IJISEM
INTERNATIONAL JOURNAL OF INNOVATIONS IN SCIENCE ENGINEERING AND MANAGEMENT

A Review on Effect of Fluid Flow and Heat Transfer in Various Types of Cavities or Enclosed Object

OPEN ACCESS

Manuscript ID:

AG-2023-3011

Volume: 2

Issue: 3

Month: September

Year: 2023

ISSN: 2583-7117

Published: 04.09.2023

Citation:

Shivam, Lakshmi Yadav and Dr. B. Suresh. "A Review on Effect of Fluid Flow and Heat Transfer in Various Types of Cavities or Enclosed Object" International Journal of Innovations In Science Engineering And Management, vol. 2, no. 3, 2023, pp. 80–84.

This work is licensed under a Creative Commons Attribution-Share Alike 4.0 International License

Shivam Burman[1], Lakshmi Yadav[2], Dr. B. Suresh[3]

[1]*Research Scholar, Department of Mechanical Engineering, Corporate Institute Of Science & Technology, Bhopal*
[2]*Assistant Professor, Department of Mechanical Engineering, Corporate Institute Of Science &Technology, Bhopal*
[3]*HOD Department of Mechanical Engineering Corporate Institute of Science and Technology Bhopal*

Abstract

This study is a systematic review of research on heat transfer analysis in cavities, and it seeks to provide a comprehensive understanding of flow and heat transfer performance in different types of cavities with different types of heated ribs in the presence of an obstruction. The study also investigates the effects of various forces, such as drag and lift force in obstacles, as well as the thermophoresis effect on heat transfer in Fluid by heated ribs. This study also concentrates on a variety of fluids, including air, water, Newtonian and Non-Newtonian fluids, in cavities. In addition, only single-phase flow and heat transfer phenomena are covered in this review. It discusses the various validation techniques utilized in numerical studies and the various lattice types and sizes utilized by researchers. This study is a comprehensive review of a small number of research articles, the majority of which have been published since 2013, covering the most recent advancements in the field of heat transmission analysis in cavities. This study's review of the relevant literature demonstrates that cavities with impediments such as fins and rotating cylinders have a significant impact on enhancing heat transfer. In addition, it has been discovered that the use of Newtonian, Non-Newtonian, nanofluids, and hybrid nanofluids has a greater impact on enhancing heat transfer. The study concludes by proposing prospective directions for research in the field of fluid heat transfer.

Keyword: Newtonian fluid, Novel cavity, Heat transfer, heated circular obstacle, Hydrodynamics forces.

I. Introduction

The study of the mechanical properties and the forces exerted by fluids (including gases, liquids, and plasmas) is known as the fluid mechanics. It may be broken down into fluid statics, which examines fluids while they are at rest, and the fluid dynamics, which looks at how forces influence fluid motion.[1] It is a subfield of the continuum mechanics, the study of modelling matter from the macroscopic instead of microscopic perspective, without relying on the knowledge that it is composed of atoms. The study of fluids, and more specifically fluid dynamics, is a dynamic and sometimes mathematically challenging area of study. Numerical approaches, often implemented on computers, are optimal for solving many open issues. Today, this method is the focus of "computational fluid dynamics" (CFD), a relatively new field of study. Another experimental technique that makes use of fluid flow's strong visual appeal is particle image velocimetry.[2]

A. Boundary layers and separation

It bears repeating that vorticity may enter the fluid in the potential flow at the interface with the solid and at the fluid's free surface. The simple example might show how it spreads after it has entered a system. Think of a vast, still body of fluid that is put in motion by the action of a massive solid plate submerged in a fluid in its own plane. Frictional forces which prevent slide between the two transmit the motion to the fluid, along with a velocity profile. The "partial differential equation" turns out to be a good description of its time-dependent evolution.[3]

B. Drag

Any impediment in the course of a fluid stream will experience a drag force FD, and this force will remain constant even if the barrier moves while the fluid remains at rest. Designers of all types of moving vehicles, as well as designers of the cooling towers and the other buildings who want to make sure that the structures would not collapse in winds, have a vested interest in knowing how big it is and how it might have decreased.[4]

C. Lift

To perform its purpose, the wing of a moving aircraft must feel both the downward force of drag and the upward force of lift. The lift force is produced since the pressure acting on an air foil from below is greater than a pressure acting on it from above, and since the speed at which the displaced air moves over a top of an air foil is greater than a speed at which it moves over the bottom. However, it may also be considered as an unavoidable result of the limited circulation around an airfoil. As was observed in the introduction to a Magnus effect, rotating an obstruction may help to build circulation around it. However, the "air foil's forward motion" generates the circulation around it; this circulation begins as soon as the air foil travels quickly enough to escape the initial eddy.[5]

D. Laminar Flow

The flow behavior of a fluid in the laminar domain is simple and straightforward to see and analyze. Flow is fully influenced by its environment, which typically consists of just gravity and a driving pressure. It is a common misconception that only incompressible, non-viscous fluids may experience laminar flow. For incompressible flows, inviscid, the "Euler equations" might be used for prediction, whereas the more generic "Navier-Stokes equations" are utilized when turbulence is present or when the fluid is compressible.[6]

E. Turbulent Flow

The opposite end of a flow spectrum corresponds to very high flow rates and is known as "turbulent flow regime". The mathematical complexity of turbulence makes the study of flow behaviour difficult, necessitating the use of numerical methods and the specialized models. When turbulence develops, the fluid flow changes from laminar to convective (vortex-like), starting at a boundary layer and expanding in the direction of flow. Although statistical methods may be used to assess the degree of the turbulence, behavior of the turbulent flow is not always random.[7]

II. Literature Review

(Fatima et al., 2023)[8] The finite element approach is used to deal with the mathematical formulations. The research uses a range of "Reynolds and Hartmann numbers", and draws conclusions from the data using both graphical and tabular representations. Adjusting the "Reynolds and Hartmann numbers" results in pressure, velocity, and temperature charts, which depict the numerical results. We analyzed hydrodynamic (drag and lift) forces for three distinct heater setups. Line integration around the hot obstruction at various granularities is used to represent these forces as lift & drag coefficients. We found that the lift & the drag coefficient values for the rectangular heater are greater in the presence of the magnetic field than those for a triangle or the semicircular heater.

(Saha et al., 2023)[9] In order to better understand how different types of cavities perform in terms of the flow and the heat transfer, this study conducts a systematic review of previous research on cavity heat transfer analyses. The paper summarizes the state of the art in heat transfer analyses in cavities by analyzing 297 research publications, the vast majority of which have been published after the year 2000. This assessment of the relevant literature demonstrates that the presence of barriers in cavities, like fins and revolving cylinders, significantly improves heat transmission. Nanofluids & hybrid nanofluids are also proven to have a stronger impact on improving heat transfer. Finally, the study points to potential avenues of investigation for the study of heat transport in cavities. The results of this work have broad applications, from electronic cooling and energy storage to the solar thermal technologies and the nuclear reactor systems.

(Bilal et al., 2022)[10] Cold cylinder placement options for improving heat transfer characteristics in confined two-dimensional domains are investigated. Numerical finite element analysis using "COMSOL Multiphysics version 5.6" is used for the parametric investigation of the governing equations. Node velocity and temperature data may be extracted from elements with the use of quadratic interpolating functions. For the purpose of discretizing the domain into the rectangular and the triangular components, hybrid meshing is realized. A range of $0.5 \leq \leq 1.5$, $5 \leq \leq 35$ and $10 \leq \leq 10$ is explored for optimal variation in flow topologies. Graphs of stream-lines, isotherms, and regional and overall Nusselt values project the attained findings. Both the kinetic energy and the wall heat flux are tabulated for your convenience. An increase in "Rayleigh number" is assumed to result in the rise in magnitude of the kinetic

energy & convective heat transfer, whereas the opposite trend is shown in relation to power-law index.

(Hobiny & Abbas, 2022)[11] In this study, we investigate the influence of temperature fluctuations in a semiconducting media using the extended photothermoelastic theory. Under the influence of the "uniform unit step temperature" on a surface of the material, the finite element approach presents numerical solutions to the issue. Others produce numerical answers by ignoring a new parameter, and the behaviors of the physical quantities for numerical solutions are investigated to verify the correctness of the suggested technique.

(Meng et al., 2022)[12] Since flame stability is crucial to the scramjet's performance, the POD technique was used to a cavity-stabilized scramjet to study its combustion mode & instability features. Separating averaged CH* chemiluminescence pictures of 3 ethylene fuel jet equivalency ratios allowed us to identify three distinct combustion regimes. Schlieren photographs and pressure features were used to understand the formation process. Using proper "orthogonal decomposition" (POD), we were able to identify the POD modes (PDMs) present in a series of flame CH* chemiluminescence pictures. For three common stable combustion modes, PSD (power spectral density) of PDM spectra displayed strong peaks between 100 and 600 Hz. The findings reveal the pulsational features of three distinct scramjet combustion phases.

(Rehman et al., 2022)[13] Newtonian fluid in a new cavity is studied statistically to determine the thermal flow field. Triangular, rectangular, and circular heated ribs are inserted case-wise along the bottom hollow wall. These equations are solved using the finite element method. Final measurements on pressure, temperature, velocity, and hydrodynamic forces are reported using line graphs, contour plots, and tabular data. Lift and drag coefficients, in particular, are calculated by integrating a line. Nusselt's pattern of variation is the same for lower wall ribs of all shapes and sizes, including triangles, rectangles, and circles. In the case of the heated rectangular rib, values of coefficients with respect to heated obstruction are larger than in the cases of the heated triangular & circular ribs.

(Giwa et al., 2021)[14] Hydro magnetic behaviours of Nano fluids in the square enclosures were the focus of this work, which used both computational and experimental methodologies. Natural convection has been evaluated for first time in connection to other concepts such as bio-nan fluid (green Nano fluid), bio convection, ionic Nano fluid, & hybrid Nano fluid. Micro polar and hybrid Nano fluids are described as special examples of MHD natural convection in cavities. It has been shown that the presence of the magnetic fields modifies convective heat transport in the square cavities.

(Singh et al., 2021)[15] In this study, we introduce a new plus-shaped cavity in optical fiber and show that it may be used for "RI-based sensing of Escherichia coli" (E-Coli). This "plus-shaped cavity" enhances sensitivity by maximizing signal-to-analyte contact. By using both the horizontal and vertical slots in the optical fiber core construction, the suggested sensor model may be realized. The "plus-shaped cavity's sensing" capabilities have been tested in the visible spectrum, where the RI of E. coli is between 1.38 and 1.395. The "finite difference time domain" (FDTD) technique has been used for the numerical exploration of sensor structure. According to the geometric study of "plus-shaped cavity", the optimum slot width of 600 nm yields the highest output power. The suggested cavity has an autocorrelation function & sensitivity of 99.54 percent and 2125 nm/RIU, respectively, for detecting the presence of E-Coli.

(Feng et al., 2020)[16] Here, a unique idea is presented and analyzed in which a single cavity is used to accommodate two different modes. Coupled operation of the fundamental mode and a specific harmonic mode in the single cavity offers the possibility of linear bunch lengthening during acceleration, which has significant implications for enhancing beam brightness in storage rings, linacs, and the other accelerating structures. The comprehensive design analysis defines a structure that exhibits the combined fundamental and selected second harmonic modes. The bimodal cavity design is thoroughly examined for its RF performance, and it may be used for a variety of purposes to enhance either the beam's dynamic features or compactness of the targeted system.

(Geridonmez & Oztop, 2020)[17] The effects of the partial heater & the cross partial magnetic fields on flow and temperature of a nanofluid are investigated numerically in this work. "Thin plate spline radial basis functions" (Rbfs) are used to solve space derivatives of the "governing stream function-vorticity equations", while the backward Euler technique is used to solve time derivatives. Both buoyancy force & cross partial magnetic field force are factors in the "vorticity transport equation". Isotherms, Streamlines, and vorticity contours, along with the average Nusselt number, are used to illustrate the findings. Convective heat transfer improves with increasing nanoparticle concentration, partial

heater length, and Rayleigh number, but degrades with increasing Lorentz force.

III. Conclusion

Numerous authors have investigated heat transfer analysis inside of cavities. Various geometries have been applied to the confined cavity in order to optimize heat transfer and flow behavior within the cavity. The majority of published articles dealt with rectangular and square cavities, while the remaining geometries included triangles, trapezoids, semicircles, etc. The impact of different cavity configurations has been the subject of extensive research. Future research will also investigate the addition of fins in heated ribs or objects and obstacle within a cavity, as well as the use of various rib configurations, in order to enhance heat transfer in fluids.

References

[1] A. Malleswaran and S. Sivasankaran, "A Numerical simulation on MHD mixed convection in a lid-driven cavity with corner heaters," J. Appl. Fluid Mech., vol. 9, no. 1, pp. 311–319, 2016, doi: 10.18869/acadpub.jafm.68.224.22903.

[2] I. Zahan, R. Nasrin, and M. A. Alim, "Mixed convective hybrid nanofluid flow in lid-driven undulated cavity: Effect of MHD and joule heating," J. Nav. Archit. Mar. Eng., vol. 16, no. 2, pp. 109–126, 2019, doi: 10.3329/jname.v16i2.40585.

[3] A. Gaetano, S. A. Zavattoni, M. C. Barbato, P. Good, G. Ambrosetti, and A. Pedretti, "Design Optimization of a Novel Receiving Cavity for Concentrated Solar Power Applications by Means of 3D CFD Simulations," Energy Procedia, vol. 69, pp. 379–387, 2015, doi: 10.1016/j.egypro.2015.03.044.

[4] Y. Tang, L. Luo, P. Zhang, and M. Ma, "Effect of rear wall inclination on cavity acoustic characteristics at high Mach numbers," J. Phys. Conf. Ser., vol. 2252, no. 1, 2022, doi: 10.1088/1742-6596/2252/1/012007.

[5] M. K. Raihan et al., "Flow of non-newtonian fluids in a single-cavity microchannel," Micromachines, vol. 12, no. 7, 2021, doi: 10.3390/mi12070836.

[6] Y. Khan et al., "Numerical Computations of Non-Newtonian Fluid Flow in Hexagonal Cavity with a Square Obstacle: A Hybrid Mesh–Based Study," Front. Phys., vol. 10, no. May, pp. 1–14, 2022, doi: 10.3389/fphy.2022.891163.

[7] P. Sikdar and S. M. Dash, "A numerical study on the lid-driven cavity with power-law fluids at different moving lengths of the top lid," CFD Lett., vol. 12, no. 6, pp. 107–117, 2020, doi: 10.37934/cfdl.12.6.107117.

[8] N. Fatima, N. Kousar, K. U. Rehman, and W. Shatanawi, "Magneto-thermal convection in partially heated novel cavity with multiple heaters at bottom wall: A numerical solution," Case Stud. Therm. Eng., vol. 43, no. December 2022, p. 102781, 2023, doi: 10.1016/j.csite.2023.102781.

[9] G. Saha, A. A. Y. Al-Waaly, M. C. Paul, and S. C. Saha, "Heat Transfer in Cavities: Configurative Systematic Review," Energies, vol. 16, no. 5, 2023, doi: 10.3390/en16052338.

[10] S. Bilal, N. Z. Khan, I. A. Shah, J. Awrejcewicz, A. Akgül, and M. B. Riaz, "Heat and flow control in cavity with cold circular cylinder placed in non-newtonian fluid by performing finite element simulations," Coatings, vol. 12, no. 1, pp. 1–19, 2022, doi: 10.3390/coatings12010016.

[11] A. D. Hobiny and I. Abbas, "The impacts of variable thermal conductivity in a semiconducting medium using finite element method," Case Stud. Therm. Eng., vol. 31, no. January, p. 101773, 2022, doi: 10.1016/j.csite.2022.101773.

[12] Y. Meng, W. Sun, H. Gu, F. Chen, and R. Zhou, "Supersonic Combustion Mode Analysis of a Cavity Based Scramjet," Aerospace, vol. 9, no. 12, 2022, doi: 10.3390/aerospace9120826.

[13] K. U. Rehman, N. Fatima, N. Kousar, and W. Shatanawi, "Modelling of thermal energy individualities in novel enclosure with uniformly heated circular obstacle and multi-shaped heated ribs," Case Stud. Therm. Eng., vol. 34, no. April, p. 102014, 2022, doi: 10.1016/j.csite.2022.102014.

[14] S. O. Giwa, M. Sharifpur, M. H. Ahmadi, and J. P. Meyer, A review of magnetic field influence on natural convection heat transfer performance of nanofluids in square cavities, vol. 145, no. 5. Springer International Publishing, 2021. doi: 10.1007/s10973-020-09832-3.

[15] L. Singh, G. Kumar, S. Jain, and B. Kumar Kaushik, "A novel plus shaped cavity based optical fiber sensor

for the detection of Escherichia-Coli," Results Opt., vol. 5, no. September, p. 100156, 2021, doi: 10.1016/j.rio.2021.100156.

[16] H. Feng, W. Huang, C. Tang, and D. Li, "Design of a novel bimodal cavity," Rev. Sci. Instrum., vol. 91, no. 6, 2020, doi: 10.1063/5.0008027.

[17] B. P. Geridonmez and H. F. Oztop, "MHD natural convection in a cavity in the presence of cross partial magnetic fields and Al2O3-water nanofluid," Comput. Math. with Appl., vol. 80, no. 12, pp. 2796–2810, 2020, doi: 10.1016/j.camwa.2020.10.003.