates. Flow rates are determined based on measurements of time for filling a storage tank with a specified water volume and repeated changes in flow conditions. Pressure drops in the system can be measured using pressure gauges across gate valves and other system components to better understand the losses within the system. The process is of utmost importance because it verifies CFD results by establishing experimental benchmarking.

Modelling: This involves creating 3D geometries using SolidWorks. This is parametric design technology, characterized by user-friendliness and adaptability. Under this pack, users can express design intent through both automatic and user-defined relations. This ensures that the geometric constraint dynamically evolves, responding to changes made in the software. These models are imported into the ANSYS Workbench in the form of an IGES file for analysis under CFD. ANSYS is one of the highly powerful simulation tools to handle geometries, meshing, and physics-driven simulations with high accuracy. Meshing is done to divide the domain into small volumes for the numerical solution of the governing equations. The boundary and initial conditions, along with their associated parameters, have been defined, including the inlet velocity, wall properties, and fluid types. This process utilizes the FLUENT module of ANSYS to perform the CFD simulation of water and hydrocarbon fluids. Water and hydrocarbon fluids have been selected for this application, specifically cyclohexene, with the molecular formula C_6H_{10}. This involves the study of the behavior of fluids, along with critical parameters such as pressure, velocity, shear stress, and kinetic energy. The results are presented in graphical contour plots that accurately depict the motion of the fluid within the pipe. These contours describe how pressure and velocities vary across the pipe, with regions of turbulence and shear stresses important prerequisites for understanding geometric effects on dynamics in pipe flows.

Values are listed for comparison over a range of fluids and geometries. Minor losses, such as sudden expansions, contractions, and bends, are among the synthesized results obtained from both the experimental and computational approaches. According to the experiment, loss coefficients can be obtained accurately using ANSYS. It makes a comparison with experimental observations, validating the numerical model iteratively so that phenomena are accurately captured by the model rather than producing unreliable results. It further elaborates on the relevance of the approach as these applications' software tools utilise it. Feature-based design streamlines the process of creating 3D models in SolidWorks, allowing for rapid prototyping and subsequent iterative adjustments. Additionally, the integration of processes such as geometry handling, meshing, and post-processing streamlines the workflow for ANSYS Workbench. Advanced simulations, including multi-physics, enable the effective analysis of complex fluid dynamic situations, such as turbulent flows with intricate geometries.

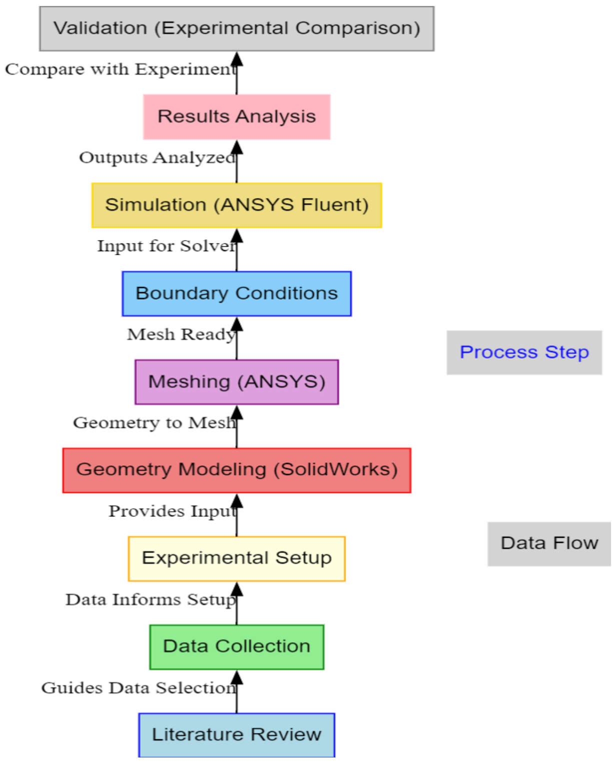

Figure 1: Workflow of Computational Fluid Dynamics (CFD) analysis for bent pipes

Figure 1 illustrates the Bent pipe CFD flow diagram, outlining all methodology steps. A literature review, highlighted in light blue, will be conducted to compile basic knowledge, which serves as the foundation for the actual study. This leads further to "Data Collection" in light green, where information regarding the critical data associated with pipe sizes and fluid properties is collected. The "Experimental Setup" is depicted in light yellow, which outlines the physical test rig setup for benchmarking purposes. The workflow further leads to "Geometry Modelling" in light coral, where the pipe is modelled using SolidWorks software. Then this geometry developed would be transmitted to the "Meshing" stage by the plum, which subdivides the domain for Simulation. Boundary Conditions, light sky blue, incorporate the real simulation parameters and can be further drawn in the Simulation, light goldenrod, to characterise flow via ANSYS Fluent. Then "Results Analysis shall be in the light pink that has proven validations in the comparison, and has light grey that has been given between the dataset that has experimentally been produced and the result from the simulated one. The diagram illustrates the correct mapping of the interconnections between the steps and data flow, ensuring a clear understanding of the systematic approach. In this flow, completeness in analysis is ensured also due to the focus on accuracy at each stage, which is combined with validation.

The Simulation of flow begins with building and meshing the geometry. Several simplifications would depend on the complexity of the model to be imposed, while retaining the vital features important for fluid behavior. The meshing process divides the domain into small, manageable volumes, allowing numerical methods to be applied iteratively to solve the governing equations. Much care is taken in defining the boundary and initial conditions to represent real operating scenarios. Simulations are then performed, and the results are interpreted through contour plots, graphs, and tabulated data. Outputs in such forms are useful for interpreting pressure distributions and velocity profiles with energy loss, and are particularly taken as design improvement guides. One of the techniques involved in bent pipe analysis combines an experimental approach with a computational approach, thereby achieving a comprehensive understanding of fluid dynamics. With the advanced tools like SolidWorks and ANSYS, one can model complex geometries and simulate realistic flow conditions. Moreover, one can validate their findings with experimental data. This integrated approach not only improves the accuracy of simulations but also helps in optimising the design of pipes for industrial applications. Such studies provide insights into the development of piping systems that are efficient and reliable in performance, thereby addressing problems associated with energy losses, turbulence, and structural integrity.

4. Results

The results of the CFD analysis of curved pipes will illuminate the dynamics of flow and energy losses caused by geometrical disruptors, such as sudden expansions and contractions. Five different geometries of pipes with ovality were simulated using ANSYS software. The software was used in the computation to analyse reasons for their velocity profiles and pressure distributions, among other factors that involve turbulence intensity and shear stress. Each Simulation demonstrated how the pressure drops and the flow efficiency are highly affected by the level of ovality of the pipes. Sharp inefficiencies in the geometrical configurations were observed at higher ovality. These results are particularly useful for identifying regions with high-pressure gradients that are prone to cracking and structural failure. The risk of sudden expansion and contraction was more pronounced when visualizing stress concentrations and turbulent flow patterns. Continuity equation is:

$$\nabla \cdot u = 0 \qquad (1)$$

Where u is the velocity vector. Navier-Stokes equation:

$$\rho(\frac{\partial u}{\partial t} + u\nabla u) = -\nabla p + \mu\nabla^2 u + F \qquad (2)$$

Where ρ is the fluid density, p is pressure, μ is dynamic viscosity, and F represents body forces. Energy equation is:

$$\rho C_p\, (\frac{\partial T}{\partial t} + u\, \nabla T) = k\nabla^2 T + \Phi \qquad (3)$$

Where C_p is the specific heat at constant pressure, T is the temperature, k is the thermal conductivity, and Φ represents viscous dissipation.

Table 1: The final result of the pipe elbow under various ovality

Ovality	0%	5%	10%	15%	20%
Thickness	1.9558	1.9558	1.9558	1.9558	1.9558
Bend pipe radius	101.35	101.35	101.35	101.35	101.35
MJR_O(ellipse)	30.1625	29.4088	28.6543	27.9003	27.1462

MIR_O(ellipse)	30.1625	30.9165	31.6706	32.4246	33.1787
Pipe radius	31.1875	31.1875	31.1875	31.1875	31.1875

Table 1 shows the ovality effect in various parameters under investigation for the pipe elbow. As ovality deviates from an ideal circular cross-section, a variation of between 0 and 20% was considered; the pipe's wall thickness remained constant at 1.9558 mm; the bend radius was 101.35 mm, ensuring the pipe's geometry remained constant for the analysis. MJR_O and MIR_O are the outer elliptical profile major and minor radii. The major and minor radius increase with the increment of the ovality. The major radius varies from 30.1625 mm by 0% ovality to 27.1462 mm by 20% ovality, and the minor radius varies from 30.1625 mm to 33.1787 mm. Still, the pipe radius remains constant at all levels of ovality, at 31.1875 mm. This means that ovality indeed significantly affects the cross-sectional shape of the pipe, but has a negligible effect on the overall dimensions. One of the important findings of the present study concerns the eddies created by turbulence, the dissipation of energy, and hence the loss of efficiency of the flow in the form of pressure drops. Abrupt contractions caused pulses in velocity and unstable flow, leading to spikes in velocity that increased inefficiency. Pipe elbows experienced secondary flows and localised turbulence, which made the contribution of pipe elbows to the overall uniformity of the flow rather desirable. Through comparison with other geometries, it is now possible to confirm that some of these design alterations were indeed made, as they significantly reduced energy loss, thereby achieving the maximum overall efficiency in the flow. Contours, pressure, and velocity graphs explaining the complex fluid dynamics were generated using ANSYS software.

The most important outcome of this study is the pressure-induced crack dangers. They were achieved in specific geometries, high-ovality or abrupt-transition geometries. These deficiencies required correction at the design stage to ensure the system's structural integrity and longevity. The numerical experiments were useful for engineers in providing appropriate metrics for calculating small losses, allowing inefficiencies in fluid transport to be predicted and overcome. For instance, the level of ovality was found to be directly related to pressure drops over a sudden contraction, and smooth transitions decreased such losses. Furthermore, alterations in the turbulence kinetic energy at elbows underscore the importance of incorporating only suitable modifications in curvature and pipe diameter design. Besides, material properties and boundary conditions determine the response of fluids. The findings are also strongly manifested in the calculation. Water simulations in these pipes demonstrated an impressively high correlation between flow velocity, pressure gradients, and stress within the structure. Such results have a significant impact on industries whose operations depend on efficient fluid transport, such as water distribution, geothermal energy, and chemical processing. The results indicated that CFD analysis is predictive, making it useful for identifying design defects in pipe configurations to increase efficiency in pipe operations and subsequently reduce energy use.

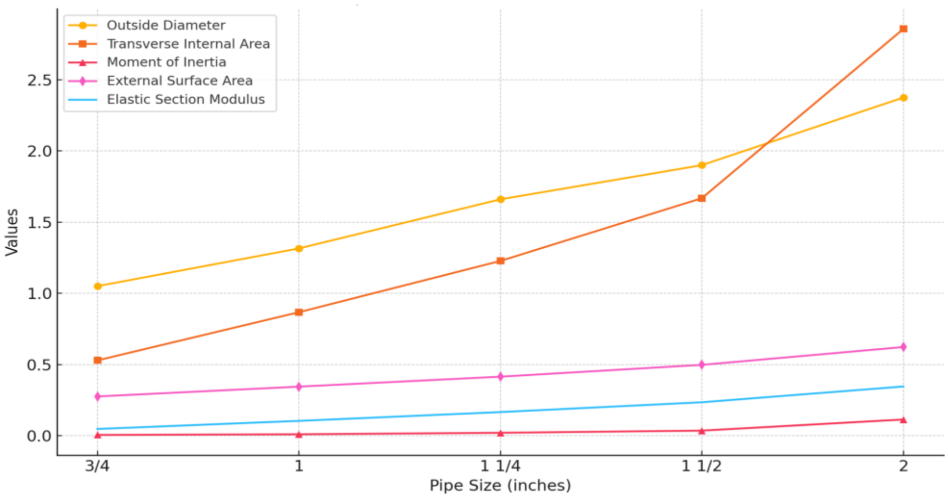

Figure 2: Representation of pipe characteristics based on size

Figure 2 illustrates the trend for some of the most critical parameters of steel pipes, categorized by size, where various parameters are plotted in relation to the increase in pipe sizes from 3/4 inches to 2 inches. The outside diameter increases in a straight-line, proportional manner along the yellow line for the pipe dimensions as the size increases. The same applies to the Transverse Internal Area, or the cross-sectional area available for fluid flow. It increases, and hence the larger pipes have a better flow capacity. The Moment of Inertia is a measure of resistance to bending. This increase is directly related to the pipe's size, indicating that larger pipes possess structural advantages in resisting mechanical stress. The External Surface Area is represented by the magenta curve, indicating that it is an increasing function, which is important for the transfer of heat and in surface treatments. And finally, the Elastic Section Modulus, which measures a pipe's resistance to deformation when bent under load, increases with pipe size; therefore, all pipes are stronger as they become larger. This graph provides an overview of the interrelationship between pipe size and structural properties, illustrating how these interrelationships develop with size and on which trends decisions can be based for selecting the appropriate pipe size in applications such as fluid transport, construction, and mechanical systems. This graph represents the balance between dimensional scalability and mechanical performance in the design and implementation processes. Wall shear stress (τ_w):

$$\tau_w = \mu \frac{\partial u}{\partial y} |_{wall} \quad (4)$$

Where u is the velocity parallel to the wall and y is the perpendicular distance to the wall. Friction factor (f) in pipes:

$$f = \frac{\Delta p}{\frac{L}{D}\frac{\rho u^2}{2}} \quad (5)$$

Where Δp is the pressure drop, L is the pipe length, D is the pipe diameter, and u is the mean velocity. Your provided equations:

$$C_0 = \frac{D_{\max} - D_{\min}}{D_0} \times 100 \quad (6)$$

$$D_0 = \frac{D_{\max} + D_{\min}}{2} \quad (7)$$

where D_(max) and D_(min) are maximum and minimum diameters, respectively, and D_0 is the average diameter.

Table 2: Cost estimation

S. No	Description	Cost (In Rupees)
1.	Software purchase	16000
2.	Project planning	350
3.	Printing and binding	1360
4.	Indirect expense cost	250
Total Cost		17960 Rupees

Table 2 is based on cost estimation. The cost of the products is 17,960 Rupees. This can be divided into a cost of 16,000 Rupees for buying software, as specialised computational tools are required in the analysis. Other costs include 350 Rupees for paper planning and 1,360 Rupees for printing and binding, which are necessary for documenting and presenting the results. Above Rs 250 would be considered an indirect cost, which would be utilized as a tool to account for scattered costs, such as electrical power or smaller consumables. This approach ensures an explicit breakdown of the cost, making it useful for allocating resources, particularly in analyses derived from Computational Fluid Dynamics and financial analysis. From the table, it is quite evident that software is one of the significant costs, as only advanced tools are relied upon for such technical analyses. Besides identifying problems in flow efficiency when passing through curved pipes, work done here introduced several solutions for practical improvement, such as fewer abrupt changes of pipe geometry to be used during construction, usage of more resistive materials, which will be beneficial for resistance due to differential pressure, and the use of flow-straightening devices. These can improve performance and reliability during the construction process by being applied in designs.

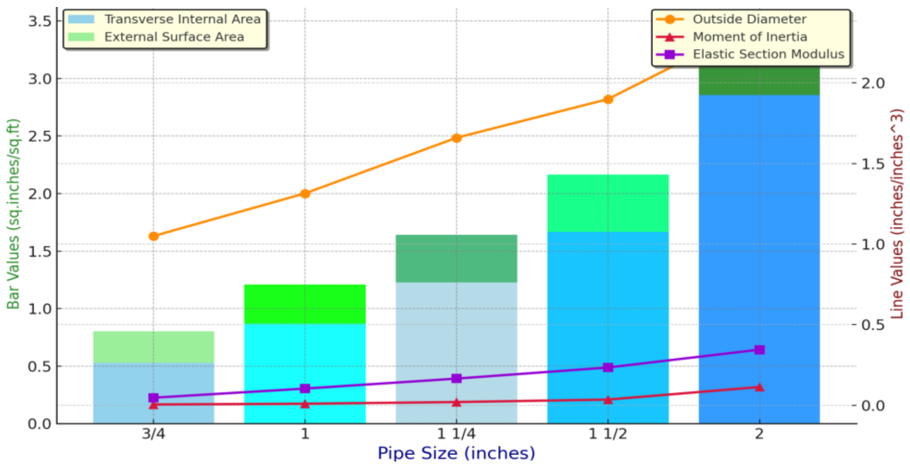

Figure 3: Enhanced representation of pipe size dependence on various characteristics of the pipes

Another consequence of the experimental validation of CFD results involves simulated results that closely represent actual conditions and are thus applicable to a wide range of industrial scenarios. Detailed studies have been conducted to compare five different ovality pipe geometries, providing a sound basis for understanding the complexities in fluid dynamics within bent pipes. All such explorations can be further carried out very easily with highly advanced simulation software, for example, ANSYS, and will provide more detailed insights into the behavior of flow, along with major zones of stress that may be useful for visualizing their turbulent interaction. All such discoveries go far beyond the simple quantification of minor losses, which will be closed up with theoretical models and real practices. This research provides actionable insights for engineers and researchers interested in optimising piping systems in industries. It incorporates detailed visualization, quantitative metrics, and targeted recommendations that form the basis for designing more efficient, reliable, and sustainable fluid transport solutions. The CFD analysis of bent pipes elucidates how profoundly geometry affects the efficiency and structural integrity of the flow. Such subtleties in minor losses were identified, and the importance of having sophisticated simulation tools to optimize pipe designs is underscored. The lessons acquired provide industries with a means to improve piping systems by overcoming energy dissipation, turbulence, and material stress, resulting in more resistant and efficient structures. A successful CFD analysis, therefore, is not some abstract academic exercise but the practical means toward solving real-world problems in fluid dynamics and engineering.

Figure 3 illustrates the dependence of pipe size on various characteristics. The bar stacks are combined with different colour schemes and a bold line plot for aesthetic appeal. In the bar stacks, the areas of Transverse Internal Area are shaded in distinct shades of blue, and those of External Surface Area are in shades of green. Both of the parameters are, therefore, directly proportional to the increased pipe size. At first glance, flow capacity and surface area appear more favourable for larger pipes. Colour differentiation by stacking enables the comparison of pipe characteristics. Added to the top of the bars, in bright orange, crimson, and purple are line plots for Outside Diameter, Moment of Inertia, and Elastic Section Modulus, respectively. The Outside Diameter has scaled with pipe size and is characteristic of proportional scaling. Both Moment of Inertia and Elastic Section Modulus quantify the resistance of this pipe to bending and deformation. Therefore, in this graph, both volumetric and structural properties of the influence that an increase in the size of pipes has on those properties are presented. It is both informative and aesthetically pleasing due to its rich combination of bright colours. It can be considered a powerful visualisation tool for engineers and designers, helping them select the appropriate pipe sizes in industrial applications, taking into account both flow efficiency and structural requirements.

5. Discussions

The major findings of the analysis, encompassing various aspects such as geometrical dimensions, structural properties, and economic considerations related to curved pipes, are presented in the discussion. To gain an understanding of the effect of Ovality, Table 1 has been taken into consideration. The results show that increasing the ovality percentage from 0% to 20% leads to significant changes in both the mother and radii. The thickness is uniform at 1.9558 mm, and the bend pipe radius is uniform at 101.35 mm. However, there is a gradual reduction in the major radius, and under higher ovality conditions, deformation becomes visible. On the contrary, a minor increase in radius demonstrates the dynamic variation in geometry, which immediately affects the fluid dynamics of the structure, thereby influencing its performance. The smooth pipe radius of 31.1875 mm, despite all ovality percentages, demonstrates that even with such external geometrical distortion, the internal, core part of the structural pipe remains reliable. This is a key finding because it now makes bent pipes in general susceptible to deformation-related inefficiencies, with particular emphasis on high percentages of ovality.

In Figure 2, the outside diameter, transverse internal area, moment of inertia, and elastic section modulus are all interrelated, illustrating how structural strength and hydraulic efficiency scale upward with increased pipe diameters. The bigger the pipe sizes are, the greater the magnitudes of those quantities, so more stresses and fluid volumes can be carried by them. A similar combined graph of this nature, illustrating the relationship between transverse internal area, external surface area, and structural parameters such as the moment of inertia and elastic section modulus, is shown below in Figure 3. The colour-filled chart illustrates how larger pipe sizes inherently optimise both flow capacity and structural resilience. From the results provided in Table 1, proper size selection during design stages, in addition to ovality effects for the pipe, will be taken into account. This is followed by the illustration that will be used to determine minor losses due to sudden expansion, contraction, and bending, where the fluid flow behavior in curved pipes will be graphically represented. Figure 3: Stacked bar representations of trends for transverse internal area and external surface area; line graphs give an idea about responses in the structures. It appears that higher values of ovality percentage increase the loss of pressure and vulnerability, necessitating the use of sharp computational models for these predictions. In most aspects, the regions of high stress, along with turbulence regions, signify risk mitigation of cracking and deformation, which directly affects areas dependent on fluids transported through such industries, particularly chemical processing, geothermal energy, and water distribution.

Table 2 presents the economic feasibility of this study, which will be further illustrated by comparing costs with those of bent pipes for CFD analyses. The overall cost of 17,960 rupees covers the purchase of software, planning of the paper, printing and binding, as well as indirect costs. This estimation also reflects that high-simulation tools like ANSYS are available to optimize pipe design, so that costly operational inefficiencies can be avoided. Such an investment in software is worthwhile because it provides rich insights, enabling engineers to achieve better performance and durability in industrial piping systems. The paper provides a general overview of bent pipe CFD analysis, encompassing geometric, structural, and economic analyses. Based on the results from Figures 2 and 3, combined with data from Tables 1 and 2, this study examines the critical interaction among three areas: pipe geometry, fluid dynamics, and structural integrity. It ensures that CFD tools can solve problems related to ovality, turbulence, and stress concentration. Such studies have emphasized the optimization of design to achieve flow efficiency, as well as provide structural robustness in bent pipes, through advanced simulations in many industrial applications. The analyses prove to be economically feasible, leading to the development of cheaper and more sustainable solutions for engineering practices.

6. Conclusion

As presented, the calculation results of the curved pipes revealed that the simulation tools like ANSYS were capable of approximating fluid dynamics characteristics with a fair degree of accuracy. The comparison between experimental outcomes and those obtained via ANSYS, for most geometry types, including elbows, reductions, and enlargements, showed a relatively accurate agreement. End. In the case of pipe bends, the smaller loss coefficient from experiment was compared to that resulting from ANSYS. Less, therefore, is the validation of methods. Considering contractions, the results from ANSYS yielded a higher value than the experimental result, further reiterating the notion that the computed model is correct and valid. The deviation, however, turns out to be significant in the expansion results. During the case of expansion, even a slightly high value of the coefficient loss of the ANSYS model compared with the experiment was reported. However, here also, the gap was not large, and even both approaches allowed reasonably good comparisons to be drawn between them. It brings consistency to the ability of CFD applications in modelling reality with high precision for real fluids, thus substituting designers and engineers with the convenience of practical substitutes rather than time-consuming and expensive experiments. The predictive power of ANSYS is confirmed herein, especially for complex geometries such as bent pipes, whose loss coefficients determine the optimization of flow efficiency or energy dissipation. It bridges the gap between experimental observations and computational predictions in the study, reinforcing the need for simulation-based approaches in the design and analysis of industrial piping systems. With CFD, application studies can enable improvements in dependability, efficacy, and productivity for bent-pipe systems applications across various industrial sectors on the path to sustainability through creative innovation.

6.1. Limitations

Bent pipe analysis using CFD is highly effective. However, this study has several limitations. For example, mostly idealized boundary conditions and assumptions, such as steady-state flow, incompressible fluids, or isotropic turbulence, are rarely achieved in real practice. The quality of the mesh also determines the accuracy of the results for CFD; in overly meshed cases, it leads to numerical error and instability, particularly in regions of high curvature, such as bends. High fidelity is computationally intensive, requiring strong hardware and longer computation times, as it considers complex geometries and transient flow. The phenomenon of interest here cannot be fully captured using simulations due to the complex underlying physics of events occurring in deposition, particularly the effects of fouling and multiphase interaction within bent pipes. Therefore, experimental validation is primarily necessary to validate against CFD results. Secondly, turbulence models are often inaccurate for high-turbulence flows because they fail to capture the precise dynamics of complex vortex structures, even with k-epsilon standard models or LES. Material properties, thermal effects, and wall roughness can cause errors if not properly accounted for. This, therefore, limits the validity of CFD results to the real world.

6.2. Future Scope

Future Scope in Analysis for CFD on Bent Pipes Computational Technique: The technology has advanced to form a robust platform for handling large computational volumes, thereby enabling a broad scope of applications. The advancement in Machine Learning and Artificial Intelligence holds the promise of significant enhancements and improvements in simulations, aiming to optimise accuracy while reducing computation time. High-performance computer-based clouds increase access while reducing costs, enabling the analysis of more extensive phenomena. Future areas of research would include multiphase flows and particle-laden fluids, as well as non-Newtonian fluids in bent pipes, to accommodate the varied requirements of different industries. Hence, with better turbulence models such as hybrid RANS-LES and DNS, a sharper understanding of the vortex formation and flow separation phenomenon in complex geometries would be revealed. It will be combined with experimental techniques, such as PIV, to facilitate the validation and better understanding of flow phenomena. Variable wall roughness, thermal gradients, and material properties in curved pipes may lead to new designs of heat exchangers, pipelines, and biomedical applications. Combining CFD with generative design tools enables engineers to optimize bent pipe systems for an efficient and cost-effective solution that meets specific needs. This places CFD at the heart of the academic and industrial challenges in fluid dynamics.

Acknowledgement: The authors would like to express their sincere gratitude to the University of Guyana, South America; Dhaanish Ahmed College of Engineering, India; and Bharath Institute of Higher Education and Research, India, for their valuable support and encouragement in completing this research work.

Data Availability Statement: The study utilizes a dataset that contains U-shaped pipe bends for Power Plants-related attributes, including traits that indicate phishing activity.

Funding Statement: No funding has been obtained to help prepare this manuscript and research work.

Conflicts of Interest Statement: No conflicts of interest have been declared by the authors. Citations and references are mentioned in the information used.

Ethics and Consent Statement: The consent was obtained from the organization and individual participants during data collection, and ethical approval and participant consent were received.

References

1. A. Aryal, I. Stricklin, M. Behzadirad, D. W. Branch, A. Siddiqui, and T. Busani, "High-quality dry etching of $LiNbO_3$ assisted by proton substitution through H_2-plasma surface treatment," *Nanomaterials*, vol. 12, no. 16, p. 2836, 2022.
2. R. L. Paldi, A. Aryal, M. Behzadirad, T. Busani, A. Siddiqui, and H. Wang, "Nanocomposite-seeded single-domain growth of lithium niobate thin films for photonic applications," *in Conf. Lasers Electro-Optics*, Optica Publishing Group, Washington, D.C., United States of America, 2021.
3. A. R. M. T. Islam, M. Abdullah-Al Mamun, M. Hasan, M. N. Aktar, M. N. Uddin, M. A. B. Siddique, M. H. Chowdhury, M. S. Islam, A. B. M. M. Bari, and A. M. Idris, "Optimizing coastal groundwater quality predictions: A novel data mining framework with cross-validation, bootstrapping, and entropy analysis," *J. Contam. Hydrol*., vol. 269, no. 2, p. 104480, 2025.
4. M. A. A. Mamun, A. R. M. T. Islam, M. N. Aktar, M. N. Uddin, M. S. Islam, S. C. Pal, A. Islam, A. B. M. M. Bari, A. M. Idris, and V. Senapathi, "Predicting groundwater phosphate levels in coastal multi-aquifers: A geostatistical and data-driven approach," *Sci. Total Environ*., vol. 953, no. 11, p. 176024, 2024.
5. M. N. Uddin, G. C. Saha, M. A. Hasanath, M. A. H. Badsha, M. H. Chowdhury, and A. R. M. T. Islam, "Hexavalent chromium removal from aqueous medium by ternary nanoadsorbent: A study of kinetics, equilibrium, and thermodynamic mechanism," *PLoS ONE*, vol. 18, no. 12, pp. 1-26, 2023.
6. M. H. Huang, L. Chen, Y. L. He, J. J. Cao, and W. Q. Tao, "A two-dimensional simulation method of the solar chimney power plant with a new radiation model for the collector," *Int. Commun. Heat Mass Transf.*, vol. 85, no. 7, pp. 100–106, 2017.
7. M. Abdelmohimen and S. A. Algarni, "Numerical investigation of solar chimney power plants performance for Saudi Arabia weather conditions," *Sustain. Cities Soc*., vol. 38, no. 4, pp. 1–8, 2018.
8. F. Attig-Bahar, M. Sahraoui, M. S. Guellouz, and S. Kaddeche, "Effect of the ground heat storage on solar chimney power plant performance in the South of Tunisia: Case of Tozeur," *Sol. Energy*, vol. 193, no. 11, pp. 545–555, 2019.
9. L. Zuo, N. Qu, Z. Liu, L. Ding, P. Dai, B. Xu, and Y. Yuan, "Performance study and economic analysis of wind supercharged solar chimney power plant," *Renew. Energy*, vol. 156, no. 8, pp. 837–850, 2020.
10. M. Tawalbeh, S. Mohammed, A. Alnaqbi, S. Alshehhi, and A. Al-Othman, "Analysis for hybrid photovoltaic/solar chimney seawater desalination plant: A CFD simulation in Sharjah, United Arab Emirates," *Renew. Energy*, vol. 202, no. 1, pp. 667–685, 2023.
11. P. M. Cuce, E. Cuce, D. K. Mandal, D. K. Gayen, M. Asif, A. Bouabidi, S. Alshahrani, C. Prakash, and M. E. M. Soudagar, "ANN and CFD driven research on main performance characteristics of solar chimney power plants: Impact of chimney and collector angle," *Case Stud. Therm. Eng., vol*. 60, no. 8, p. 104568, 2024.
12. L. Rezaei, S. Saeidi, A. Sápi, M. R. A. Senoukesh, G. Gróf, W. H. Chen, Z. Kónya, and J. J. Klemeš, "Efficiency improvement of the solar chimneys by insertion of hanging metallic tubes in the collector: Experiment and computational fluid dynamics simulation," *J. Clean. Prod*., vol. 415, no, 8, p. 137692, 2023.
13. A. Atia, S. Bouabdallah, B. Ghernaout, M. Teggar, and T. Benchatti, "Investigation of various absorber surface shapes for performance improvement of solar chimney power plant," *Appl. Therm. Eng*., vol. 235, no. 11, p. 121395, 2023.
14. C. B. Maia, F. V. Silva, V. L. Oliveira, and L. L. Kazmerski, "An overview of the use of solar chimneys for desalination," *Sol. Energy*, vol. 183, no. 5, pp. 83–95, 2019.
15. E. Cuce, H. Sen, and P. M. Cuce, "Numerical performance modelling of solar chimney power plants: Influence of chimney height for a pilot plant in Manzanares, Spain," *Sustain. Energy Technol. Assess.*, vol. 39, no. 6, p. 100704, 2020.