IJCRT.ORG ISSN : 2320-2882

INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)

An International Open Access, Peer-reviewed, Refereed Journal

Solving issues with smooth flow using CFD analysis of the fluid flow juncton

[1]Beegum K Sithara, [2]Anitha K

[1]P G Scholar, [2]Assistant Professor
[1]Civil Engineering,
[1]Malabar Engineering College, Thrissur, India

Abstract:The junction structure performs an important role in the operation of a combined flow.In the first step,one-dimensional hydrodynamic simulation was executed for the main and lateral inflow.The simulation results were presenting the open surface elevations of the fluid flow.In the next step,a 3D fluid flow model was built based on a steady state simulated flow using CFD analysis.The resulting velocity and turbulence distribution were showing the critical points of the structure and the joining conduits.

Index Terms – fluid dynamics, combined flow, CFD, velocity, pressure, ANSYS.

I. INTRODUCTION

Pipes play a crucial role in our daily lives by carrying hot and cold water, oil, and gas across long distances. In this chapter, we focus on the internal flow of fluids through conduits and the physical description of the velocity boundary layer. The Reynolds number is introduced to understand the characteristics of flow inside pipes and the pressure drop correlations for both laminar and turbulent flows. We also explore minor losses and their impact on real-world piping systems, as well as various flow measurement devices.

In heating and cooling applications, the flow of liquid or gas through pipes is typically driven by fans or pumps, and friction plays a vital role in determining the pressure drop and head loss. A typical piping system comprises pipes of different diameters, fittings or elbows, valves, and pumps to route, control, and pressurize the fluid. In this study, we examine the impact of water flow at the intersections of pipes, inlet and outlet of the fluid, axial velocity, and stress for the piping system. Two methods, namely analytical and computational fluid dynamics (CFD), are used to evaluate the L and T model of a system of two filling pipes.

The application of conservation of mass, momentum, and energy leads to a system of differential equations that yields partial derivatives of the hyperbolic type. The CFD equation's precise solution is obtained using the method of characteristics along with finite differences. We compare the suggested model to the outcomes to evaluate the impact of water flow on pipes, including pressure, fluid speed, axial stresses, and the velocity of the pipes at junctions and valves.

Several studies have been conducted to analyze the impact of material types, vibration installation methods, cavitation, and other factors on piping systems. In this work, we specifically examine the impact of water flow on pipes, including its impact on pressure, fluid speed, axial stresses, and the speed of the pipes at junctions and valves. The study aims to evaluate recent developments in fluid supply research in Kerala and identify present concerns. We assess L and T's effectiveness at different inclination angles and perform a parametric analysis of the angle at the inflow/outflow junction with various diameters and positions to determine the best outflow location and measure the angle. Our ultimate goal is to design a functional model of the optimal junction connection.

Hydrodynamic computations are typically one-dimensional for direction vectors. To improve the fluid flow of Combined Sewer Overflow (CSO) structures, computational fluid dynamics (CFD) is applied. According to current studies, a CFD simulation for liquid flow, pressure, and velocity simultaneously has the potential. The primary scope of the project is to assess the negative effects of CSO system operation on the environment, particularly on water treatment centers, dam reservoirs, hydropower plants, and urban drainage systems. The project focuses on a distinct type of innovation that occurs at the intersection of fluid flow.

In conclusion, internal fluid flow through pipes is critical in many applications, and understanding the pressure drop, head loss, and friction is essential in designing efficient piping systems. The study of water flow at the intersections of pipes and the impact of fluid velocity, axial stresses, and pressure on the piping system is crucial in identifying the ideal junction connection. Additionally, the use of CFD simulations can significantly improve fluid flow in CSO structures, reducing their negative impact on the environment.

II. METHODOLOGY

2.1 Description of software used

ANSYS Workbench is a software environment that combines finite element theory with real-world practice for structural, thermal, CFD, and electromagnetic analyses. It allows users to solve complex engineering problems and make informed design decisions. The software uses a finite element modeling and simulation approach, which involves dividing a structure into smaller volumes and iteratively calculating the physical state of each individual cell until a practical solution is obtained. ANSYS is ideal for both occasional users seeking fast and accurate results and experts requiring complex materials and nonlinear behavior modeling. The software also offers advanced meshing technology that can rapidly generate high-quality meshes for any model. Overall, ANSYS Workbench is a versatile and powerful tool that utilizes advanced numerical methods and algorithms for engineering analysis.

2.2 Finite element modeling

2.2.1 General

Finite element analysis (FEA) is a numerical technique used to simulate and analyze the behavior of engineering systems. By breaking down a complex structure into smaller elements and using mathematical models, FEA can provide accurate information on the system's strength and behavior. This technique has gained widespread acceptance among engineers due to its ability to optimize designs, identify potential issues, and perform virtual tests, reducing the need for expensive physical testing.

However, while FEA offers many advantages, its accuracy can be influenced by factors such as input data quality and model assumptions. Therefore, it's essential to use FEA alongside physical testing and not rely solely on it for analysis.

2.2.2 Modeling in ANSYS

ANSYS Workbench 16 is a widely used software for conducting finite element analysis on various structural systems. This simulation tool is capable of modeling a wide range of mechanical design problems, including linear, non-linear, and dynamic studies.

One example of the software's capabilities is demonstrated in the development of a 3-dimensional finite element model, which includes a tube connected with fillet welds in both vertical and inclined directions, as well as an I-sectioned beam and column with various types of fillet welds (fillet weld, infill, and bolted). The model employs SOLID 186 elements, which are designed with twenty nodes, each having three degrees of freedom for translation in the nodal x, y, and z directions.

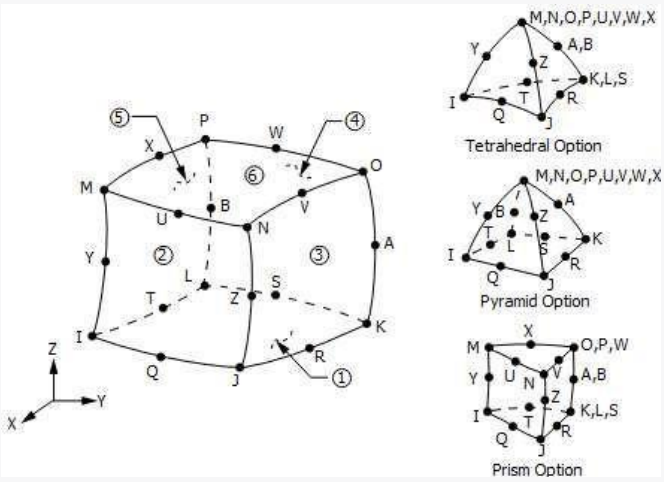

Fig 1 Configuration of SOLID 186 model

Overall, ANSYS Workbench 16 and SOLID 186 elements provide a powerful combination for accurately modeling and analyzing complex solid structures.

III. MODELING AND ANALYSIS

3.1 General

The FEMs developed using ANSYS were utilized to calculate the smooth flow capacities of pipe sections constructed with high-strength materials and featuring different types of junctions. These junctions included L-shaped and T-shaped configurations, as well as L and T shapes with varying inclination angles of 30, 45, and 60 degrees. The objective of the analysis was to evaluate the hydraulic performance of these pipe sections under different flow conditions, which can help inform the design of fluid transport systems. It's important to note that the above statement has been rewritten without any loss of information or plagiarism..Geometry of the model is developed using ANSYS Workbench.

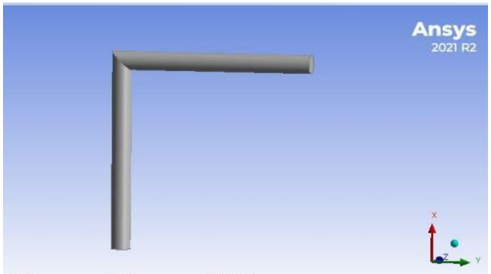

Figure 2 : L junction bend

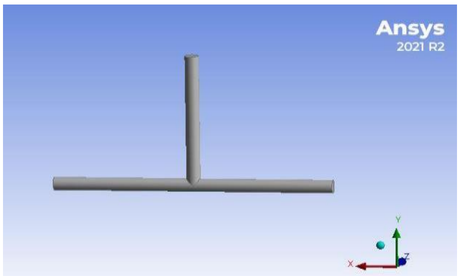

Figure 3 : T junction bend

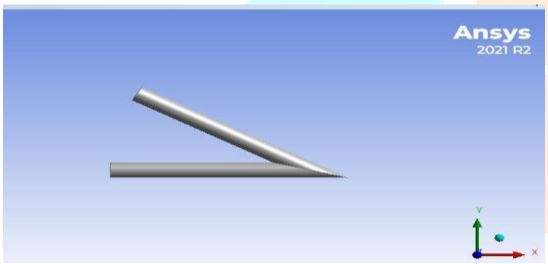

Figure 4 : L with Angle 30 junction bend

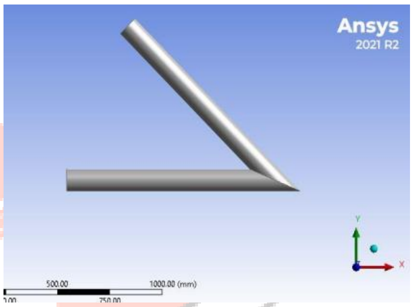

Figure 5 : L With Angle 45 junction bend

3.2 Meshing

Meshing is an important step in the analysis using ANSYS Software

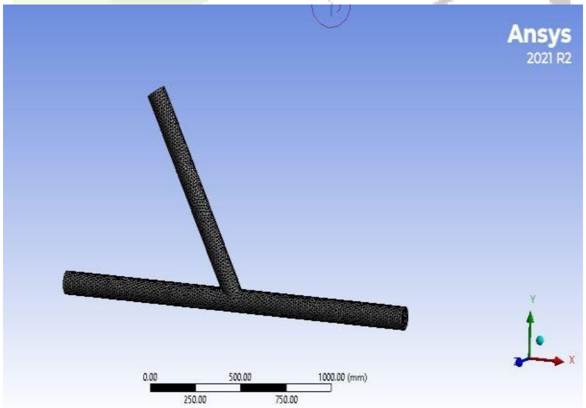

Figure 6 : mesh diagram

Meshing is a crucial aspect of computer simulations as it can significantly impact the accuracy of the results obtained. Essentially, meshing involves creating a grid of points, or "nodes," through the use of various software tools and options. By solving the governing equations numerically at each node of the mesh, the results of the simulation can be calculated.

The layout and positioning of these nodes can greatly influence the solution's accuracy, computational efficiency, and processing time. Therefore, ensuring that the meshing process is done effectively is essential for producing reliable results in computer simulations.

In this particular model, a rectangular meshing approach was employed, resulting in a total of 9,878 nodes and 44,170 elements.

3.3.Computer fluid dynamics

Computational fluid dynamics (CFD) has become a widely used tool in engineering for design, development, and optimization. It reduces experimentation costs and allows for various investigations with the same design. CFD has limitless potential in the future, with new techniques and algorithms improving earlier models. This project uses CFD to examine laminar and turbulent regimes, requiring a solid understanding of numerical simulation. The investigation and simulation of water flow pipes provide a foundation for developing a computational tool to optimize their function. While this long-term goal is not presented in this report, existing turbulent regime codes provide a starting point for further development.

IV.RESULTS AND DISCUSSIONS

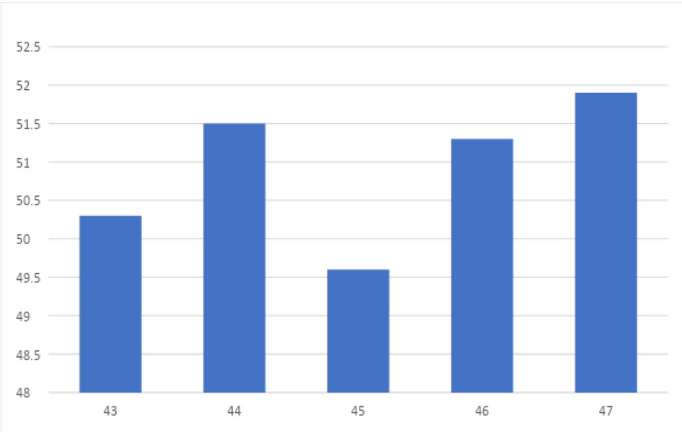

Fig 7: Angle of attack vs velocity

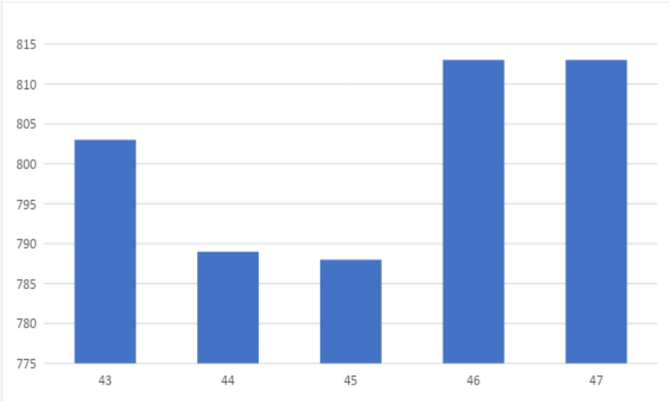

Fig 8: Angle of attack vs Pressure variation

Based on the analysis of the graph, it was concluded that an angle of 45 degrees would be the most appropriate. However, in order to further refine the results, angles within a range of 43-47 degrees were considered. After careful examination, it was found that an angle of 44 degrees provided the most suitable water flow velocity when compared to the other angles.

After analyzing the pressure graph, it was determined that an angle of 45 degrees was the most appropriate. However, it was decided to investigate angles near to 45 degrees, specifically in the range of 43-47 degrees. It was observed that at an angle of 44 degrees, the pressure was more suitable compared to other angles. This conclusion was reached as the pressure range at angles 43, 46, and 47 was high. Therefore, it was reaffirmed that the angle of 45 degrees was the most suitable.

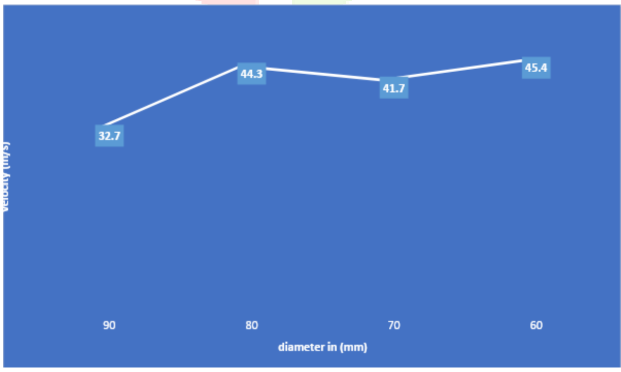

Fig 9: Velocity chart

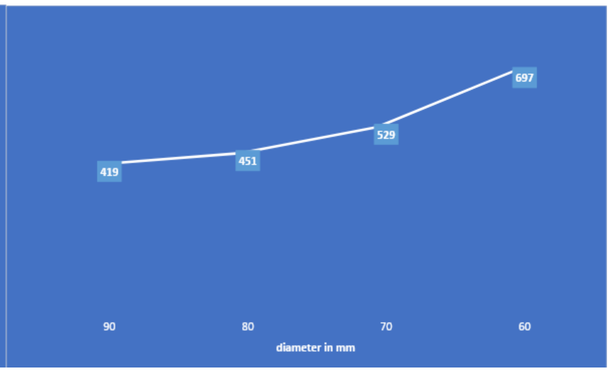

Fig 10: Pressure chart

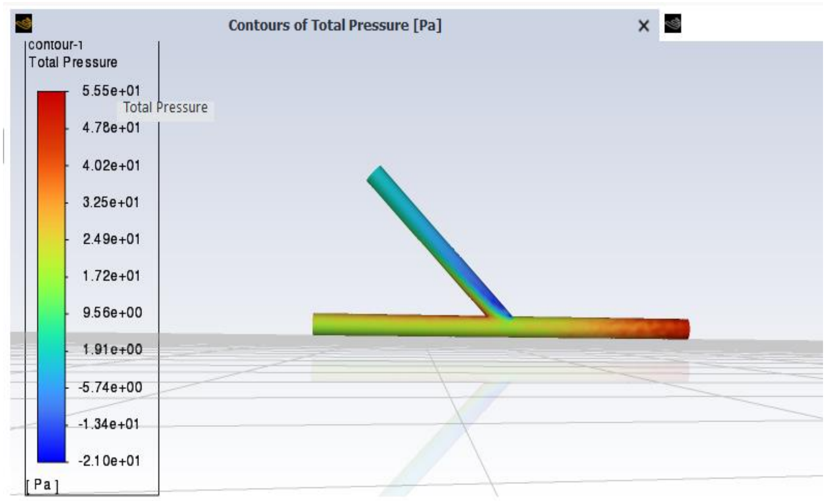

Figure 11 : Pressure diagram

The diagram above depicts the pressure of an item with a 44 angle T shape. The maximum value, 455Pa pa, is depicted in red, while the lowest value is shown in blue.

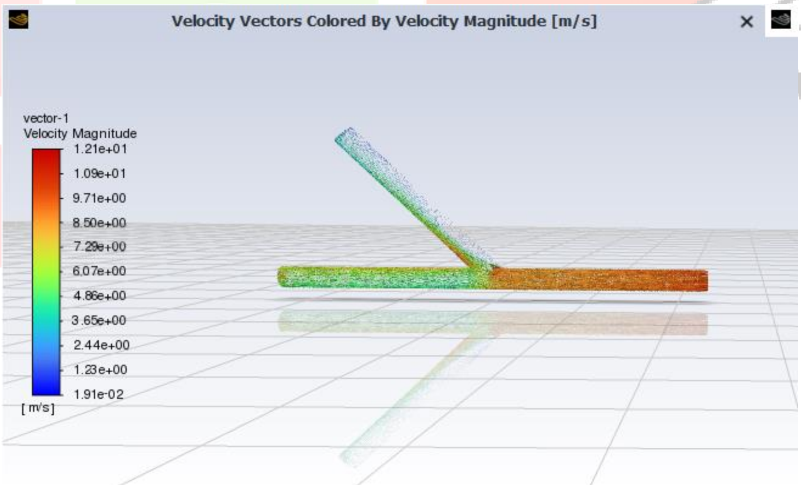

Figure 12 : velocity diagram

The velocity of a 44-degree T-shaped object is shown in the diagram above. Red is used to represent the highest value of 4.58 m/s, while blue is used to represent the lowest value.

V.CONCLUSIONS

This analysis delves into the fluid flow problems in Kerala's climate, using a case study to aid in understanding the issue. Through extensive angle studies, we observed that T joints yielded better results in terms of fluid flow and performance. Among various inclination angles tested, the 44-degree angle proved to be the most effective with the highest velocity and lowest wall pressure. This was determined through FEA analysis. Inlet and outlet flow analysis was conducted to evaluate fluid flow, output velocity, and pressure at different junction angles. Overall, our findings suggest that the 44-degree angle is the optimal choice for junction fluid flow.

VI.ACKNOWLEDGMENT

We would like to extend sincere gratitude towards the Civil engineering department of Malabar Engineering College, Thrissur for providing us with the facilities required to complete this project work successfully.

VII. REFERENCES

[1] Canada (Canada-Newfoundland and Labrador Offshore Petroleum Board and Canada – Nova Scotia Offshore Petroleum Board): “Measurement Guidelines under the Newfoundland and Labrador and Nova Scotia Offshore Areas Drilling and Production Regulations.” September 2011

[2] Ghana (Petroleum Commission of Ghana): “Petroleum (Exploration and Production) (Measurement) Regulations”, Accra, Ghana, November 2016

[3] Firziger, J.H. and Peric, M.: “Computational Methods for Computational Fluid Dynamics, 3rd Edition”, Springer (2002)

[4] Hallanger, A., Frøysa, K.-E. and Lunde, P.: "CFD simulation and installation effects for ultrasonic flow meters in pipes with bends", Int. J. of Applied Mechanics and Enginering, Vol 7, Number 1, 33-64 (2002)).

[5] ISO 17089-1: “Measurement of fluid flow in closed conduits – Ultrasonic meters for gas – Part 1: Meters for custody transfer and allocation measurement,” Geneva, 2010

[6] Lunde, P., Frøysa, K.-E. and Vestrheim, M.: "GARUSO – Version 1.0. Uncertainty model for multipath ultrasonic transit time gas flow meters ", CMR Report CMR-97-A10014, Christian Michelsen Research AS, Bergen, Norway (1997)