International Journal of Trend in Scientific Research and Development (IJTSRD)
Volume 8 Issue 2, March-April 2024 Available Online: www.ijtsrd.com e-ISSN: 2456 – 6470

Numerical Investigation of Heat Transfer in Triangular Plate

Rakesh Kumar Sahu[1], Mr. Sudhir Singh Rajput[2]

[1]Student, [2]Assistant Professor,
[1,2]Raipur Institute of Technology, Raipur, Chhattisgarh, India

ABSTRACT

The primary aim of these investigations was to explore the characteristics of flow and heat transfer under varying Rayleigh numbers and aspect ratios. The results obtained through Computational Fluid Dynamics (CFD) simulations, presented graphically in the form of heat flow and stream functions, reveal that the nature of heat transfer is influenced by both the Rayleigh number and aspect ratio. Moreover, the study validates the strong dependence of the Nusselt number on the Rayleigh number, aspect ratio, and enclosure parameters. The CFD simulations effectively capture the boundary conditions of the natural convection problems under consideration.

Solutions are derived for a range of Rayleigh numbers, maintaining a constant Prandtl number (Pr = 0.70) at an aspect ratio of 1. The Pressure-Velocity Coupling Method, coupled with the Non-Iterative Time Advancement (NITA) scheme, is employed to effectively solve the problem. The distribution of temperature and stream function is analysed as a function of thermal and geometrical output parameters for the specific problem at hand. The investigation involves a two-stage approach to discern variations in results stemming from different computational approaches.

KEYWORDS: *CFD (Computational Fluid Dynamic), Ra: (Rayleigh Number), AR: (Aspect Ratio)*

How to cite this paper: Rakesh Kumar Sahu | Mr. Sudhir Singh Rajput "Numerical Investigation of Heat Transfer in Triangular Plate" Published in International Journal of Trend in Scientific Research and Development (ijtsrd), ISSN: 2456-6470, Volume-8 | Issue-2, April 2024, pp.706-708, URL: www.ijtsrd.com/papers/ijtsrd64747.pdf

IJTSRD64747

INTRODUCTION

Natural convection is a heat transfer phenomenon that occurs spontaneously due to density differences in a fluid, either liquid or gas. When a region of a fluid is heated, it becomes less dense, causing it to rise, and cooler, denser fluid descends to replace it. This movement creates a convective flow that transfers heat without the need for an external force such as a pump or fan.

Key Characteristics of Natural Convection:

- **Driven by Temperature Differences:** Natural convection is primarily driven by variations in temperature within a fluid.
- **Gravity-Dependent:** The force of gravity plays a crucial role in natural convection, causing the buoyancy effect that leads to fluid movement.
- **Occurs in Enclosed Spaces:** It is often observed in enclosed spaces, where fluid motion is constrained

The extensive study of natural convection flow and heat transfer within rectangular enclosures has been driven by its versatile applications. In vertical orientations, these enclosures serve as effective insulation for building doors and windows, the air conditioning compartments of trains, industrial furnaces, chimneys, and various heat transfer equipment. Alternatively, when positioned at an incline, these enclosures find utility in skylights, roof windows, solar collector storage, and numerous other solar applications.

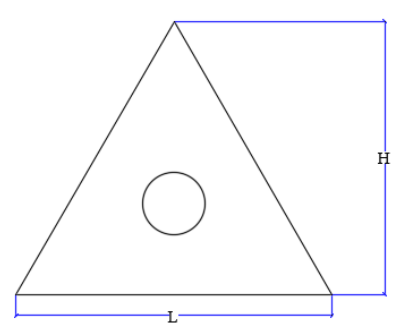

Fig. 1. Triangular enclosed vertical lamina

METHODOLOGY:

The enclosure geometry is constructed using Ansys 16 software, specifically utilizing the geometry generation workbench. Various simulations are conducted with Rayleigh numbers ranging from 10^3 to 10^6, maintaining a consistent aspect ratio of 1. In the context of this study, an aspect ratio of 1 signifies the utilization of a triangular geometry for the analysis. The heat transfer analysis is executed using Ansys Fluent.in order to validate the result of triangular enclosure and find out the nature of heat transfer phenomenon, three different geometry of cavity with different inner hole position is created and tested for analysis.

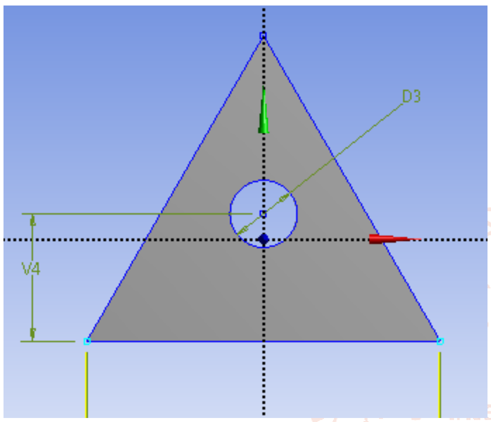

Fig. 2 Geometry of vertical enclosed space for Ra=10^3

Once the geometry is established, the next step involves dividing the control volume into a more manageable configuration of nodes and finite-sized elements, a process referred to as the finite volume method. Meshing, or the division of the control volume into smaller finite-sized volumes, is crucial for this method. In cases where the geometry is straightforward, a structured grid is favoured over an unstructured grid due to its superior performance.

In the realm of computational fluid dynamics (CFD), the ANSYS Fluent Workbench emerges as a powerful tool for unravelling the complexities of fluid flow phenomena. At the heart of this simulation prowess lies the strategic utilization of the Pressure-Velocity Coupling Method, a methodology that intricately weaves together the pressure and velocity fields within the computational domain, forming the backbone of accurate numerical solutions.

RESULT AND DISCCUSION:

To validate the code, a simulation was conducted for two-dimensional, laminar, steady natural convection of air within a vertical cavity with differentially heated side walls. The thermal conditions and dimensions of the enclosures were chosen to span a broad range of Rayleigh numbers, varying from 10^3 to 10^6.

In this simulation, the triangular edges surrounding the cavity were consistently maintained in a cold condition, while only the circular edges of the cavity were subjected to a hot condition. The properties of air were determined at the mean temperature of both the hot and cold walls.

Utilizing a commercial Computational Fluid Dynamics (CFD) software, the computed results for the average Nusselt number were obtained.

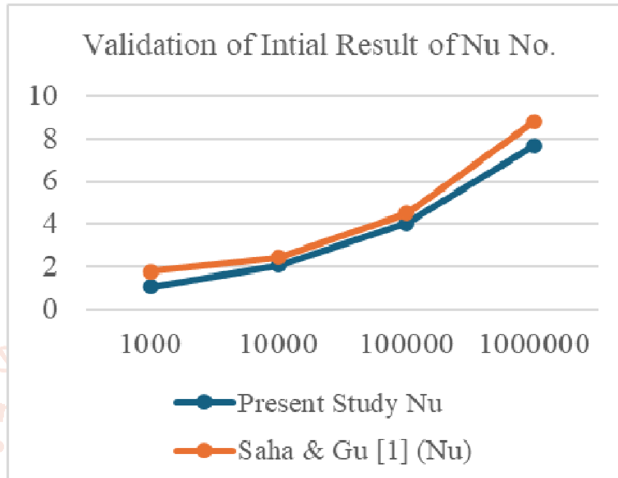

Fig.3 Validation of Initial Result of Nu No.

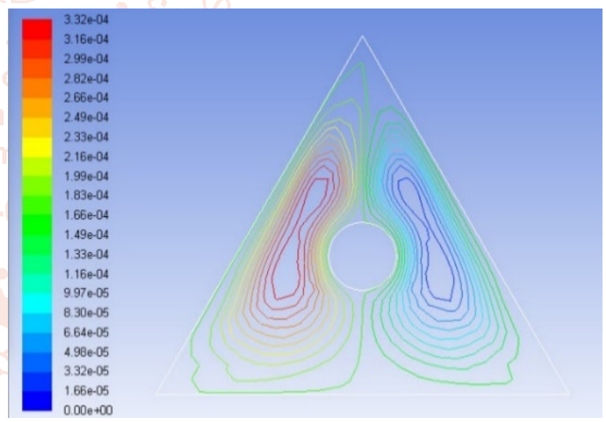

Fig.4: Stream function for lamina

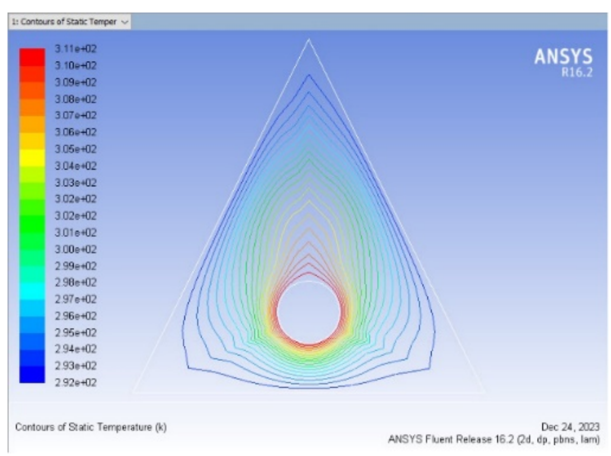

Fig. 5 Temperature plot for lamina

Conclusion

In the realm of computational fluid dynamics (CFD) analysis, the validation of numerical simulations is a critical step to ensure the accuracy and reliability of results. This study focuses on investigating natural convection heat transfer in a vertical cavity through the execution of two distinct validation methods. The comparison involves assessing Fluent simulation results against empirical findings by Saha [1].

To validate the CFD simulations, a two-fold approach was employed. Firstly, the results were compared with the empirical data provided by Saha [1], a recognized source in the field of natural convection. Secondly, a graphical representation in the form of a curve comparison, as illustrated in Figure 3, was utilized to visually assess the agreement between the Fluent simulation and the reference data.

References

[1] **Suvash C. Saha & Y.T. Gu** "Natural convection in a triangular enclosure heated from below and non-uniformly cooled from top", 2014 Elsevier Ltd.

[2] **Falah Assi Abood et al, "**Steady Laminar Natural Convection Heat Transfer inside Air-Filled Horizontal Triangular Enclosure Containing Three Cylindrical Rods", Basrah Journal for Engineering Science /2011.

[3] **Rameshwar B. Hagote, Sachin K. Dahake,** " Enhancement of Natural convection heat transfer coefficient by using V-fin array", International Journal of Engineering Research and General Science Volume 3, Issue 2, March-April, 2015 ISSN 2091-2730.

[4] **Manoj K. Triveni & Rajsekhar Panua**, "Numerical analysis of natural convection in a triangular cavity with different configurations of hot wall". International Journal Of Heat And Technologyissn: 0392-8764 Vol. 35, No. 1, March 2017, Pp. 11-18.

[5] **Qiugang Wang et al.,** "Comparison study of natural convection between rectangular and triangular enclosures" AIP Advances 11, 045303 (2021).

[6] **Raees-ul-Haq Muhammad**, "Analysis of Heat Transfer in a Triangular Enclosure Filled with a Porous Medium Saturated with Magnetized Nanofluid Charged by an Exothermic Chemical Reaction", Hindawi Mathematical Problems in Engineering, Volume 2019, Article ID 7451967.

[7] **Aaron Propst,** "CFD Analysis Methods for Systems Driven by Natural Convection" South Dakota State University Open PRAIRIE: Open Public Research Access Institutional Repository and Information Exchange.

[8] **Rahul Gundla, Dr. Jitendra A. Hole ,** A Comprehensive Study of Split-Triangular Fins under Natural Convection Field: Experimental and Numerical Investigation", International Engineering Research Journal Special Edition PGCON-MECH-2017.

[9] **B. Nageswar Bhukya et al**, " Experimental Setup for Heat Transfer Analysis on Rectangular Plate by Natural Convection", International Journal of Engineering Research and Applications www.ijera.com ISSN: 2248-9622, Vol. 11, Issue 1, (Series-I) January 2021, pp. 01-07.

[10] **Emin Fuad Kent,** "Numerical Computation Of Laminar Natural Convection In Triangular Shaped Cavities" WIT Transactions on Engineering Sciences, Vol 128, ISSN 1743-3533.

[11] **V. A. Akinsete,** "Heat Transfer By Steady Laminar Free Convection In Triangular Enclosures", Int. J. Hear Mass Transfrr. Vol. 25, No. 7. pp. 991-998.

[12] **S. H. Barhatte et al,** "Experimental And Computational Analysis And Optimization For Heat Transfer Through Fins With Different Types Of Notch", Journal of Engineering Research and Studies E-ISSN 0976-7916.

[13] **Seyed Milad Mirabedin, [13]** "CFD Modeling Of Natural Convection in Right-Angled Triangular Enclosures", International Journal Of Heat And Technology", ISSN: 0392-8764 Vol. 34, No. 3, September 2016, Pp. 503-506.

[14] **Suhad A. et al,** "Effect of orientation on the natural convection heat transfer from a heated triangular prism embedded in porous media", Elsevier Ltd. 28 May 2022.

[15] **ShantanDutta,** Numerical analysis of natural convection in isosceles triangular enclosure in presence of thermal radiation", research gate Conference Paper August 2021.