IJAREEIE

e-ISSN: 2278-8875
p-ISSN: 2320-3765

International Journal of Advanced Research in Electrical, Electronics and Instrumentation Engineering

Volume 13, Issue 4, April 2024

INTERNATIONAL STANDARD SERIAL NUMBER INDIA

Impact Factor: 8.317

9940 572 462 | 6381 907 438 | ijareeie@gmail.com | www.ijareeie.com

International Journal of Advanced Research in Electrical, Electronics and Instrumentation Engineering (IJAREEIE)

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

Design and Simulation of Dual Path U-shaped Coriolis Flow Meter in ANSYS

Kavin G, Vignesh S, Subhasan R, M.Shanmugavalli

Dept. of Instrumentation and Control Engineering, Saranathan College of Engineering, Trichy India

ABSTRACT: The design and simulation of a dual-path Coriolis flow meter using ANSYS software are investigated in this study. Coriolis flow meters are widely utilized in various industries for their accuracy and reliability in measuring flow rates. The dual-path configuration offers enhanced accuracy by mitigating the effects of fluid density variations and flow disturbances. The study focuses on the design process, including geometry definition and boundary condition specification, followed by detailed simulations conducted in ANSYS Fluent. Various parameters such as fluid density, flow velocity, and meter geometry are systematically varied to analyze their impact on meter performance. Through comprehensive simulation analyses, the study aims to optimize the design parameters to achieve optimal accuracy and reliability in flow rate measurement. The findings of this study provide valuable insights for the design and optimization of dual-path Coriolis flow meters, contributing to advancements in flow measurement technology.

KEYWORDS: Dual-path Coriolis flow meter, ANSYS software, Simulation, Flow measurement, Accuracy, Reliability, Fluid density variations, Flow disturbances, Parameter analysis, Optimization.

I. INTRODUCTION

In the field of industrial flow measurement, Coriolis flow meters have demonstrated their effectiveness in accurately determining the flow rate of liquids and gases. One key design variation of Coriolis flow meters is the dual-path U-shaped configuration, offering enhanced accuracy and stability in flow measurement applications. However, the precise simulation of such complex flow meter geometries poses significant challenges. This research aims to utilize the advanced capabilities of ANSYS software to simulate the fluid dynamics within a dual-path U-shaped Coriolis flow meter. By leveraging ANSYS' computational fluid dynamics tools, this study seeks to gain insights into the flow behavior, pressure distribution, and overall performance of the dual-path Coriolis flow meter. Ultimately, this research intends to enhance the understanding and optimization of dual-path U-shaped Coriolis flow meter designs through detailed numerical simulations.

The operating principle of Coriolis flow meters hinges on the inertia created by fluid flowing through an oscillating tube, causing the tube to twist proportionally to the mass flow rate. This "swinging" motion is induced by vibrating the tube(s) through which the fluid flows. The degree of twist is directly proportional to the mass flow rate of the fluid passing through the tube(s). Sensors and a Coriolis mass flow meter transmitter are employed to measure the twist and generate a linear flow signal.

Existing research has explored various aspects of Coriolis flow meter design and analysis. Studies have delved into modal analysis, resonance frequency evaluation, and vibration characteristics of different tube geometries. Moreover, micro-sized Coriolis flow meters have garnered attention for their potential applications, with research focusing on modeling and simulation of their behavior under varying input factors.

This paper aims to contribute to the body of knowledge by elucidating the process of designing a micro-sized dual-path Coriolis flow meter, specifically developed in U-shaped and straight tube configurations. Utilizing finite element analysis (FEA), computer-aided design (CAD), and ANSYS software, we simulate the fluid flow through the tubes, analyzing velocity and pressure contours. Our fluid medium for simulation is aerated water, providing insights into the performance of the dual-path U-shaped Coriolis flow meter.

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

II. METHODOLOGY

A. Description of dual path U-shaped coriolis flow meter

The flow meter consists of two parallel, symmetric U-shaped tubes through which the fluid flows. Each tube experiences an oscillating motion perpendicular to the direction of flow, induced by the Coriolis effect when the tubes are set into vibration. As the fluid flows through these oscillating tubes, it undergoes a Coriolis force, causing a phase shift between the oscillations of the two tubes. This phase shift is directly proportional to the mass flow rate of the fluid and is measured by sensors located at the inlet and outlet of the U-shaped tubes.

The symmetric design of the U-shaped tubes ensures that any external forces or disturbances, such as vibration or temperature changes, affect both tubes equally, minimizing their impact on flow measurement accuracy. Additionally, the parallel arrangement of the tubes allows for redundancy and improved reliability, as the flow meter can continue to operate even if one of the tubes becomes obstructed or damaged.

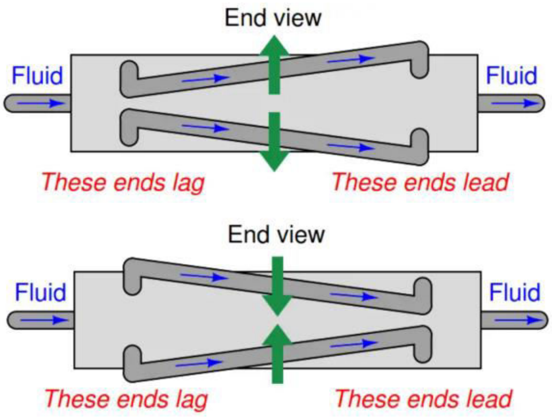


Figure 1

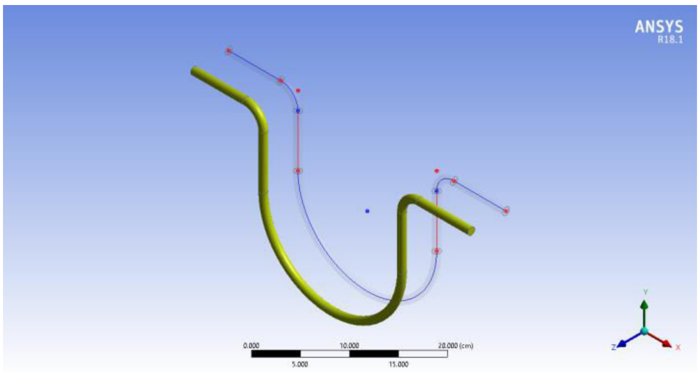


Figure 2

This figure represent the isometric view of design.

B. Flow parameters and geometry

Dual-path u-shaped Coriolis flow meters rely on specific flow parameters and dimensions for accurate measurement. These include the diameter and curvature radius of the u-shaped tubes, which affect the flow capacity, sensitivity, and natural frequency of vibration. The path separation distance between the two parallel tubes is critical for precise measurement of the Coriolis effect, influencing sensitivity and resolution. Tube length determines residence time and dynamic performance, while material properties such as density, elasticity, and thermal expansion impact mechanical behavior and temperature stability. Fluid properties such as density, viscosity, and temperature affect measurement accuracy, requiring calibration adjustments.

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

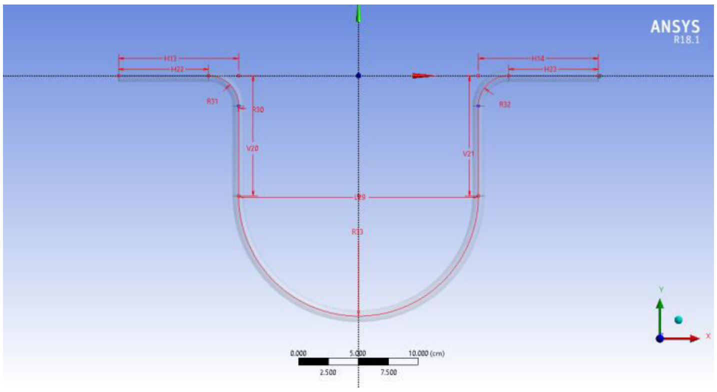


Figure 3

H13	10 cm
H14	10 cm
H22	7.5 cm
H23	7.5 cm
L29	20 cm
R30	2.5 cm
R31	2.5 cm
R32	2.5 cm
R33	10 cm
V20	10 cm
V21	10 cm

Table 1

C. Selection and boundary conditions

In the ANSYS simulation software, the selection of the appropriate geometry and the definition of accurate boundary conditions were crucial steps in ensuring a reliable and insightful analysis. The study began by selecting or creating the geometry of the dual-path U-shaped Coriolis flow meter in ANSYS, ensuring it accurately represented the physical flow meter. Efforts were made to simplify the geometry to reduce computational complexity while retaining essential features.

Once the geometry was prepared, boundary conditions were defined based on the operational conditions of the flow meter. Inlet and outlet boundaries were identified, and appropriate boundary conditions such as velocity profiles, mass flow rates, or pressure values were specified. Material properties of the fluid, including density, viscosity, and other relevant parameters, were defined. Additional boundary conditions, such as wall boundaries for solid surfaces and symmetry or periodic boundaries for symmetric geometries, were also considered as necessary.

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

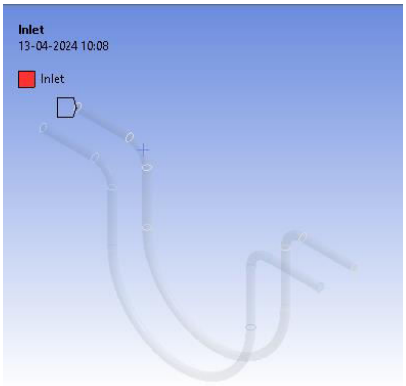


Figure 4

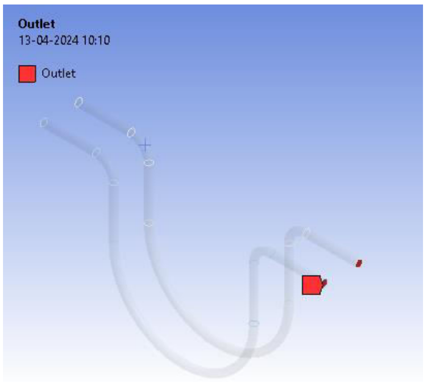


Figure 5

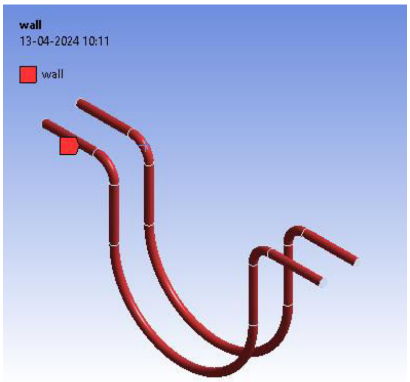


Figure 6

D. Simulation parameters

The simulation employs water with a consistent density of 998 kg/m³ and dynamic viscosity of 0.001 kg/ms. The fluid flow is assumed to be incompressible. Boundary conditions are established with a prescribed velocity at the inlet and zero initial gauge pressure. The velocity profile within the inlet pipe is considered fully developed. It is assumed that all walls adhere to a no-slip boundary condition. The simulation is conducted using ANSYS FLUENT 2018.1, a computational fluid dynamics (CFD) software. Employing the finite volume method and a first-order upwind difference scheme, the differential equations are transformed into algebraic equations. The coupling procedure is employed to calculate the pressure-velocity coupling of discrete equations. The computational grid, depicted in Figure 2, is unstructured and comprised of 6805 nodes and 30468 elements, featuring five boundary layer cells with an element size of 0.5mm.

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

Nodes	62624
Elements	55896

Table 2

Firstly, we carefully selected an appropriate solver within ANSYS Fluent, considering the specific characteristics of the flow and any relevant flow phenomena, such as turbulence.Next, we meticulously configured solver settings to ensure accurate convergence of the solution. This included defining convergence criteria to set the tolerance for solution convergence, ensuring reliable results.If the simulation involved transient flow phenomena, we defined the time step size to accurately capture temporal changes in flow behavior, balancing computational efficiency with accuracy. Physics models were activated based on the specific flow characteristics and phenomena of interest. This included turbulence models, multiphase flow models, and heat transfer models, among others.We reviewed and refined boundary conditions to ensure consistency with the selected solver and solver settings, optimizing simulation accuracy.Prior to initiating the simulation, we carefully initialized flow field variables, setting initial values for flow velocity, pressure, and other relevant variables to ensure a stable starting point. Once all simulation parameters were configured, we executed the simulation within ANSYS, monitoring the progress closely and making adjustments as necessary to ensure convergence and stability.

E. Mesh analysis

Meshing process to calculate the point where the loads are applied. Contact region (input and output) flow and fluid domain are created. SIMULATION and FLUID FLOW (FLUENT) are used to mesh the solid and fluid domain, respectively. A fluent mesh analysis in workbench describes when the files that are generated and used by fluent meshing and the cell zones change as in T-joint through venturi section with a fluent component system in workbench. Basic fluid flow analysis starting from an imported mesh it is also possible to bypass the geometry cell, and begin the simulation process by importing a mesh Right click the mesh cell and select mesh file In the dialogue open, browse to find u-tube mesh file and select open There are three basic steps to creating a mesh: create geometry for the meshing application.

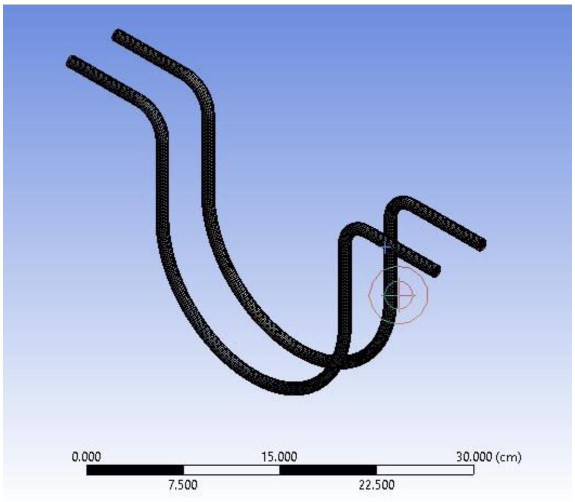


Figure 7

F. Solution and methods

Turbulent steady-state flow conditions were meticulously simulated using ANSYS Fluent, a renowned computational fluid dynamics software. The study focused on a robust solution methodology to accurately depict the intricacies of turbulent fluid dynamics. Utilizing the finite volume method, the governing equations, including the Navier-Stokes equations, were discretized to capture the turbulent flow phenomena within the computational domain. For turbulence modeling, the widely adopted k-epsilon model was employed, with parameters set to typical values of turbulent kinetic energy (k) at 0.01 m^2/s^2 and dissipation rate (epsilon) at 0.1 m^2/s^3. The pressure velocity coupling algorithm ensured convergence to a steady state, with a convergence criterion set at 10^-6 for the residual values. Through meticulous iterations, the simulation accurately depicted the turbulent flow behavior, shedding light on its complexities and providing valuable insights for various engineering applications.

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

G. Modal analysis

Modal analysis aims to determine the natural frequencies and mode shapes of a structure, in this case, the vibrating tubes of a dual-path U-shaped Coriolis flow meter. Set up the modal analysis by specifying the type of analysis (eigenvalue extraction), solver settings, and desired output parameters (natural frequencies and mode shapes). Analyze the modal analysis results to identify the natural frequencies and corresponding mode shapes of the vibrating tubes. Natural frequencies represent the frequencies at which the structure will vibrate in the absence of external forces, while mode shapes describe the spatial distribution of vibration across the structure. Identify any localized modes of vibration that may occur due to geometric discontinuities or irregularities in the structure. These localized modes can have significant effects on the overall vibration behavior and structural integrity.

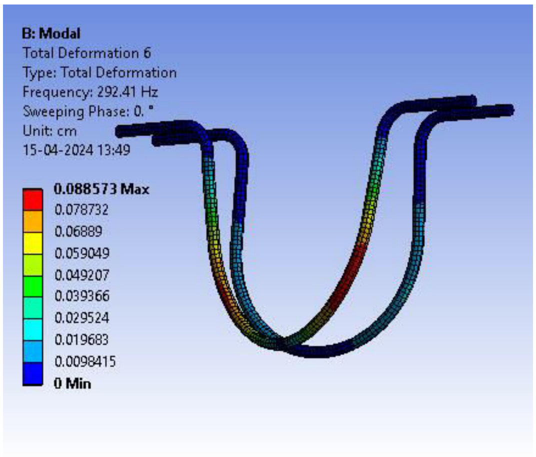


Figure 8

H. Setup and post process

Once the geometry and boundary conditions were set up, the simulation parameters were configured within ANSYS Fluent. This involved selecting an appropriate solver and setting solver settings such as convergence criteria and time step size. Any additional physics models or multiphase flow models required for the specific conditions of the flow meter were also activated.

After setting up the simulation, it was executed in ANSYS Fluent. The progress of the simulation was monitored, and convergence was ensured to obtain reliable results. If necessary, adjustments were made to solver settings or boundary conditions, and the simulation was ran.

In the post-processing phase, the results of the simulation were analyzed using ANSYS Fluent post-processing tools. Flow patterns, velocity contours, pressure distributions, and other relevant variables were visualized to gain insights into the flow behavior within the dual-path U-shaped Coriolis flow meter. Quantitative data such as flow rates, pressure drops, and turbulence characteristics were extracted for further analysis and comparison with experimental data or design specifications.

The solution gives the graph between the iteration and the flow of the pipe in continuity and in X, Y, Z velocity it is repeated for 500 iteration of the flow. Setup and solution procedure is followed by mesh file into ANSYS fluent and setup T-joint tube model.

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

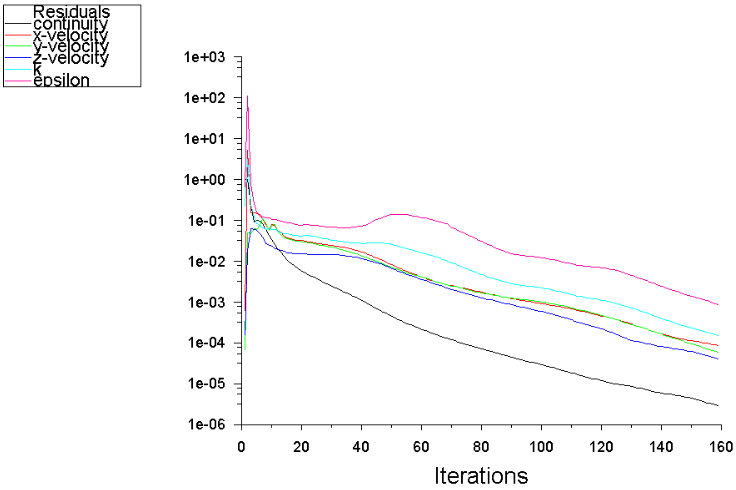


Table 3

III. RESULT AND DISCUSSION

Following the design, simulation, and post-processing stages of the T-joint Venturi meter project, the results unveiled significant insights into the fluid flow dynamics. Streamline visualization illustrated how fluid particles traversed the system, showcasing acceleration as they passed through the Venturi throat. Velocity distribution plots confirmed this acceleration, with higher velocities near the throat and lower velocities elsewhere. Pressure contour plots highlighted regions of high and low pressure, with the lowest pressure occurring at the Venturi throat, aligning with Bernoulli's principle. These findings were crucial for understanding energy losses and flow resistance within the system, aiding in its optimization. Additionally, comparison with theoretical and experimental data validated the simulation model, ensuring its reliability for future applications.

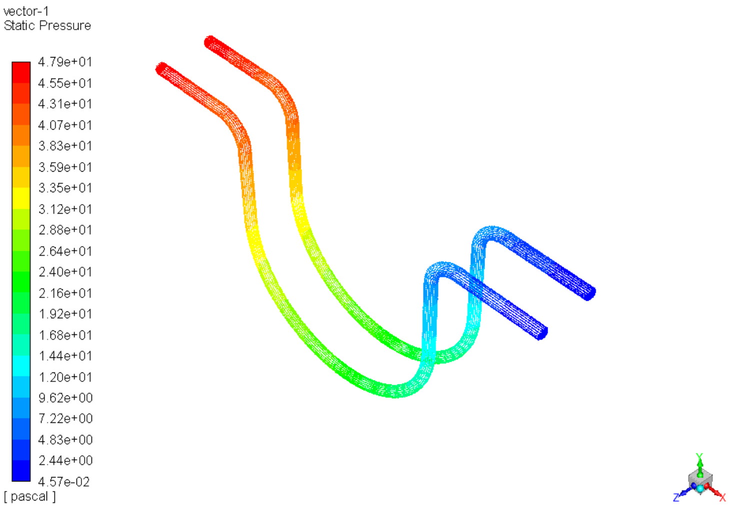


Figure 9

This figure represents the vectors of pressure distribution among the design.

| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

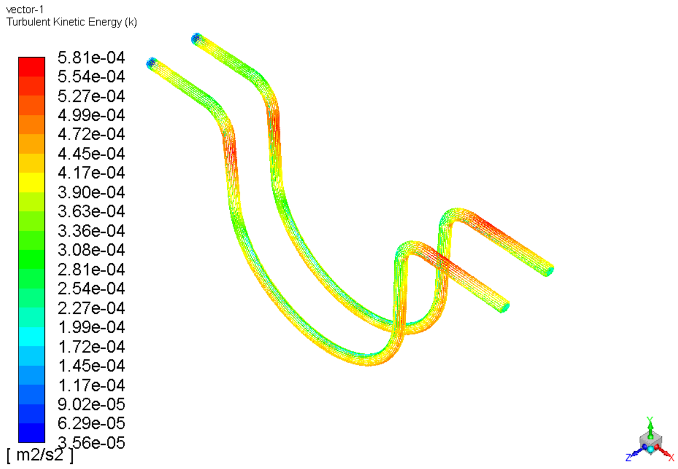


Figure 10

This figure represents the vectors of turbulence distribution among the design.

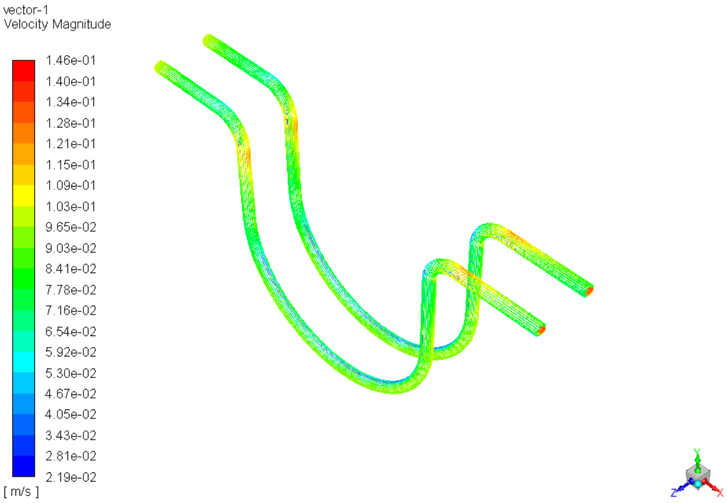


Figure 11

This figure represents the vectors of velocity distribution among the design.

IV. CONCLUSION

In fluid dynamics and computational engineering, our study focuses on understanding dual-path U-shaped Coriolis flow meters using ANSYS Fluent software (version 2018.1). Through our analysis, we explore how these meters measure fluid flow accurately.

We examine how fluid moves inside the meter and how factors like fluid properties, flow speed, and meter shape affect its performance. Using ANSYS Fluent, we simulate different scenarios to see how the meter responds in various conditions. This helps us find ways to make the meter work better and give more precise measurements.

Additionally, we look at how the meter's structure behaves when it vibrates. By studying these vibrations with ANSYS, we can ensure the meter stays stable and reliable in real-world situations.

Our research lays the groundwork for future studies to improve flow meter design and performance. By using computational tools like ANSYS Fluent, we aim to push the boundaries of engineering and enhance the technology used in industries like oil and gas, pharmaceuticals, and food processing.

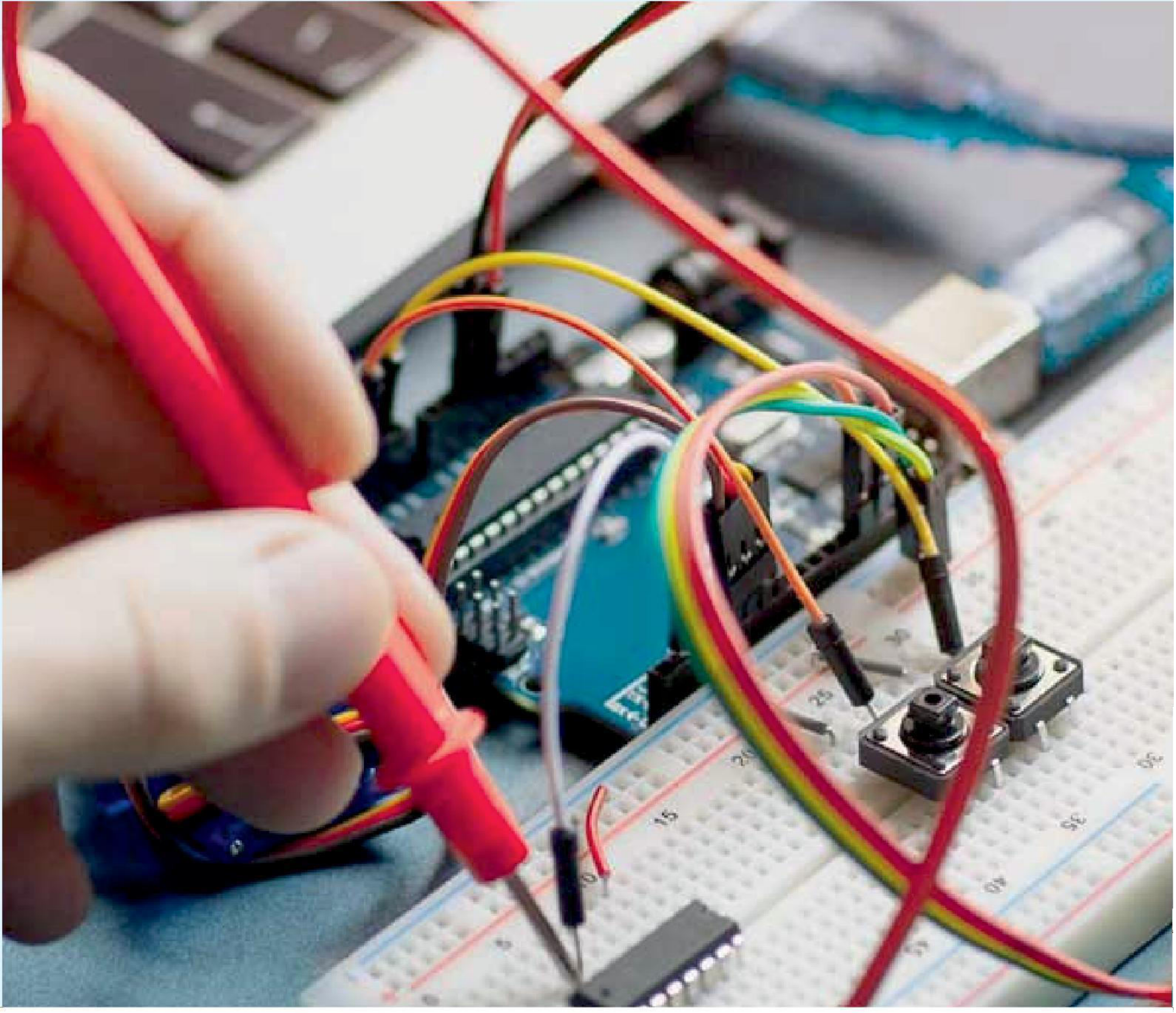
| e-ISSN: 2278 – 8875, p-ISSN: 2320 – 3765| www.ijareeie.com | Impact Factor: 8.317|| A Monthly Peer Reviewed & Referred Journal |

||Volume 13, Issue 4, April 2024||

| DOI:10.15662/IJAREEIE.2024.1304032 |

REFERENCES

[1] Bovonik, G., N.Mole., B.Stok, 1. Bajsic (2005). "Coupled Finite Volume/Finite Element Modelling of the straight tube Coriolis flowmeter" Journal of fluids and structures, 20, pp. 785-800.
[2] Sharma Satish C., Pravin P. Patil, Major Ashish Vasudev, S. C. Jain, "Performance Evaluation of an Indigenously Designed Copper (U)tube Coriolis Mass flow sensors, Measurement", 43(9) (2010) pp. 1165-1172. [3] Martin Anklin, Wolfgang Drahm, and Alfred Rieder. "Coriolis mass flowmeters: Overview of the current state of the art and latest research. Flow Measurement and Instrumentation", 17(6):317-323, December 2006.
[4] M. P. Henry, C. Clark, M. Duta, R. Cheesewright, and M. Tombs. "Response of a coriolis mass flow meter to step changes in low rate". Flow Measurement and Instrumentation, 14(3):109- 118, June 2003.
[5] A. Mehendale, J.C. Lötters and J.M. Zwikker, "Mass flow meter of the Coriolis type", EP Patent 1,719,982, 2006.
[6] H. Raszillier and V. Raszillier." Dimensional and symmetry analysisof coriolis mass flow meters". Flow Measurement and Instrumentation, 2(3):180-184, July 1991.
[7] LuoRongmo, Wu Jian, "Fluid-structure coupling analysis and simulation of viscosity effecton Coriolis mass flowmeter", APCOM & ISCM11-14th December, 2013, Singapore.
[8] Pravin PPatil, Satish C. Sharma, Vipul Paliwal, Ashwani Kumar, "ANN Modelling of Cu Type Omega Vibration Based Mass Flow Sensor", Procedia Technology, 14 260-265. (2014), pp.
[9] H. Raszillier and F. Durst. Coriolis effect in mass flow metering. Archive of Applied Mechanics, 61:192-214, 1991.
[10] R.M. Saravanan, Christ W. Raj and M. Shanmugavalli, "Design and Simulation of Coriolis Flow Tube in Meso and Micro Level to Determine Its Resonant Frequency". Middle-East Journal of Scientific Research 23 (Sensing. Signal Processing and Security): 23 (2015) pp. 239-242,ISSN 1990-9233. DOI: 10.5829/idosi.mejsr.2015.23.ssps.101.



International Journal of Advanced Research

in Electrical, Electronics and Instrumentation Engineering

9940 572 462 6381 907 438 ijareeie@gmail.com