e-ISSN:2582-7219

INTERNATIONAL JOURNAL OF MULTIDISCIPLINARY RESEARCH IN SCIENCE, ENGINEERING AND TECHNOLOGY

Volume 7, Issue 4, April 2024

INTERNATIONAL STANDARD SERIAL NUMBER INDIA

Impact Factor: 7.521

6381 907 438 6381 907 438 ijmrset@gmail.com www.ijmrset.com

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

Airflow Optimization on Circular Duct by CFD Analysis

J. Sangilimuthukumar, S. Antony Raphel Vino, B. Babu, C. Basil, A. Immanuel

Assistant Professor, Department of Mechanical Engineering, Francis Xavier Engineering College, Tirunelveli, Tamilnadu, India

UG Student, Department of Mechanical Engineering, Francis Xavier Engineering College, Tirunelveli, Tamilnadu, India

UG Student, Department of Mechanical Engineering, Francis Xavier Engineering College, Tirunelveli, Tamilnadu, India

UG Student, Department of Mechanical Engineering, Francis Xavier Engineering College, Tirunelveli, Tamilnadu, India

UG Student, Department of Mechanical Engineering, Francis Xavier Engineering College, Tirunelveli, Tamilnadu, India

ABSTRACT : The connection between an intake fan and a duct must be designed in such a way that it minimizes the energy waste due to singularity losses. As a result, the questions of which radius of curvature to use and if guide vanes have to be included need to be answered. In that case, the variables such as the number, upstream and downstream, duct length, radius of curvature and width of the duct need to be defined. Computational fluid dynamics was used to determine the airflow in the circular duct of 2.26 m diameter and to optimize the best performing configuration. This study investigates the optimization of airflow in circular ducts through Computational Fluid Dynamics (CFD) analysisThe objective of this research is to explore different strategies for optimizing airflow in circular ducts using CFD simulations. The literature review reveals key considerations in circular duct design, including the effect of inlet and outlet configurations, turbulence, and pressure drop. Challenges in modelling and validating CFD simulations in circular ducts are also highlighted. Through CFD analysis, the study examines the impact of different parameters on airflow patterns and pressure distribution within circular ducts. Results demonstrate the effectiveness of optimization strategies in enhancing airflow efficiency and reducing energy consumption. Case studies and real-world applications further illustrate the practical implications of optimized circular duct designs. The findings of this research contribute to the understanding of airflow optimization in circular ducts and provide valuable insights for engineers and designers seeking to improve the performance of ventilation and air conditioning systems. Future research directions include the development of advanced CFD modeling techniques and experimental validation methods to enhance the accuracy and reliability ofairflow optimization in circular ducts. Finally, the airflow is optimized from oneinlet and four outlets.

KEY WORDS : Circular duct , No surface contact , Less friction , Easy installation

I . INTRODUCTION

Circular ducts play a fundamental role in numerous engineering applications, ranging from heating, ventilation, and air conditioning (HVAC) systems to automotive air intake systems. The efficient flow of air within these ducts is paramount for ensuring optimal performance, energy efficiency, and indoor air quality. However, achieving and maintaining optimal airflow in circular ducts poses significant challenges due to complex fluid dynamics and various design constraints. Computational Fluid Dynamics (CFD) analysis offers a powerful toolset for investigating and optimizing airflow characteristicsin circular ducts.

The primary objective of this study is to explore the methodologies and strategies employed in **optimizing airflow within circular ducts through CFD analysis**. By leveraging computational simulations, researchers and engineers can gain valuable insights into the flow behavior, pressure distribution, and turbulence effects within circular ducts, enabling them to design more efficient and cost-effective duct systems.

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

The introduction of this research begins by providing an overview of the importance of airflow optimization in circular ducts, emphasizing its implications for energy conservation, system performance, and occupant comfort. Subsequently, key challenges and considerations in circular duct design are discussed, including geometric constraints, inlet/outlet configurations, and flow regime transitions. The limitations of traditional design approaches and the need for advanced computational techniques are highlighted to underscore the significance of CFD analysis in addressing these challenge. Furthermore, this introduction outlines the structure of the subsequent sections, which include a comprehensive literature review, methodology overview, results discussion, and conclusion. Through a systematic investigation of existing research and case studies, this study aim to elucidate the current state of knowledge regarding airflow optimization in circular ducts and propose novel approaches for enhancing duct system performance Overall, this research endeavors to contribute to the advancement of HVAC engineering and fluid dynamics by providing actionable insights and practical recommendations for optimizing airflow in circular ducts. By harnessing the capabilities of CFD analysis, engineers and designers can overcome the complexities associated with duct design and achieve superior performance in various engineering applications.

II. CONCEPT OF CFD

Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical analysis and data structures to analyze and solve problems that involve fluid flows. CFD analysis plays an important role in the structure safety of the components in automobiles CFD is an advanced numerical method that involves simulating the flow of fluids and their interaction with solid surfaces. Engineers and researchers use CFD to analyze complex fluid flow problems that are not easily solved using experimental techniques or traditional mathematics. CFD is used to optimize heat transfer in everything from automobile engine designs to coffeemachine designs. It is also vital to a myriad of other simulation types mentioned on this list (like HVAC) where heat transfer is an integral component of a product's core function. CFD is also used to predict the drag, lift, noise, structural and thermal loads, combustion., etc.

Computers are used to perform the calculations required to simulate the free-stream flow of the fluid, and the interaction of the fluid (liquids and gases) with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved, and are often required to solve the largest and most complex problems. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial validation of such software is typically performed using experimental apparatus such as wind tunnels. In addition, previously performed analytical or empirical analysis of a particular problem can be used for comparison. A final validation is often performed using full-scale testing, such as flight tests.

CFD is applied to a wide range of research and engineering problems in many fields of study and industries, including aerodynamics and aerospace analysis, hypersonics, weather simulation, natural science, and environmental engineering, industrial system design and analysis, biological engineering, fluid flows and heat transfer, engine and combustion analysis, and visual effects for film and games, etc….

III. STEPS INVOLVE IN CFD ANALYSIS

Computational Fluid Dynamics (CFD) involves several key steps to simulate and analyze fluid flow and heat transfer phenomena within a computational domain. Here are the typical steps involved in a CFD analysis:

3.1 Problem Definition and Geometry Acquisition:

Define the objectives and scope of the analysis, including the physical phenomena to be simulated and the desired outputs. Acquire or generate the geometry of the computational domain, which may include solid boundaries, fluid volumes, and inlet/outlet boundaries.

3.2 Mesh Generation:

Discretize the geometry into small, finite-sized elements or cells to create a computational mesh. The mesh quality and resolution significantly impact theaccuracy and convergence of the CFD solution.

3.3 Boundary Condition Specification:

Define boundary conditions at the inlet, outlet, and solid boundaries, including velocity profiles, temperature distributions, pressure values, and wall conditions. Boundary conditions should accurately represent the physical system being simulated.

3.4 Solver Setup and Solution:

Configure solver settings, including convergence criteria, discretization schemes, and turbulence models. Initiate the

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

solution process to iteratively solve the discretized equations until convergence is achieved. Monitor solution convergence and adjust solver settings if necessary.

3.5 Post-Processing and Visualization:

Analyse the CFD results, including velocity profiles, pressure distributions, temperature fields, and other relevant quantities. Perform post- processing operations such as contour plotting, vector visualization, streamline tracing, and quantitative analysis. Interpret the simulation results to gain insights into the fluid flow behavior and validate against experimental data if available.

3.6 Validation and Verification:

Validate the CFD results by comparing them with experimental data or analytical solutions. Verify the accuracy and reliability of the simulation by assessing mesh convergence, sensitivity analysis, and benchmarking against known solutions.

3.7 Documentation and Reporting:

Interpret the CFD results in the context of the problem objectives and provide insights into the fluid flow behavior. Prepare reports, presentations, or technical documents summarizing the analysis methodology, results, conclusions, and recommendations.

By following these steps systematically, CFD practitioners can effectively simulate and analyze fluid flow phenomena in a wide range of engineering applications, aiding in design optimization, performance prediction, and problem-solving.

IV. MOTIVATION

Optimizing airflow within circular ducts can lead to significant improvements in energy efficiency and system performance. By reducing pressure drop, minimizing turbulence, and optimizing flow distribution, engineers can enhance the overall efficiency of HVAC systems, ventilation systems, and other fluid transport systems. Energy consumption associated with fluid flow through ducts represents a substantial operational cost for many industrial and commercial applications.

By optimizing airflow patterns and reducing energy losses, organizations can achieve cost savings in terms of reduced energy consumption and maintenance expenses. In HVAC systems, optimized airflow contributes to improved indoor air quality, thermal comfort, and occupant well-being. Proper ventilation and airflow distribution help maintain uniform temperature levels, prevent stagnation zones, and minimize the risk of indoor air pollutants, allergens, and contaminants.

In existing duct systems, airflow optimization can help address performance issues, such as uneven distribution, air leakage, and pressure imbalances. By diagnosing flow problems through CFD analysis, engineers can identify root causes, implement corrective measures, and improve system reliability and effectiveness. Conducting airflow optimization studies using CFD analysis contributes to the advancement of knowledge and understanding of fluid dynamics phenomena in circular ducts. Research findings can inform the development of improved design guidelines, optimization methodologies, and simulation techniques for future engineering applications.

V. PROPOSED THEME

Circular ducts are a common component in HVAC systems, offering advantages such as reduced air resistance, efficient airflow distribution, and space-saving design. However, ensuring optimal airflow within circular ducts requires careful design, installation, and maintenance. This theme explores strategies for maximizing airflow efficiency in circular duct systems to enhance HVAC performance and energy efficiency. Circular ducts offer lower air resistance compared to rectangular ducts due to their streamlined shape, resulting in reduced energy consumption and fan power requirements. Proper sizing and layout of circular ducts are essential to minimize pressure drops and maintain uniform airflow distribution throughout the system. Design considerations such as duct diameter, length, and curvature influence airflow characteristics and must be optimized to achieve the desired performance. Computational fluid dynamics (CFD) simulations provide valuable insights into airflow behavior within circular duct systems. By modeling airflow patterns, velocity profiles, and pressure distributions, engineers can identify potential bottlenecks, optimize duct configurations, and enhance system performance. CFD analysis enables iterative design improvements and facilitates

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

the prediction of airflow characteristics under varying operating conditions.

The choice of duct material affects airflow efficiency, durability, and maintenance requirements. Smooth, corrosion-resistant materials such as galvanized steel, aluminum, or fiberglass are commonly used for circular duct construction to minimize airflow resistance and prevent contamination. Proper insulation and sealing are essential to reduce heat loss, condensation, and air leakage, optimizing system performance and indoor air quality. Implementing dampers, diffusers, and variable air volume (VAV) controls enables precise airflow regulation within circular duct systems. By adjusting airflow rates based on occupancy, temperature, and air quality requirements, VAV systems optimize energy usage and maintain comfort conditions while reducing operational costs. Balancing airflow distribution across duct branches ensures uniform air delivery to each zone, enhancing occupant comfort and system efficiency.

Regular inspection, cleaning, and maintenance of circular ducts are essential to prevent obstructions, airflow restrictions, and system degradation. Routine airflow measurements, pressure testing, and duct leakage assessments help identify potential issues and ensure optimal system performance. Incorporating smart sensors and monitoring technologies enables real-time airflow monitoring, fault detection, and predictive maintenance, maximizing system reliability and longevity. Optimizing airflow in circular duct systems contributes to overall HVAC energy efficiency and sustainability goals. By minimizing energy consumption, reducing carbon emissions, and enhancing indoor air quality, efficient duct design and operation support environmentally responsible building practices. Integrating energy recovery ventilation (ERV) systems further enhances efficiency by recovering heat or coolness from exhaust air, reducing HVAC load and operating costs.

Optimization of airflow in circular duct systems is essential for maximizing HVAC performance, energy efficiency, and occupant comfort. Through careful design, simulation, material selection, and control strategies, engineers can achieve efficient airflow distribution, minimize energy consumption, and ensure sustainable building operation. Continuous monitoring, maintenance, and innovation in duct design and technology will drive further advancements in airflow optimization and HVAC system performance.

VI. METHODOLOGY

❖ **Problem Definition:**

Clearly define the objectives of the airflow optimization study, including improving airflow uniformity, minimizing pressure losses, or reducing energy consumption. Identify the specific circular duct system or component to be analyzed and optimized.

❖ **Geometry Preparation:**

Create a detailed geometric model of the circular duct system using CAD software. Ensure the geometry accurately represents the physical system, including duct dimensions, bends, junctions, and any other relevant features. Simplify the geometry as needed while retaining key flow characteristics.

❖ **Mesh Generation:**

Generate a computational mesh for the geometric model using meshing software. Choose an appropriate mesh density to capture flow features and resolve boundary layers adequately. Verify mesh quality and refine the mesh near walls and regions of interest to improve solution accuracy.

❖ **Boundary Conditions:**

Define boundary conditions for the CFD simulation, including inlet velocity, pressure outlets, and wall conditions. Specify operating conditions such as flow rate, temperature, and turbulence intensity. Ensure boundary conditions accurately represent the real-world operating environment of the circular duct system.

❖ **Solver Setup:**

Select an appropriate CFD solver such as ANSYS Fluent or ANSYS CFX for airflow analysis. Choose turbulence models and numerical schemes suitable .

for simulating turbulent flow in circular ducts. Configure solver settingsincluding convergence criteria, solution methods, and time step size.

❖ **Simulation Run:**

Run the CFD simulation to analyze airflow behavior within the circular duct system. Monitor convergence and solution

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

stability throughout the simulation process. Store relevant flow field data such as velocity vectors, pressure distributions, and turbulence parameters for post-processing.

❖ **Analysis and Optimization:**
Analyze simulation results to identify areas of flow separation, recirculation, or high pressure drop. Evaluate key performance metrics such as airflow uniformity, pressure losses, and energy consumption. Identify opportunities for design optimization to improve airflow efficiency and system performance.

❖ **Design Modification:**
Propose design modifications aimed at optimizing airflow within the circular duct system. Explore design variations such as changes in duct diameter, curvature, or smoothness to minimize pressure losses and improve flow uniformity. Implement design changes in the geometric model for further simulation.

❖ **Documentation and Reporting:**
Document the entire procedure, including problem definition, geometry preparation, simulation setup, analysis, optimization process, and results. Prepare a comprehensive report summarizing the findings, recommendations, and implications of the airflow optimization study.

VII. DESIGN

7.1 DESIGN

Designing a circular duct involves selecting appropriate materials, determining the required airflow parameters based on heating or cooling loads, calculating the duct diameter to maintain efficient air velocity, and planning the layout to minimize pressure losses. Factors such as insulation, sealing, and support must be considered to prevent heat transfer, air leakage, and structural issues. Careful attention to detail in design, including insulation thickness, sealing methods, and proper duct support, ensures optimal performance, energy efficiency, and compliance with industry standards and regulations. Regular testing and commissioning validate the design and confirm that the circular duct system meets specified performance requirements while delivering consistent airflow throughout the HVAC system.

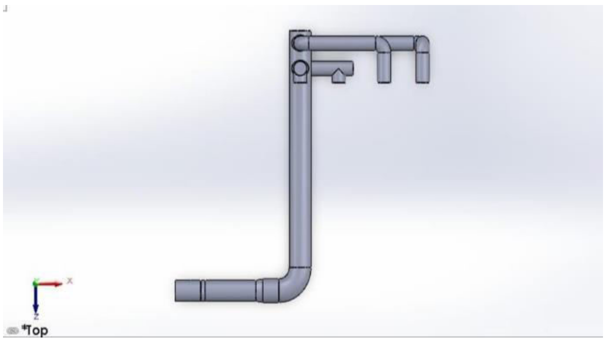


Fig 1: Side view

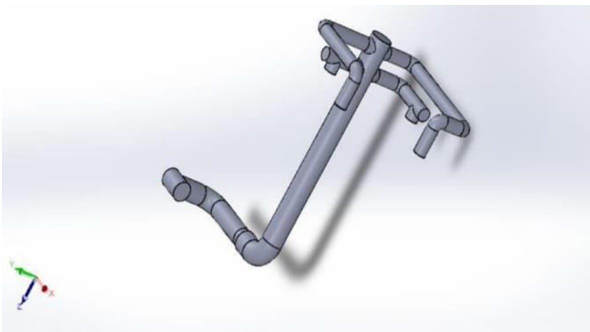


Fig 2: Overall view

7.2 ANALYSIS

Specifications

Inlet BC : 374000kg/hr

Inlet Temp : 196degC

Outlet Pressure : 0.07 kg/cm2

Fluid : Flue gas

International Journal Of Multidisciplinary Research in Science, Engineering and Technology (IJMRSET)

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

Geometry and Mesh

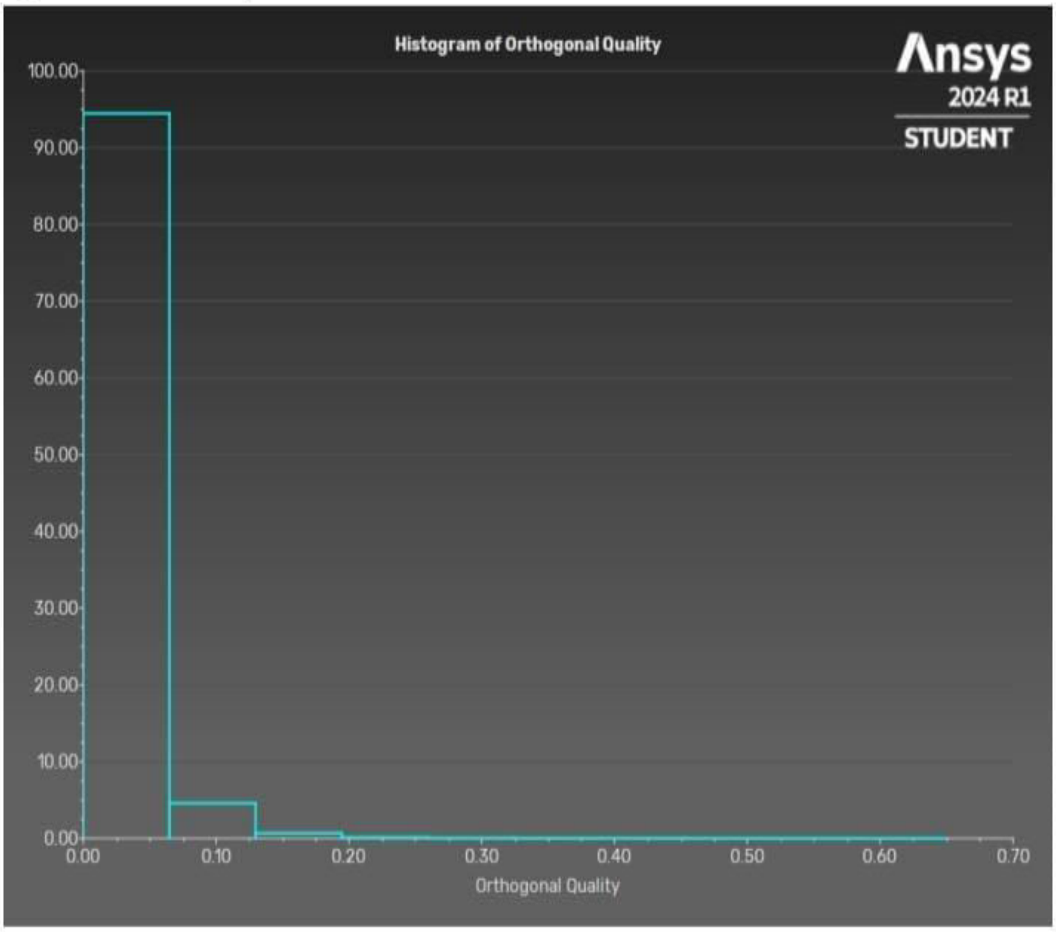
Mesh Size

Cells	Faces	Nodes
132566	309413	44374

Mesh Quality

Name	Type	Min Orthogonal Quality	Max Aspect Ratio
part_1	Tet Cell	0	20415746

Orthogonal Quality



International Journal Of Multidisciplinary Research in Science, Engineering and Technology (IJMRSET)

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

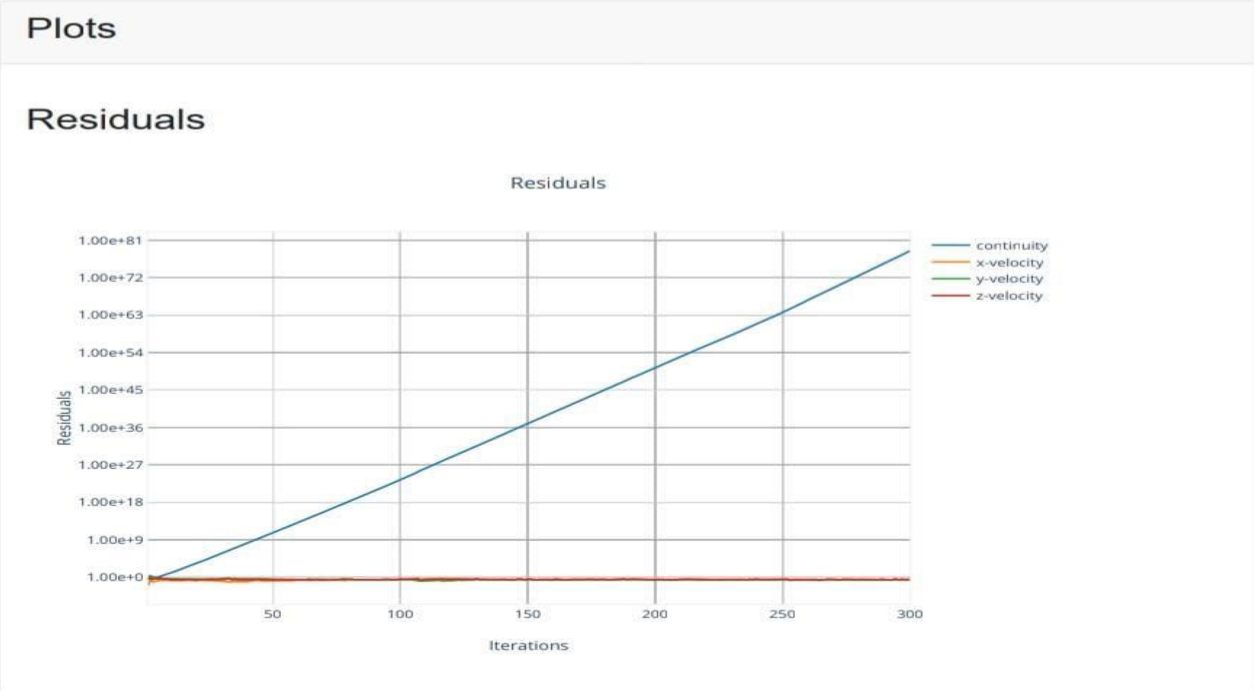
Boundary Conditions

− Inlet	
− inlet1	
Reference Frame	Absolute
Mass Flow Specification Method	Mass Flow Rate
Mass Flow Rate [kg/s]	10.39
Supersonic/Initial Gauge Pressure [Pa]	0
Direction Specification Method	Direction Vector
Coordinate System	Cartesian (X, Y, Z)
Component of Flow Direction (x,y,z)	(1, 0, 0)
− Outlet	
− outlet1	
Backflow Reference Frame	Absolute
Gauge Pressure [Pa]	6864.655
Pressure Profile Multiplier	1
Backflow Direction Specification Method	Direction Vector
Coordinate System	Cartesian (X, Y, Z)
Component of Flow Direction (x,y,z)	(1, 0, 0)
Backflow Pressure Specification	Total Pressure
Build artificial walls to prevent reverse flow?	no
Radial Equilibrium Pressure Distribution	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no
− outlet2	
Backflow Reference Frame	Absolute
Gauge Pressure [Pa]	6864.65
Pressure Profile Multiplier	1
Backflow Direction Specification Method	Direction Vector
Coordinate System	Cartesian (X, Y, Z)
Component of Flow Direction (x,y,z)	(1, 0, 0)
Backflow Pressure Specification	Total Pressure
Build artificial walls to prevent reverse flow?	no
Radial Equilibrium Pressure Distribution	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no
− outlet3	
Backflow Reference Frame	Absolute
Gauge Pressure [Pa]	6864.655
Pressure Profile Multiplier	1
Backflow Direction Specification Method	Direction Vector
Coordinate System	Cartesian (X, Y, Z)
Component of Flow Direction (x,y,z)	(1, 0, 0)
Backflow Pressure Specification	Total Pressure
Build artificial walls to prevent reverse flow?	no
Radial Equilibrium Pressure Distribution	no

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |



7.3 RESULTS

a) **Mass flow rate in outlet:**

"Surface Integral Report"

Mass Flow Rate	[kg/hr]
inlet1	374000
outlet1	2.6009597e+12
outlet2	3.1390566e+12
outlet3	-2.6473582e+12
outlet4	2.7981135e+12
Net	5.890772e+12

| ISSN: 2582-7219 | www.ijmrset.com | Impact Factor: 7.521| Monthly Peer Reviewed & Referred Journal |

| Volume 7, Issue 4, April 2024 |

| DOI:10.15680/IJMRSET.2024.0704077 |

b) Pressure drop:

```
                    "Surface Integral Report"

           Area-Weighted Average
                 Static Pressure                    [Pa]
-------------------------------- --------------------
                          inlet1       -1.2183942e+184
                         outlet1              6864.655
                         outlet2               6864.65
                         outlet3       -8.8159428e+178
                         outlet4       -6.8123368e+179
                            wall       -1.7618524e+183
                ---------------- --------------------
                             Net       -1.7600599e+183
```

VIII. CONCLUSION

In conclusion, airflow optimization on circular ducts through CFD analysis presents a robust methodology for enhancing HVAC system performance and energy efficiency. By leveraging computational tools, engineers can gain valuable insights into airflow characteristics, pressure distributions, and velocity profiles within the duct system. Through iterative simulations and design modifications, areas of inefficiency can be identified and addressed, leading to improvements in airflow uniformity, pressure losses, and energy consumption.

The streamlined shape of circular ducts, coupled with their space efficiency and ease of installation, makes them an ideal choice for HVAC applications. By employing CFD analysis for airflow optimization, engineers can achieve optimal system designs that meet performance requirements, improve occupant comfort, and reduce operational costs. Ultimately, integrating CFD analysis into the design process empowers engineers to make informed decisions and drive innovation in HVAC system design and operation.

REFERENCES

1. Dipankar De, Tarun Pal, Santanu Bandyopadhyay, "Spiral baffle design duct with CFD analysis", International Journal of Air and Technology, Vol. 35, No. 2, June 2017, pp. 378-383.
2. Kvenumadhav, Sudhanshu Kumar, Chandrashekar, "Performance Analysis of Different ducts Design Using Cfd Simulation", IJRAET, Volume 6, Issue 2 OCT 2017.
3. Mohammed Irshad, Mohammed Kaushar, G. Rajmohan, "Airflow pressure drop in HVAC ductwork", IJESC, Volume 7 Issue No.4, 2017.
4. Q. Miao, Z.Wang, A.Wang, "A Failure Analysis of Air Based on Modal Calculation and Structural Interaction Simulation", International Journal of Analysis and Manufacturing, Vol. 5, No. 3, August 2017.
5. S.Hasu, J. Govardhan, Satyanarayana Rao, "Air moving through gradually expanding duct", Anveshana's International Journal of Research in Engineeringand Applied Sciences, Volume 1, Issue 3, 2016.
6. K Ashok Reddy, "A Review of Resistance to change the flow of air system", IJSDR, Volume 1, Issue 5, May 2016.
7. J. BalaBhaskara Rao, V. Ramachandra Raju, "A simplified ducted air system", Rao and Raju International Journal of Mechanical and MaterialsEngineering Volume 11, Issue 6, DOI 10.1186/s40712-016-0059-x.
8. Mayank Bhola, Vinod Kumar, Satyendra, " The ventilation ducts with convex or concave wall cavity by ANSYS FLUENT", International Journal of Engineering Research & Technology (IJERT), Vol. 4 Issue 04, April-2015.
9. Pankaj D. Lad, G. Taji, Baban Londhe, "Airflow velocity distribution in three-way ducts", International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE), Volume: 2 Issue: 10, ISSN: 2349-7947, 2015.
10. Roshan.V. Marode, J. Keche, "Bends in ducting and effect reduces airflow", International Journal of Engineering Research and General Science Volume 3, Issue 1, January-February, 2011

INNO SPACE
SJIF Scientific Journal Impact Factor

INTERNATIONAL STANDARD SERIAL NUMBER INDIA

निस्केयर
NISCAIR

INTERNATIONAL JOURNAL OF
MULTIDISCIPLINARY RESEARCH
IN SCIENCE, ENGINEERING AND TECHNOLOGY

| Mobile No: +91-6381907438 | Whatsapp: +91-6381907438 | ijmrset@gmail.com |

www.ijmrset.com