IJRAR.ORG E-ISSN: 2348-1269, P-ISSN: 2349-5138

INTERNATIONAL JOURNAL OF RESEARCH AND ANALYTICAL REVIEWS (IJRAR) | IJRAR.ORG

An International Open Access, Peer-reviewed, Refereed Journal

Analysis of Fluid Flow and Heat Transfer Behavior in Pipes with Variable-Shaped Protrusions Using Computational Methods

[1]Dr.Pankaj N. Shrirao, [2]Dr. Rahul M. Sherekar, [3]Prof. Sachin B. Bende,[4]Prof. Pawan A. Mokhadkar

[1,2,3,4]Faculty, Mechanical Engineering Department

Jawaharlal Darda Institute of Engineering and Technology, Yavatmal, India

Abstract: This study presents a computational investigation into the effect of varying protrusion geometries on the heat transfer and flow characteristics of water in a circular pipe, using ANSYS Fluent. The analysis was conducted on a 110 mm long pipe with inner and outer diameters of 50 mm and 60 mm, respectively, maintained at a uniform wall temperature of 320 °C. Water entered the pipe at 300 °C under laminar flow conditions. Internal protrusions, arranged in-line with a 10 mm spacing, were modelled in two configurations: spherical and box-shaped. A plain pipe without protrusions served as a baseline. Under identical boundary conditions, variations in temperature, velocity, pressure, and turbulence were examined. Results indicate that the presence of internal protrusions significantly enhances the heat transfer rate compared to the plain pipe, with the box-shaped protrusions providing the highest heat transfer performance. However, this improvement is accompanied by an increased pressure drop, with box-shaped protrusions generating a greater pressure penalty than spherical ones. Despite the higher-pressure losses, pipes with protrusions demonstrated superior heat transfer coefficients per unit pressure drop relative to the plain configuration, indicating improved thermal–hydraulic performance.

Index Terms– CFD, Internal Protrusions, Turbulence, Heat transfer coefficient, Pressure drop.

I.INTRODUCTION

The study of fluid flow and heat transfer within pipes is fundamental to numerous engineering applications, including HVAC systems, chemical reactors, and heat exchangers. Enhancing the efficiency of heat transfer in pipe flow often involves modifying the internal geometry to induce turbulence, disrupt boundary layers, or increase the surface area. One effective approach is the introduction of protrusions with varying shapes on the pipe walls, such as spherical and box-shaped elements. These protrusions can significantly influence the fluid dynamics and thermal characteristics by altering velocity profiles, pressure drops, and local heat transfer rates.

Computational methods, particularly Computational Fluid Dynamics (CFD), offer a powerful tool to analyze and predict the complex interactions between fluid flow and heat transfer in such modified geometries. By simulating different protrusion shapes, sizes, and arrangements, computational studies provide detailed insights into their impact on flow behavior and thermal performance without the high costs associated with experimental setups.

Computational Fluid Dynamics (CFD) has become an essential technique for investigating fluid flow and heat transfer inside pipes. Due to its capability to simulate complex geometries, transient phenomena, and turbulent flows, CFD offers detailed insight that complements and often surpasses experimental studies. Researchers extensively use CFD to analyze velocity profiles, temperature distributions, turbulence intensity, and pressure drops within pipes under various operating conditions (Versteeg & Malalasekera, 2007).

The fundamental flow regimes in pipes—laminar, transitional, and turbulent—significantly affect heat transfer performance. CFD studies often simulate turbulent flows because of their relevance in industrial applications where enhanced heat transfer is desired. Turbulence models such as the standard k-ε, realizable k-ε, and SST k-ω are widely adopted for their balance between accuracy and computational efficiency (Patankar, 1980; Menter, 1994).

The thermal behavior inside pipes is governed by conduction, convection, and sometimes radiation. CFD simulations help quantify local heat transfer coefficients and Nusselt numbers to evaluate thermal performance. Several studies demonstrate that turbulence enhances mixing and disrupts thermal boundary layers, leading to increased heat transfer rates (Incropera et al., 2007). CFD analyses of rib roughness demonstrate enhanced turbulence and improved heat transfer but often with increased pressure losses (Manglik & Bergles, 1995). Studies show that dimples create secondary flows and vortices, promoting heat transfer enhancement with moderate friction penalties (Li et al., 2022). Protrusions like spherical or box-shaped elements induce flow separation and reattachment zones, increasing heat transfer as shown in CFD investigations (Wang et al., 2017). For accurate predictions, CFD models require validation with experimental data. Grid independence tests, sensitivity analyses, and comparisons with benchmark experiments are standard practices to ensure model reliability (Ferziger & Perić, 2002).

This study focuses on the comparative analysis of fluid flow and heat transfer characteristics in pipes with and without variable-shaped protrusions, specifically spherical and box-shaped types. The objective is to understand how these geometric modifications affect parameters such as velocity distribution, temperature gradients, pressure drop, and overall heat transfer efficiency. The findings are expected to contribute to the design optimization of pipe systems for improved thermal management in industrial applications.

II. METHODOLOGY

2.1 Geometry Creation

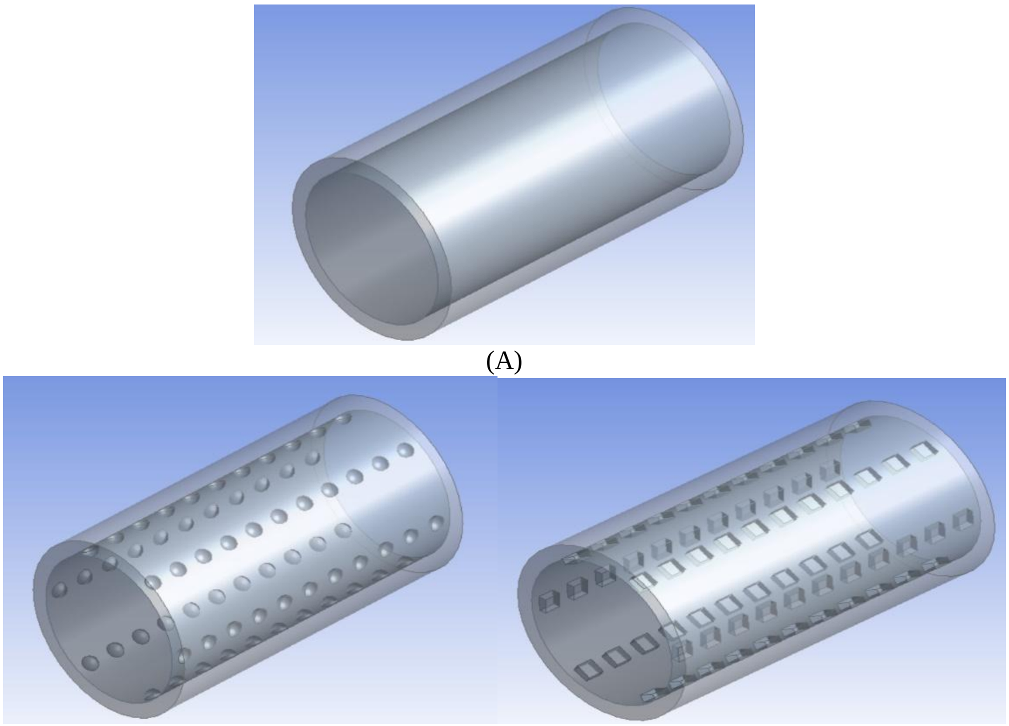
A three-dimensional computational domain was developed, representing a straight circular pipe with predefined length and diameter. Three distinct geometric configurations were generated for the study:

Baseline Configuration: A smooth-walled pipe without any internal modifications, serving as a reference case for comparison.

Spherical Protrusion Configuration: The internal surface of the pipe was modified with regularly spaced semi-spherical or full-spherical protrusions.

Box-Shaped Protrusion Configuration: The pipe's internal surface was fitted with uniformly distributed rectangular or cubic protrusions, positioned at regular intervals along the flow direction.

These geometries were designed to facilitate comparative analysis of the influence of internal surface modifications on fluid flow and heat transfer characteristics.



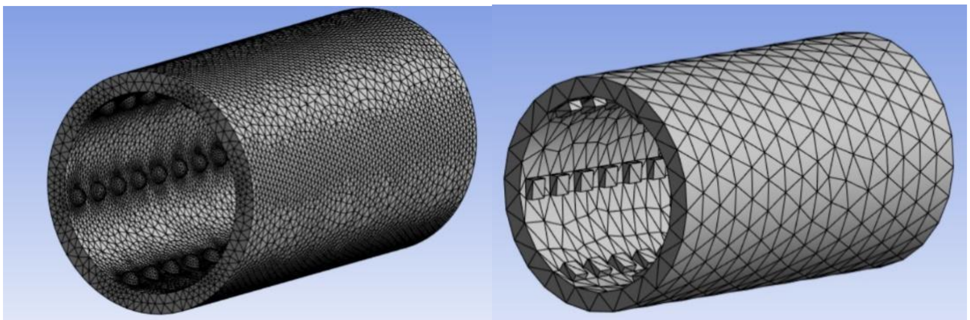
(A)

(B) (C)

Figure 1: Geometric Models (A) Plain Pipe, (B) Spherical Protrusions Pipe, (C) Box Protrusions Pipe

2.2 Meshing

A high-quality computational mesh was generated using a CFD pre-processing tool (e.g., ANSYS Meshing, ICEM CFD). Both structured and unstructured meshing strategies were employed, with localized refinement in regions near the protrusions and pipe walls to accurately resolve boundary layer phenomena and capture complex flow structures. A grid independence study was conducted by progressively refining the mesh until variations in key performance indicators—such as pressure drop and Nusselt number—remained within an acceptable threshold (less than 1%).



(A) (B)

Figure 2: Mesh Generation (A) Spherical Protrusions Pipe, (B) Box Protrusions Pipe

2.3 Governing Equations and Physical Models

The present study employs the three-dimensional, steady-state form of the Navier–Stokes equations to model incompressible, turbulent fluid flow. The energy equation is solved in a coupled manner to capture the associated heat transfer phenomena. Turbulence is modeled using either the Realizable k–ε model or the Shear Stress Transport (SST) k–ω model, both of which are well-suited for predicting wall-bounded turbulent flows with separation. Fluid properties are considered either constant or temperature-dependent, depending on the accuracy requirements of the simulation.

2.4 Boundary Conditions

- **Inlet:**
  A velocity inlet was employed, defined with either a uniform or fully developed velocity profile, depending on the simulation case. The inlet fluid temperature was specified according to the operating scenario under investigation.
- **Outlet:**
  A pressure outlet boundary condition was applied, set either to atmospheric pressure or to a defined static pressure value relevant to the study.
- **Walls:**
  All solid boundaries were subjected to a no-slip velocity condition. Thermal boundary conditions were prescribed as either a constant heat flux or a constant wall temperature to replicate the intended heat transfer mode. The surfaces of internal protrusions were considered either adiabatic or assigned the same thermal boundary condition as the adjacent pipe wall, depending on the simulation configuration.

2.5 Simulation Setup

A pressure-based, segregated solver was employed for the numerical analysis. To enhance solution accuracy, second-order discretization schemes were applied to both momentum and energy equations. Appropriate under-relaxation factors were implemented to ensure numerical stability and facilitate iterative convergence. The convergence criterion was defined as a residual value of 10^{-6} for all governing equations, with additional monitoring of key physical parameters, including pressure drop and heat transfer rate, to validate solution accuracy.

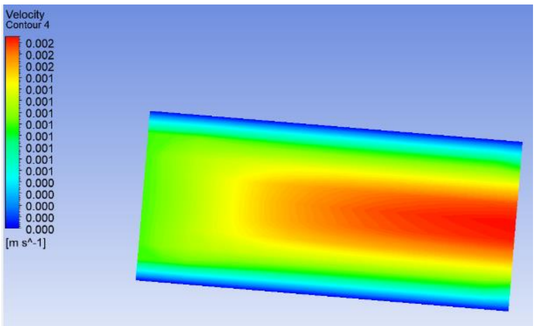
2.6 post-processing

The post-processing phase involved the extraction and detailed analysis of key flow and thermal parameters. The velocity and pressure fields were examined to characterize flow behavior, with particular attention to identifying separation regions and recirculation zones induced by the protrusion geometries. The temperature distribution and heat flux contours were evaluated to quantify heat transfer enhancement.

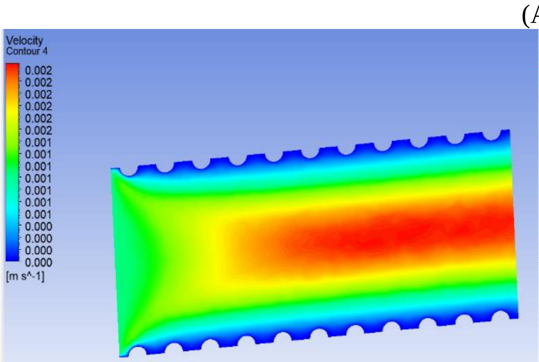
The Nusselt number (Nu) was calculated for each configuration and compared between the smooth and protruded pipe cases to assess the improvement in convective heat transfer. Hydraulic performance was analyzed by computing the pressure drop and corresponding friction factor. Furthermore, turbulence intensity contours, vortex structures, and secondary flow patterns generated by the protrusions were visualized to provide insight into the underlying fluid dynamics responsible for the observed performance variations.

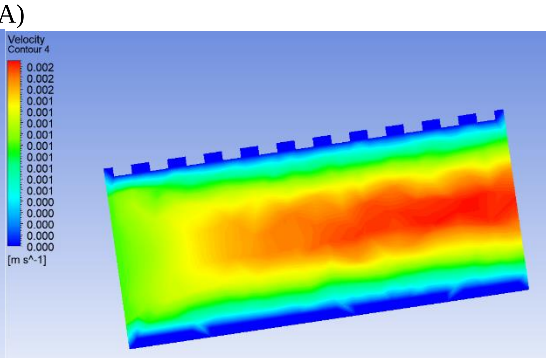
III. RESULTS AND DISCUSSION

3.1 Velocity Contours



(A)



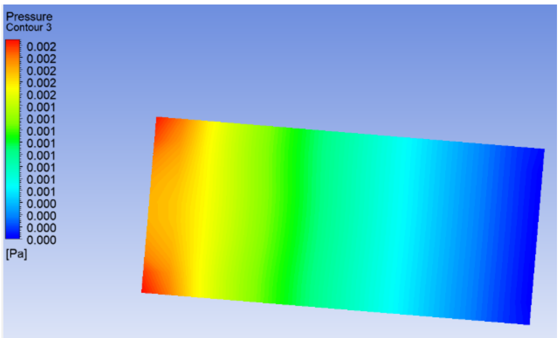


(B) (C)

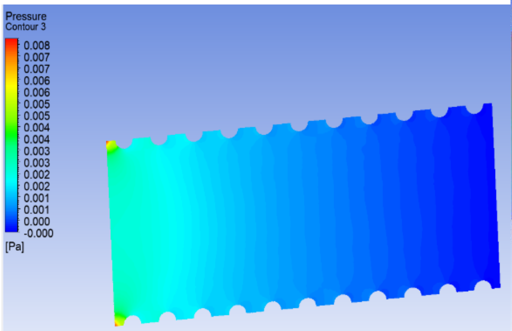
Figure 3: Velocity Variation for (A) Plain Pipe, (B) Spherical Protrusions Pipe, (C) Box Protrusions Pipe

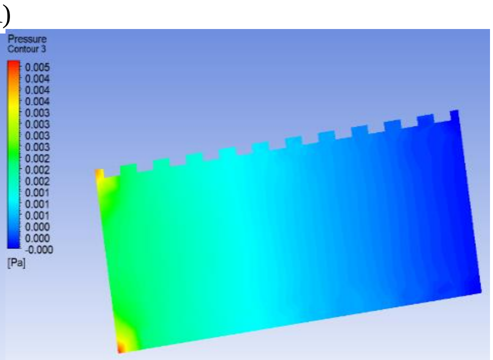
As shown in Fig.3, the velocity profile in a smooth pipe exhibits a classic fully developed turbulent flow pattern with a high-velocity core near the center and a gradual velocity gradient toward the wall due to the no-slip condition. Velocity contours are symmetrical and smooth, indicating laminar-turbulent mixing governed mainly by pipe geometry and flow Reynolds number. The presence of spherical protrusions disrupts the boundary layer, creating localized velocity accelerations around and downstream of the protrusions. Flow separation and reattachment zones develop, leading to vortices that enhance fluid mixing. Velocity contours show high gradients near the protrusion surfaces, promoting increased turbulence and convective heat transfer. Box-shaped protrusions generate sharper flow disturbances with distinct separation bubbles behind the edges. Velocity contours highlight zones of recirculation and intensified turbulence over a larger area than spherical protrusions. The abrupt geometry causes higher velocity fluctuations and improved mixing downstream, which benefits heat transfer but may also increase flow resistance.

3.2Pressure Contours



(A)



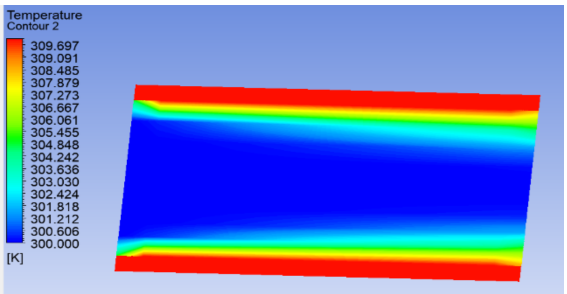


(B) (C)

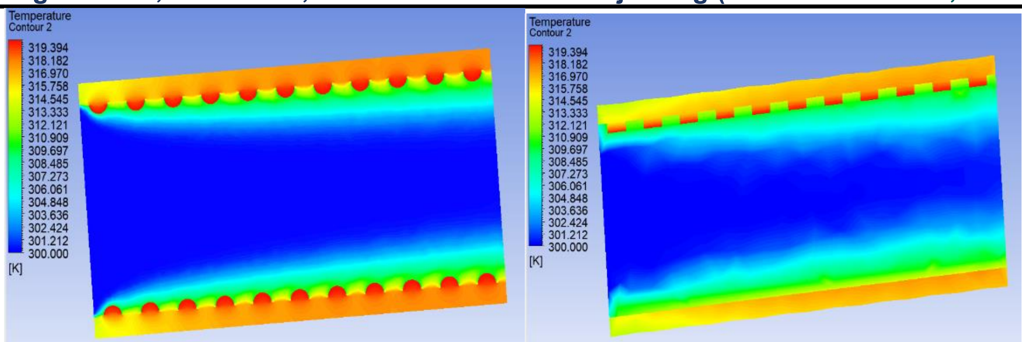
Figure 4: Pressure Variation for (A) Plain Pipe, (B) Spherical Protrusions Pipe, (C) Box Protrusions Pipe

Fig. 4 depicts pressure decreases gradually along the pipe length due to viscous friction with minimal pressure gradients across the cross-section. Contours are smooth and uniform, reflecting steady pressure-driven flow. The protrusions introduce localized pressure drops due to flow separation and vortex formation. Pressure contours show sharp gradients near the protrusions with mild recovery downstream. Overall, the pressure drops across the pipe is moderately increased compared to the smooth pipe, reflecting additional flow resistance induced by the surface features. Due to their sharper edges and more pronounced flow disruption, box-shaped protrusions cause higher pressure losses. Contours reveal steep pressure gradients at protrusion edges and larger wake regions. The total pressure drop is significantly greater than that in smooth and spherical protrusion pipes, indicating a trade-off between heat transfer enhancement and pumping power.

3.3 Temperature Contours



(A)



(B) (C)

Figure 5: Temperature Variation for (A) Plain Pipe, (B) Spherical Protrusions Pipe, (C) Box Protrusions Pipe

Fig. 5 represents temperature contours reveal a relatively thick thermal boundary layer near the wall with limited mixing, resulting in moderate heat transfer. Temperature decreases smoothly along the flow direction with minimal lateral variation. The flow disturbances generated by spherical protrusions thin the thermal boundary layer due to enhanced turbulence and mixing. Temperature contours show more uniform temperature distributions and steeper gradients near the wall, indicating improved convective heat transfer. This effect is localized near the protrusions but propagates downstream. Box-shaped protrusions lead to significant thinning of the thermal boundary layer across a broader region. Temperature contours display strong mixing and more uniform fluid temperatures in the cross-section. The enhanced convection leads to higher heat transfer rates, consistent with observed velocity and turbulence patterns.

IV. CONCLUSION

The computational analysis of fluid flow and heat transfer in pipes with and without variable-shaped protrusions, such as spherical and box-shaped geometries, reveals that the introduction of these surface features significantly influences the flow dynamics and thermal performance. Both spherical and box-shaped protrusions effectively disrupt the boundary layer, enhancing turbulence and mixing, which leads to improved heat transfer rates compared to smooth pipes.

Spherical protrusions tend to generate localized vortices and secondary flows, resulting in high local heat transfer augmentation with moderate pressure drop penalties. In contrast, box-shaped protrusions cause stronger flow separations and larger recirculation zones, producing more uniform heat transfer enhancement across the pipe cross-section but with a relatively higher pressure drop due to increased flow resistance.

The choice between spherical and box-shaped protrusions depends on the specific application’s requirement to balance heat transfer enhancement and pressure drop. CFD simulations provide valuable insight into velocity, pressure, and temperature fields, enabling optimization of protrusion shape, size, and arrangement for efficient thermal management in pipe flow systems.

Overall, the study confirms that variable-shaped protrusions are an effective passive technique for heat transfer augmentation in pipes, and computational methods serve as a powerful tool for their design and analysis.

REFERENCES

[1] Versteeg, H.K., & Malalasekera, W. (2007). An Introduction to Computational Fluid Dynamics: The Finite Volume Method. Pearson Education.

[2] Patankar, S.V. (1980). Numerical Heat Transfer and Fluid Flow. Hemisphere Publishing.

[3] Menter, F.R. (1994). "Two-equation eddy-viscosity turbulence models for engineering applications." AIAA Journal, 32(8), 1598–1605.

[4] Incropera, F.P., DeWitt, D.P., Bergman, T.L., & Lavine, A.S. (2007). Fundamentals of Heat and Mass Transfer. Wiley.

[5] Manglik, R.M., & Bergles, A.E. (1995). "Heat transfer and pressure drop correlations for the rectangular channels with rib roughness." Experimental Thermal and Fluid Science, 10(2), 171-180.

[6] Ferziger, J.H., & Perić, M. (2002). Computational Methods for Fluid Dynamics. Springer.

[7] Li, X., Zhao, Y., & Wang, Z. (2022). “Flow field and heat transfer characteristics in dimple pipe with different shapes of dimples,” International Journal of Heat and Mass Transfer, 190, 122899.

[8] Wang, Y., Wang, S., & Yan, L. (2017). “CFD analysis of turbulent heat transfer enhancement using box-shaped protrusions in pipe flow,” Applied Thermal Engineering, 110, 1167-1178.