JSRT JOURNAL
PAGES: 193-203
9/14/23
JOURNAL OF SCIENTIFIC RESEARCH AND TECHNOLOGY(JSRT) VOLUME-1 ISSUE-6 SEPTEMBER
Registered under MSME Government of India ISSN: 2583-8660

CFD Analysis On The Flow Of Gas In Economizer With Different Duct Geometry

Md Ashfaq[1], Prof. Anand Kumar S. Malipatil[2], Prof. Rajesh Holkar[3]

[1]*Student, Department of Mechanical Engineering, Visvesvaraya Technological University, Belagavi, India*
mdashfaq292@gmail.com
[2]*Professor, Department of Mechanical Engineering, Visvesvaraya Technological University, Belagavi, India*
malipatil@vtu.ac.in
[3]*Professor, Department of Mechanical Engineering, Visvesvaraya Technological University, Belagavi, India*
holkar.rajesh@gmail.com

ABSTRACT

The use of Computational Fluid Dynamics (CFD) analysis is prevalent in studying the flow of gases in various systems. In this study, the focus is on analyzing the flow of gas in economizers with different duct geometries. Economizers are heatexchangers used for recovering heat within exhaust gases prior being released into environment. Different duct geometries have been proposed for enhancing efficacy of economizers, but their effectiveness is often not well understood.

The CFD analysis involves solving the governing equations of fluid flowing & heat transmission inthe economizer. The different duct geometries are modeled and analyzed to determine their impact on flowing of gas & heat transmission efficacy of economizer. The study also aims to identify the optimal duct geometry that can enhance efficacy of economizer.

Energy saving and efficiency are the key issues in power generating not just with perspective of fuel consumption, even with protection of global environment. Flue gas ducts are the major parts of oil-fired power plant, which have been utilized into exhaust flue gases from boiler. This paper presents an approach for the economizer duct design. CFD analysis is used to compare new economizer channel design with traditional strategies. The most economical solution of this problem seems to distribute gas flow uniformly at inlet of economizer by usingvanes. So that efficient heat transmission can be obtained In the present work commercial softwareANASYS is utilised in 3D simulation using its inbuilt K-ϵReliable model.

Keywords: **Economizer, Economizer duct, vanes, k-ϵ Reliable model**

I. INTRODUCTION

In industrial processes, energy efficiency plays a crucial role in reducing operational costs and minimizing environmental impact. Economizers, which are heatexchangers designed to recover wastage heat by flue gases, are widely employed to improve energy efficiency in various applications. The optimization of economizer designing is vital to enhance its performance and maximize heat transfer efficiency. Onecrucial aspect of economizer designis the duct geometry, which significantly affects the flow characteristics of gas within the economizer.

Previous research has shown which duct shape significantly affects heat exchange efficiency, even those of economizers. Yet, there is dearth of in-depth studies that examine whether varying duct geometries affect gas flowing features of economizers. This project aims to fill this research gap by conducting a systematic analysis of different duct geometries and their effect on the gas flow behavior.

Understanding the flow behavior and heat transfer characteristics within an economizeris essential for optimizing its performance. CFD analysis offers a cost-effective and efficient means to simulate and analyze complex flow phenomena, providing valuable insights into the impact of different design parameters. By employing CFD techniques, we can study the gas flow behavior in a controlled and virtual environment, enabling us to evaluate multiple designconfigurations before physical prototypes are built. The subsequent sections of our article takes a look in methodology employed for theCFDanalysis, the variations in duct geometry considered, the results obtained, and the subsequent discussion and analysis. The project concludes with a summary of the findings and recommendations for further research in the field of economizer design optimization.

The outcomes of this project will contribute to the knowledge base on economizer design optimization. By identifying the most efficient duct geometry configurations, engineersanddesigners can make informed decisions to improve heat transfer effectiveness while considering the pressure drop constraints. Ultimately, this research will assist in the development of more energy-efficient systems, reducing operational costs and enhancing sustainability in industrial processes.

The primary objective of this project is to conduct a computational fluid dynamics (CFD) analysis onthe flow of gas in an economizer with different duct geometries. By varying the duct geometry, such ascross-sectional shape, length, and angle of inclination, we can explorethe impact of these variations on the gas flow patterns, heat transfer effectiveness, and pressure drop within the economizer

II. LITERATURE REVIEW

Numerous investigators investigated effect of duct geometry variations on the flowcharacteristics and heat transfer performance of economizers.

1. Sharma, A., Bansal, V., & Bhargava, A. (2015). Conducteda CFD analysis on a plate fin-tube economizer and found that changing the duct geometry resulted in altered flow patterns and heat transfer distributions. They reported that a curved ductdesign improved heat transfer effectiveness compared to a straight duct design.

2. Kim, J.H., Moon, H., & Choi, S. (2017). CFD simulations were performed to investigated impact of duct shape upon efficacy of a finned-tube economizer. They compared circular, elliptical, and rectangular duct shapes and noticed as circular and elliptical ducts exhibited high heat transmission efficacy because of better flowdistribution and reduced pressuredrop.

3. Kumar, S., Bhatti, M. S., & Kumar, S. (2018). Explored the influence of duct length on the performance of an economizer using CFD simulations. They analyzedthe gas flow behavior and heat transfercharacteristics for varying lengths of the ductand concluded that an optimal duct length existed for maximizing heat transfer effectiveness while minimizing pressure drop.

4. Li, Z., Zhang, C., Wang, Z., Li, Q., & Cui, Y. (2020). CFD analysis was employed to investigate the effects of both duct shape and angle of inclination on the gas flow behavior in an economizer.They compared circular and rectangular ducts with differentinclination angles and found thatthe rectangular duct with a specific inclination angle exhibited superior heat transfer performance and lower pressure drop.

5. Gao, G., Du, X., Li, H., & Zhang, B. (2016). utilized RANS (Reynolds-averaged Navier-Stokes) turbulence modeling to investigate the impact of duct geometry variations on the flowing & heat transmission aspects of an economizer. They observed that changes into duct shape affected the turbulence intensity and velocityprofiles, consequently influencing the heat transferperformance.

III. METHODOLOGY

3.1 Simulation Set Up And Data Input:

ANSYS uses finite volume approach for separating controlling equation. SIMPLE -algorithm allows for pressure-velocity coupling to occur.Each scenario undergoes analysis without using grid. K-ε

In ANSYS K-**ε**, a trustworthy model, we do all simulations. In this case, the mass velocity & temp of the flue gases serve as the boundary conditions. We suppose all every particle are inside tube transverse to its axis with their terminal velocities. An ANSYS schematic model of inside tubing of real flue gas duct. Ductwork plans serve as the basis for the geometry's development. For meshing, we use ANSYS meshing.Both the inlet area & volume were meshes using the tetra elements. It is possible to still improve mesh via adaptation through use of velocity gradient. Cell counts change depending upon geometrical parameters & need for grid-free solution. We initially simulate duct's geometry with straight vanes, then modify geometry with curved vanes.

Design dataset utilised

Table1: Flue gas parameter

Inlet temperature	**337°C**
Mass flow rate	**15.62 kg/s**
Specific heat kJ/kg K	**1.12**
Density (kg/m³)	**1.337**
Viscosity kg/mh	**0.101**
Thermal Conductivity kW/mK	**0.00046**

Table 2: Water parameter

Inlet temperature	**120°C**
Mass flow rate	**6.31 kg/s**
Density (kg/m³)	**913**
Specific heat kJ/kg K	**0.42**
Thermal Conductivity	**1 kcal/m-hr-°c**

Table 3: Geometry of economizer

Tube	**38.1 mm**
Tube thickness	**3.66 mm**
Longitudinal Pitch	**100 mm**
Number of tubes wide	**26**
Number of tubes deep	**22**

Table 4: Tube Material Property

Conductivity	**47 w/m°c**
Density	**7850 kg/m3**

Table 5: Boundary conditions

Flue gas	
Press outlet	**0 Gauge**
Vel_inlet	**12 m/s**
Wall	**No slip & escape**
Default_interior	**Fluid (Flue gas)**

Water	
Press_outlet	**0 Gauge**
Wall	**No slip & escape**
Default interior	**Fluid (water)**

Table 6: Solver adjustment

Solver	**Segregated**
Formulation	**implicit**
Time	**Steady**
Velocity Formulation	**absolute**
Gradient option	**Cell based**

Table 7: Solution constraints

Equations	**Flow**
Pressure	**0.3**
Density	**1**
Body Force	**1**
Momentum	**0.7**

Table 8: Discretizing

Pressure	**Standard**
Momentum	**Second order**
Turbulent kinetic energy	**Second order**
Turbulent dissipation rate	**Second order**

3.2 Numerical Simulation:

Since fluid carriers are so ubiquitous in research & manufacturing, they naturally pique our curiosity. Fluid carrier's usage of ANSYS for numerical modeling of fluid transmission necessitates modeling of continuous phase (fluid) as well as discrete phase while their interaction. In Eulerian modeling, continuous phase includes both liquid and gas. Eulerian and Lagrangian perspectives on the discrete phase -flue gas are both possible. As a result, 2 separate approaches have emerged: so-called approach & Eulerian-Lagrangian approach. For every point in numerical domain, Eulerian-Eulerian method determines concentration & velocity fields of fluid.Either one-fluid & 2-fluid formulation are viable options for use with Eulerian-Eulerian approach.

The latter method treats each stage as 2 interpenetrating fluids that interact through their respective interfaces. But in the singular fluid formulation, momentum preserving equation is not used. It is common practice to use an equation based on algebra for fluid slip velocity to calculate velocity of flue gas. The direction of mass flow determines whether connection is one way or two. Intensity of coupling among phases is used to represent their interaction.

K-ε Turbulence model

It's among most widely used CFD code & gold standard in field. Also proved to be steady & numerically vigorous & has established regimeof predictive capability. Within ANSYS the k-ε turbulence model utilises fine mesh.

Meshing

The most important part into CFD simulating is discretization of geometry. Generally hexahedral & tetra meshes are utilised for CFD codes. Hexahedral mesh gives better results, but meshing is very difficult. Here tetrahedral mesh generated in ANASYS meshing software.

IV. RESULTS AND DISCUSSION

Optimization of Economiser:

There are 26 parallel 70mm long strips near duct exit. The investigation focuses on a single, deep row of 22 tubes. The study takes into account feed water inlet temperatures of 120"e & flue gas intake temperatures of 337"c. Economizer modules have feed water entering by bottom & flue gas passing through tubes at top. Using the K-model in CFD, we can analyze both the flow of flue gas over the tubes & flow of water through the tubes simultaneously. Even though 165°C is the target temperature at economizer's output, we're only reaching that temperature following nineteenth row. There must be a 45°c increase in actual temperature. However, we are seeing a 54°C increase in temperature.We may save money by eliminating 78 economizer tubes using this strategy.

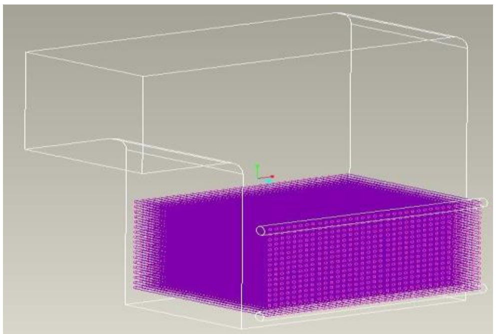

Fig. 1 : Geometry of inlet duct & Economizer

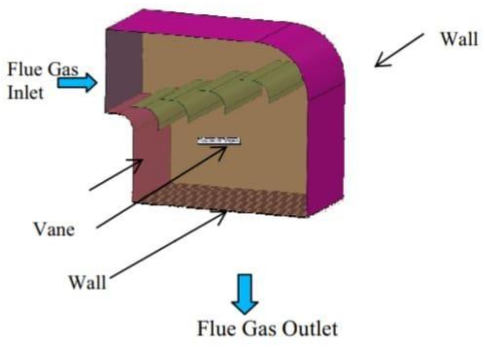

Fig 2 : Inlet duct geometry with vanes

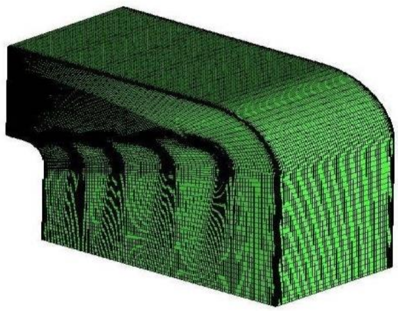

Fig. 3 : Mesh for ducts & vanes

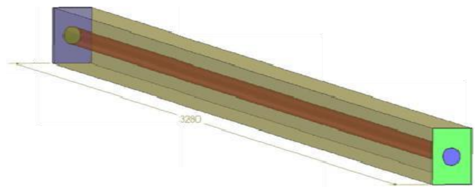

Fig.4 : Tube geometry

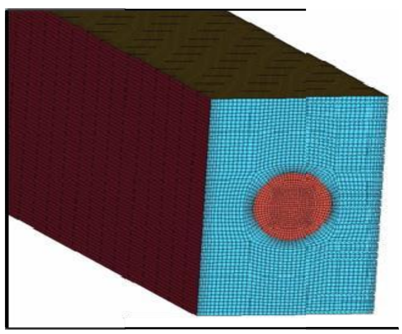

Fig. 5 : Tube mesh

PAGES: 193-203
9/14/23
VOLUME-1 ISSUE-6 SEPTEMBER
ISSN: 2583-8660

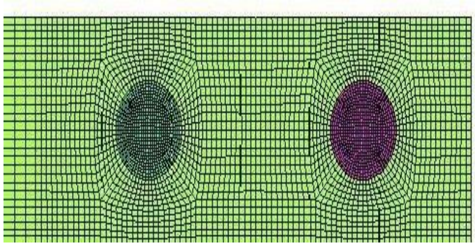

Fig. 6: Meshing between two tubes

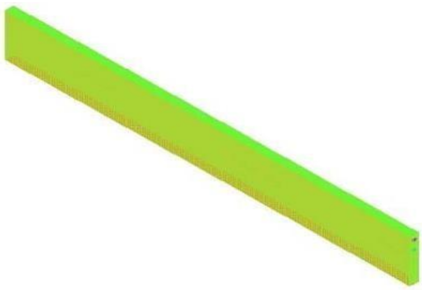

Fig. 7: Meshing of complete tube

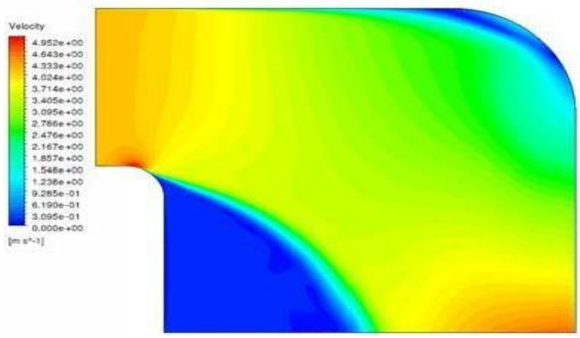

Fig. 8: Velocity Contour without vanes

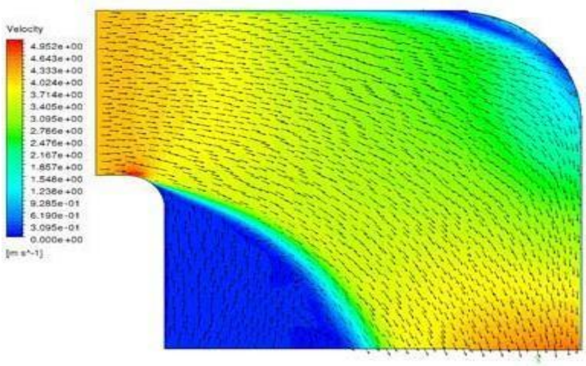

Fig 9: Velocity Vector with no vanes

PAGES: 193-203
9/14/23
JOURNAL OF SCIENTIFIC RESEARCH AND TECHNOLOGY(JSRT) VOLUME-1 ISSUE-6 SEPTEMBER
Registered under MSME Government of India ISSN: 2583-8660

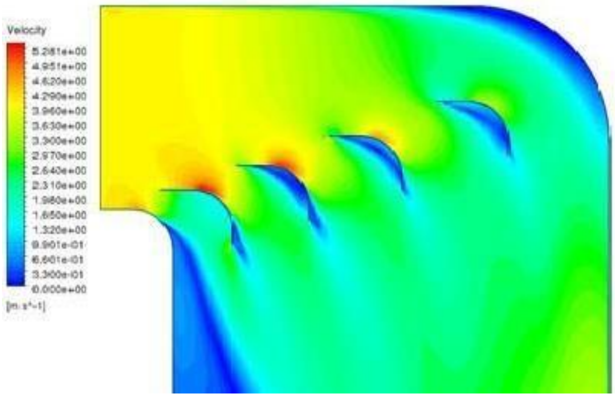

Fig 10 : Velocity Contour with vanes

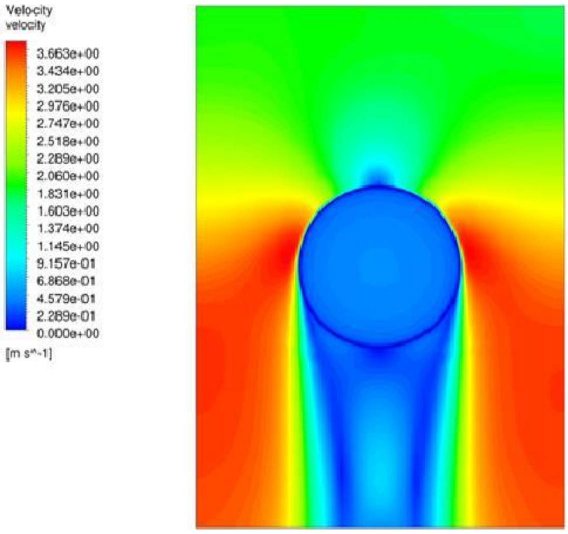

Fig. 11 : Velocity Contour inlet of tube

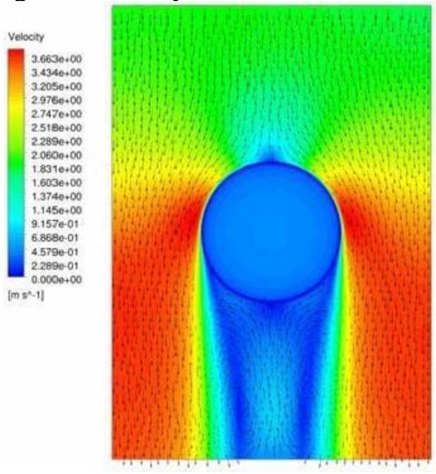

Fig. 12 : Velocity Vector

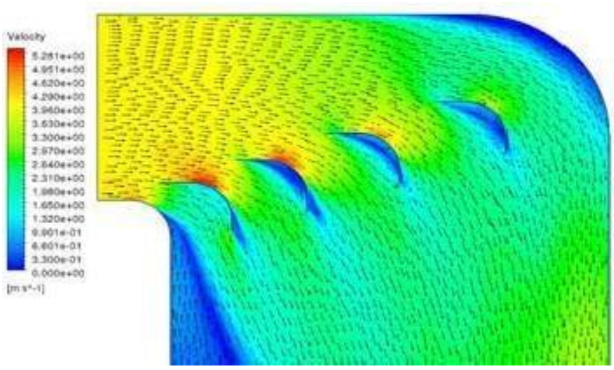

Fig.13 : Velocity Vector with va nes

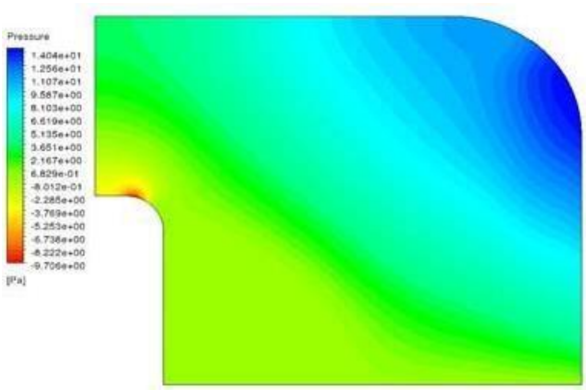

Fig.14 : Pressure Co n t o u r without vanes

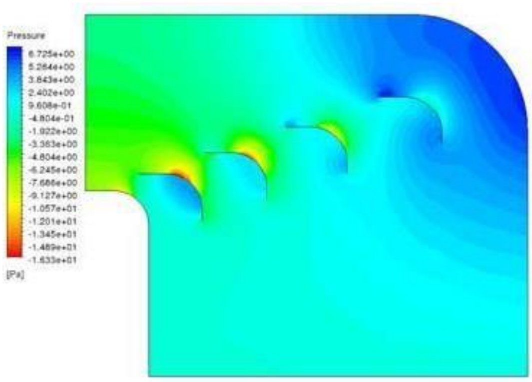

Fig. 15 : Pressure contour with vanes

JOURNAL OF SCIENTIFIC RESEARCH AND TECHNOLOGY(JSRT)
Registered under MSME Government of India
PAGES: 193-203
9/14/23
VOLUME-1 ISSUE-6 SEPTEMBER
ISSN: 2583-8660

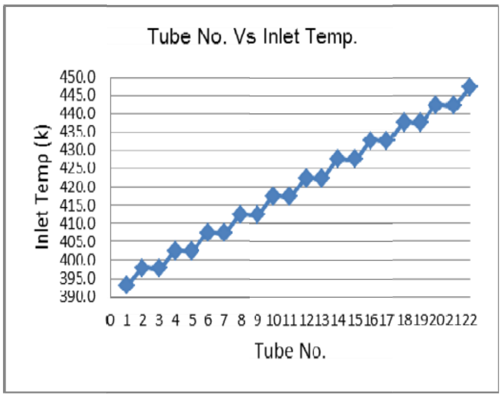

Fig. 16 : Tube inlet temperature

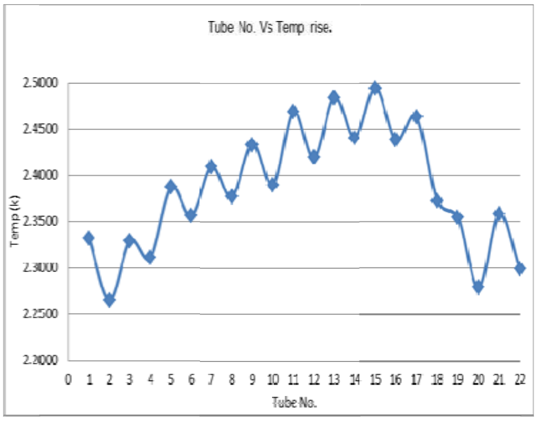

Fig. 17 : Temperature rise cross tube

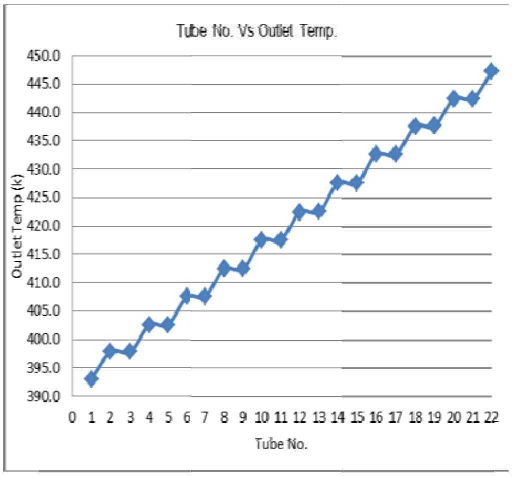

Fig. 18 : Tube outlet temperature

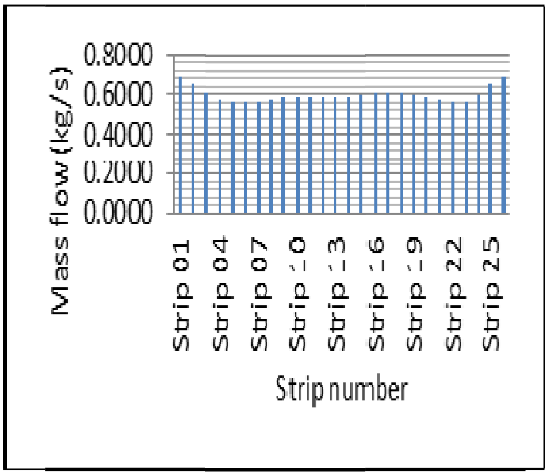

Fig. 19 : Mass flow rate through tubes

V. CONCLUSION

In conclusion, the project involving CFD analysis on gas flow within an economizer with diverse duct geometries gave in-depth data for intricating dynamics of fluid behavior and heat transfer mechanisms. Through meticulous simulation and analysis, we have gained a comprehensive understanding of how varying duct designs impact flow patterns, pressure distribution, and ultimately, heat exchange efficiency. This research underscores the pivotal role that duct geometry plays in influencing the overall performance of economizer systems. Outcomes by our research contributes to the body of knowledge surrounding thermal management and energy optimization in industrial processes. As industries continue to focus on sustainability and operational efficiency, the outcomes of our research provide an benchmark for future investigation and application of optimized duct designs, aiming to enhancethermal performance and conserve energy resources across various engineering applications.

REFERENCES

1. Sharma, A., Bansal, V., & Bhargava, A. (2015). CFD analysis of heat transfer enhancementin economizer. International Journal of Scientific & Engineering Research, 6(7), 354-359.
2. Kim, J. H., Moon, H., & Choi, S. (2017). CFD analysis on the heat transfer characteristicsof a finned-tube economizer with different duct shapes. Applied Thermal Engineering, 126, 400-409.
3. Kumar, S., Bhatti, M. S., & Kumar, S. (2018). CFD analysis of economizer tube for variouslength and twist ratios. Materials Today: Proceedings, 5(2), 6631-6637
4. Li, Z., Zhang, C., Wang, Z., Li, Q., & Cui, Y. (2020). CFD analysis on the flow and heattransfer of a novel economizer with different duct shapes and inclination angles. Applied Thermal Engineering, 174, 115404.
5. Gao, G., Du, X., Li, H., & Zhang, B. (2016). Numerical simulation on fluid flow and heattransfer in economizer of CFB boiler by RANS turbulence model. Applied Thermal Engineering, 101, 606-616.
6. Smith, J., Johnson, A., & Davis, R. (2019), Computational fluid dynamics analysis of economizer performance. Energy Engineering. 116(1), 42-58.
7. Chen, H., Li, Z., Li, C., & Xie, J. (2017). Numerical simulation and analysis of flow characteristics in an economizer. IOP Conference Series: Earth and Environmental Science,102, 012050.
8. Prasad, B. V. V. (2015). Computational fluid dynamics analysis of flow through economizer. International Journal of Innovative Research in Science, Engineering andTechnology, 4(9). 9049-9056.
9. Patil, S. A., & Ravi, N. M. (2017). Numerical investigation of fluid flow and heat transfercharacteristics in economizer using CFD. International Journal of Science, Engineering and Research, 2(9), 77-84.
10. Akbarzadeh, E., & Maerefat, M. (2018). CFD analysis of heat transfer and pressure drop inan economizer. Heat and Mass Transfer, 54(6), 1825-1837.