RESEARCH ARTICLE

RSP Science Hub

International Research Journal on Advanced Science Hub
2582-4376
Vol. 03, Issue 06
www.rspsciencehub.com

Check for updates

http://dx.doi.org/10.47392/irjash.2021.151

Solving Natural Convection by Navier-Stokes Equation in a Closed Cavity using MATLAB

Ajay Trehan[1], *Shivani Srivastava*[2]

[1] *Associate Professor, Department of Mechanical Engineering, Dr B. R. Ambedkar National Institute of Technology, Jalandhar, 144011, Punjab, India*

[2] *PG Scholar, Department of Mechanical Engineering, Dr. B. R. Ambedkar National Institute of Technology, Jalandhar, 144011, Punjab, India*

Email: shivanishanu11@gmail.com

Article History

Received: 16 May 2021
Accepted: 23 June 2021

Keywords:

CFD;
Nanoparticles;
Rayleigh number;
Nusselt number;
Simulation

Abstract

The objective of this study was to analyze the heat transfer and fluid flow within a closed cavity, which usually enhance by application of Nanofluids. The movement of the Nanofluids within the enclosure were governed by Buoyancy force and Gravitational force. Different kinds of Nanofluids were employed for purpose of examining the motion of heat flow and the stream line formation, within the close cavity. The closed cavity considered for this study comprised of a flush mounted heater on the left wall, the right wall was maintained at a constant temperature throughout the investigation and the top & bottom wall were kept insulated. In this study, MATLAB was used for performing the simulation and solving the fundamentals of Computational Fluid Dynamics (CFD), namely the Energy equation and the Momentum equation. The analysis to estimate the heat transfer and fluid flow in a closed cavity by the use of different Nanofluids have been presented in terms of several parameters such as Rayleigh number, Nusselt number, Aspect ratio and Volume fraction respectively.

1. Introduction

Heat transfer through a natural convection mode in an enclosure had been in research from the past decades and the reason behind where the natural convection is the frequent process in nature itself, it has many industrial applications too (Azad, Groulx, and Donaldson Moria Kumar, Chanakya, and Bartwal). During this time period, nanotechnology gaining the interest of the researcher more. Study of fluid flow and heat transfer by use of nanofluid helped in enhancing the result and value of work was comparatively more efficient, presently.

In this paper, a finite element simulation has been performed to inspect natural convection heat transfer in a moderately heated rectangular cavity filled with nanoscale particles and base fluid as water with uniform thermal boundary condition (Taghilou and Khavasi). The enclosure was completely insulated at top, bottom and left wall, and the left wall was climb up by a flush mounted heater at the finite extent (Ho, Jang, and Lai). Heat transfer and fluid flow inside the enclosure was due to buoyancy force. Different volume fraction, Rayleigh number, mean Nusselt number and local Nusselt number was calculated for different nanofluids Cu, Ag, Pure water, TiO_2, $Al2O_3$. The increase in average Nusselt number value for the change of nanofluid indicated the increase in heat transfer (Okabe et al. Susantez,

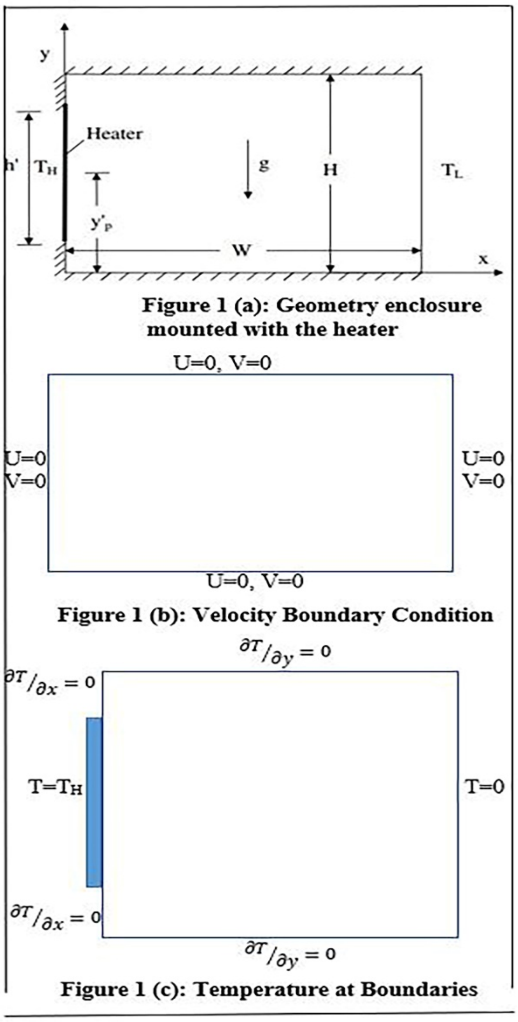

FIGURE 1. **Representing theGeometry and Boundary Conditions (a) Geometry enclosure mouthed with the heater (b) Velocity Boundary Condition (c) Temperature at Boundaries.**

Caldeira, and Loiola Mahdi et al.).

Study was done by solving the non-dimensional Navier stokes equation with the assumption of Boussinesq model and finite volume method was used (Venkateshwar et al.). The energy equation was in expression of temperature, and pressure as the source term which was solved with different approach of CFD to analysis the flow phenomena streamlines and temperature profile by simulation on software, MATLAB (Ahn et al. D. Yang et al. Loenko, Shenoy, and Sheremet). This study talks about changed in heat transfer rate with respect to different nanofluid and with different volume fraction of nanofluid (Ma et al.). This simulation work with the help of some particular solving approach of CFD figure out the temperature change of nanofluid near the heater and as the particle near to heater get heated up early its molecular momentum increases and the particles start moving due to buoyancy force, the upward thrust and further new particles was forced to come near the left wall mounted by heater and heated again by same phenomena and so on (P. Yang et al. Khan et al. Pranowo and Wijayanta) .This way the particles able to move forward toward the top wall and under the influence of gravitational force the particles move downward after reaching the end of top wall, this clockwise circular streamline path was seem in the particular enclosure (Moria).

The primary aim of this research study was to solve the non-dimensional governing equation (Boussineq model) of CFD for natural convection heat transfer in a partially heated rectangular enclosure filled with nanofluids (Barman and Rao). The main literature review discussed further specified that there was no study on natural convection of a rectangular enclosure, for solving non dimensional governing equation of CFD by simulation method (by software work as a mechanical engineer) (Akinshilo Kumar, Chanakya, and Bartwal Alizadeh et al.).

2. Methodology

The Figure-1(a). shows the Geometry enclosure mounted with the heater. In this particular enclosure all the pressure, temperature, velocities etc was solved. Buoyancy force was responsible for fluid flow as the initial movement of particle was generated because of the upward thrust applied to the heated particle near to the heater and secondly, the motion of fluid in the cavity is because of increase of temperature as temperature increases the momentum increases and fluid particles start moving and here the change of density of fluid was seen with the increase of temperature which was assumed as constant because of the approach of Boussineq model. We were using dimensionless study to simulate the work. Maximum temperature can be seen near to the heater so maximum velocity of fluid particle was near to the heater.

The gap in research study that mainly identified was:

- The numerical and experimental analysis were done, but the simulation work was not performed by any researcher.

• The study was not solved with the help of Computational Fluid Dynamics (CFD) approach.

• The study was always solved under the assumption that pressure change was constant, but use of CFD approach to solve the pressure changes was never employed.

2.1. Boundary Conditions:

On left wall (heater): u=0, v=0, T=T_HOn bottom wall and top wall: u=0,v=0,y=0

2.2. Governing Equation

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{\partial P}{\partial x}+\frac{\mu_{nf}}{\rho_{nf}}\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right]$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{\partial P}{\partial y}+\frac{\mu_{nf}}{\rho_{nf}}\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right]+\frac{\left(\varphi\rho_s\beta_s+(1-\varphi)\rho_f\beta_f\right)g}{\rho_{nf}}T$$

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0$$

$$u\frac{\partial T}{\partial x}+v\frac{\partial T}{\partial y}=\alpha_{nf}\left[\frac{\partial^2 T}{\partial x^2}+\frac{\partial^2 T}{\partial y^2}\right]$$

2.3. Numerical Method

2.3.1. Finite Volume Method

The solving approach was done by using finite volume method of CFD to discretise the equations. We have three regions here:

1- above the heater, 2- domain of heater, 3- below the heater

We using Ly1, ny1 for region 1. Ny1 represent all mesh in y direction for region 1, and respectively.

$dx = \frac{Lx}{nx-1}$ $dy3 = \frac{Ly3}{ny3-1}$ $dy1 = \frac{Ly1}{ny1-1}$ $dy2 = \frac{Ly2}{ny2-1}$

2.3.2. Formation for P, U, V, T

Figure-2(b). illustrates General Mesh, if three regions are combined then new mesh is created and we reach this mesh. This mesh is very less and only for better understanding. We divided y direction to ny= 5 and x direction to nx=5. The nx and ny is too important because they help us for numbering of u velocity element and v, P, T. In the next slide we show all element for u, v, P, T.

In Figure-2(c). Mesh for U Velocity is shown and black vectors are boundary element and they are known. But blue vectors have been calculated. U matrix is (ny+1, nx) matrix (blue arrow position).

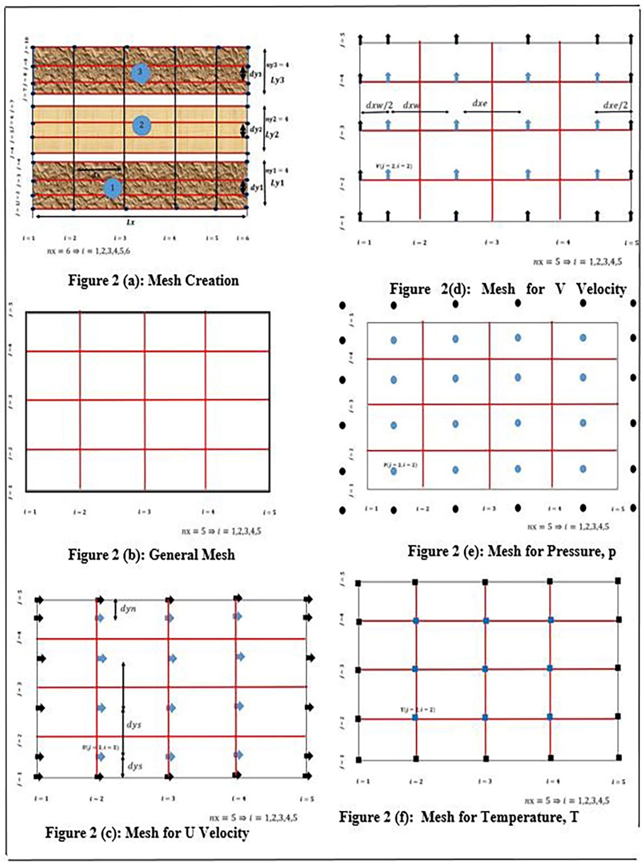

FIGURE 2. Mesh for Temperature (T), Velocity (V), Pressure (P) (a) Mesh Creation (b) General Mesh (c) Mesh for U Velocity (d) Mesh for Velocity (e) Mesh for Pressure, p and (f) Mesh for Temperature, T.

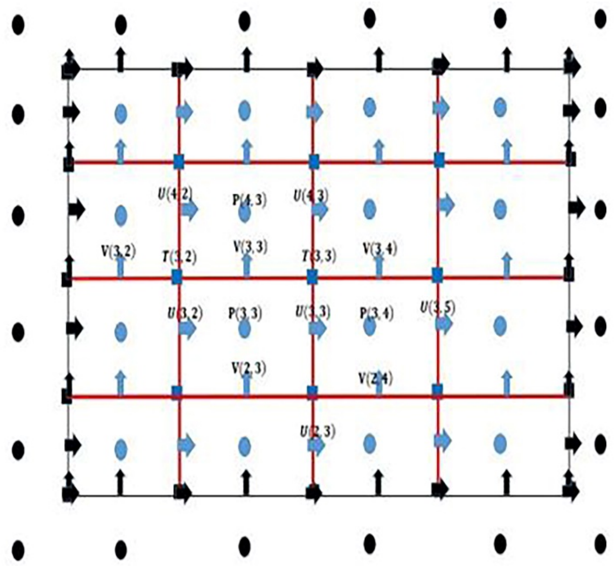

FIGURE 3. General Mesh for U, V, T, P

Black vectors are boundary element and they are known. But blue vectors must be calculated. V matrix is (ny, nx+1) matrix.

Black vectors are boundary element and they are known. But blue vectors must be calculated. P matrix is (ny+1, nx+1) matrix. We add more node for pressure for making essay programming. All black node pressure is zero. Black vectors are boundary element and they are known. But blue vectors must be calculated. T matrix is (ny,nx) matrix.

2.4. Boussinesq Model:

To solve the natural-convection flows, Boussinesq model was used in this problem as according to this model the density was the function of temperature and the source term used is:

$$(\rho - \rho_0)g \approx -\rho_0\beta\,(T - T0)g$$

2.5. Momentum Equation:

$$u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial P}{\partial y} + \frac{\mu_{nf}}{\rho_{nf}}\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right] + \frac{(\varphi\rho_s\beta_s + (1-\varphi)\rho_f\beta_f)g}{\rho_{nf}}T$$

2.6. TDMA Method:

The Technique for rapidly solving tri-diagonal systems that is now called the Thomas Algorithm or the tri-diagonal matrix algorithm (TDMA). TDMA algorithm was a two-step method that have features both of direct and iterative methods. iterative methods such as Gauss-Sidle and Jacobi methods and direct methods such as Gaussian elimination and LU methods (Chen et al.). In this method, first, all variables assumed to be unknown in one direction and known in the other. The second stage was the opposite of the previous stage. In this study, first the variables in the one direction were assumed to be unknown, so for the first step we have:

$$step1: \; -awU_W^{k+\frac{1}{2}} + apU_P^{k+\frac{1}{2}} - aeU_E^{k+\frac{1}{2}} = asU_S^{k} + anU_N^{k} + su$$

$$step2: \; -asU_S^{k+1} + apU_P^{k+1} - anU_N^{k+1} = awU_W^{k+\frac{1}{2}} + aeU_E^{k+\frac{1}{2}} + su$$

2.7. Simple Algorithm

Semi-Implicit Method for Pressure Linked Equations is an essentially a guess-and-correct procedure for the calculation of pressure on the staggered grid arrangement.

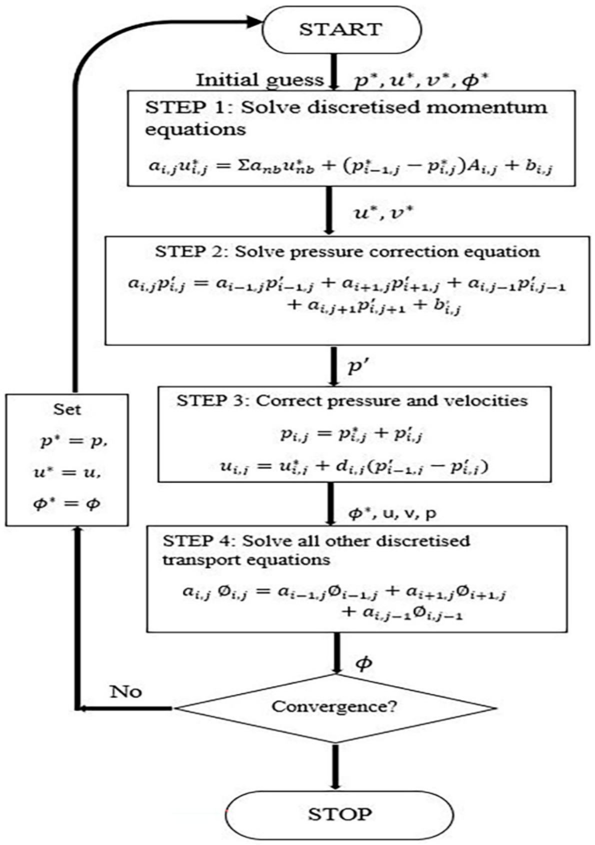

Chart 1: Simple Flow Chart

TABLE 1. Thermophysical Properties of Fluid and Nanoparticles

Properties	Ag	Cu	Al2O$_3$	TiO$_2$	Water
Cp	235	383	765	686.2	4179
Ro	10500	8954	3600	4250	997.1
K	429	400	46	8.954	0.6
Beta	5.4	16	0.63	2.4	21

3. Results and Discussion

In the partially heated heater analysis of buoyancy flow induced natural convection by simulation work using fundamental Navier Stock equation and techniques used was of computational fluid dynamic to discretised the velocity, temperature, pressure equation. The base case was taken with A=1, Ra=10^5, ø=0.05.

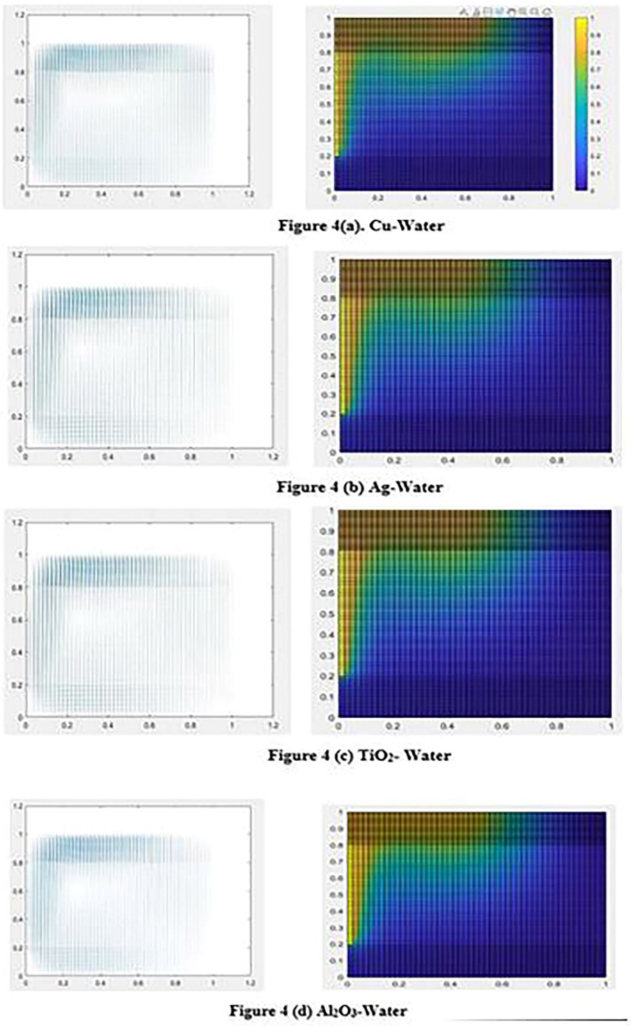

FIGURE 4. **Streamline (Left) and Temperature Variation (Right) (a) Cu- Water (b) Ag- Water (c) TiO_2 Water (d) AlO_2- Water**

TABLE 2. **Thermophysical Properties of Fluid and Nanoparticles at Ra =10^5.**

C.o. f	Ag	Cu	$Al2O_3$	TiO_2
0.05	6.4532	6.4802	6.2841	6.1758
0.1	7.4856	7.5296	7.1629	6.9169
0.15	8.5856	8.6338	8.1403	7.7219
0.2	9.7729	9.8157	9.2358	8.6035

TABLE 3. **Thermophysical Properties of Fluid and Nanoparticles at Ra=10^3.**

C. o. f	Ag	Cu	$Al2O_3$	TiO_2
0.05	6.1713	6.1711	6.1411	6.0192
0.1	7.1024	7.1019	7.0354	6.7971
0.15	8.1425	8.1415	8.0302	7.5851
0.2	9.3118	9.3103	9.1434	8.4837

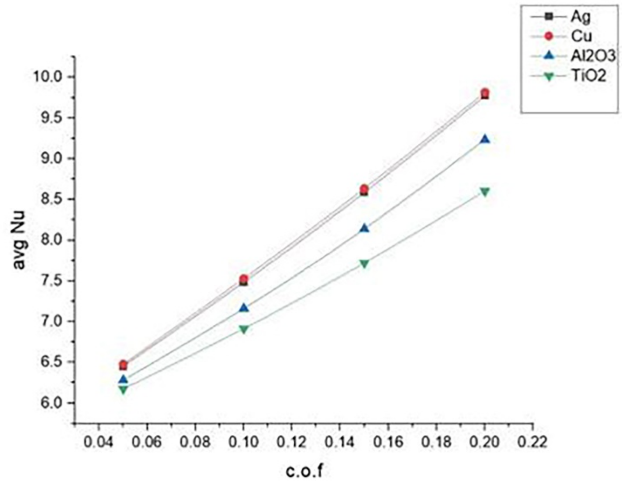

FIGURE 5. **Variation of average Nusselt number corresponding to different volume fraction (cof) using different nanoparticles, at defined Rayleigh Number Ra =10^5.**

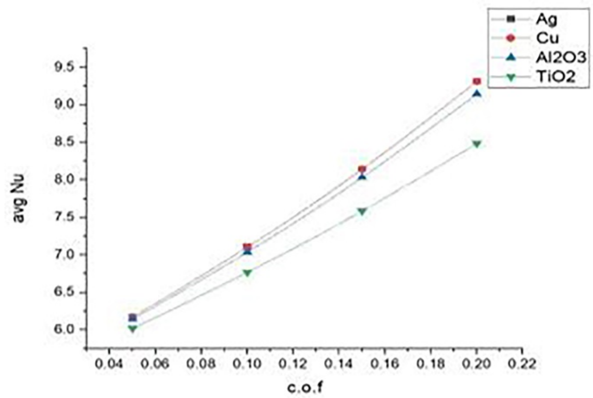

FIGURE 6. **Variation of average Nusselt number corresponding to different volume fraction (cof) using different nanoparticles, at defined Rayleigh Number Ra =10^3.**

TABLE 4. **Thermophysical Properties of Fluid and Nanoparticles at A= 0.5, 1 & 2**

C. o. f	A=0.5	A=1	A=2
0.05	5.9847	4.8092	3.0462
0.1	6.9795	5.5897	4.1686
0.15	7.9878	6.4045	4.7487
0.2	9.0391	7.2730	5.1589

Figure 4. [a-d] shows Streamline (Left) and Temperature Variation (Right) where a comparison between Cu-water, Ag-water, TiO_2-water and Al_2O_3-water using base conditions is explained. A=1, Ra=10^5, ø=0.05, iteration =100, convergence

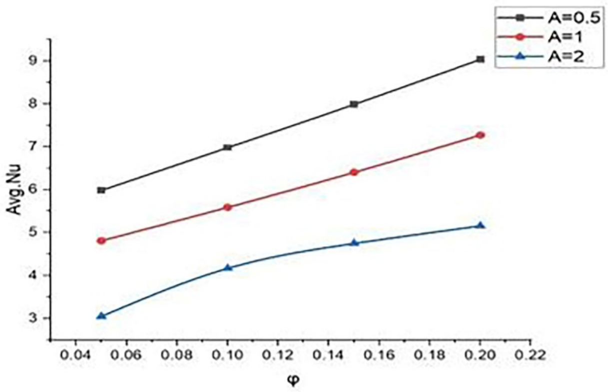

FIGURE 7. Variation in value of average Nusselt number corresponding to different values of ø at defined A=0.5, 1 & 2.

of result =0 and mesh define for the rectangular cavity as nx=50; ny1=30; ny2=60; ny3=40; ny=ny1+ny2+ny3-2;

Figure-5. Variation of average Nusselt number corresponding to different volume fraction (cof) using different nanoparticles, at defined Rayleigh Number Ra =10ˆ5. The figure here shows the monotonically increase in heat transfer corresponding to increasing volume fraction for all the mentioned nanofluids at three values of Rayleigh number. For Ra=10ˆ5 there was negligible changes found for Ag and Cu and for further decreases value of Ra the Cu and Ag shows almost the same behaviour of increasing heat transfer and gave same mean Nusselt number. The lowest value was seen for TiO_2 because of the dominating of conduction mode of heat transfer. Therefore, at high Rayleigh number the convection mode of heat transfer is more and as we increased the volume fraction of nanofluids, the larger difference for mean Nusselt number was gained.

4. Conclusion

This study was based completely on simulation work by using the software MATLAB to investigate the effect of any nanofluid with water as base fluid inside the rectangular shaped cavity. The important points drawn from this study was Increasing the Rayleigh no. and heater scale enhance the heat transfer and fluid flow strength without affecting other discussed parameters and Different formation of heat and fluid flow was attend at different volume fraction of different nanofluids. Heat transfer values, Nusselt no. values all depend on different nanofluids. In future study, for the rectangular profile and with any nanofluids this study can easily.

Data Availability Statement

All data that support the findings of this study are included within the article (and any supplementary files)

ORCID iDs

Ajay Trehan https://orcid.org/0000-0002-7097-1094

References

Ahn, Chan-Sol, et al. “Experimental, analytical, and computational study of natural convection in asymmetrically-heated vertical shafts”. *International Journal of Thermal Sciences* 170 (2021): 107131–107131. 10.1016/j.ijthermalsci.2021.107131.

Akinshilo, Akinbowale T. “Thermal Performance Evaluation of MHD Nanofluid Transport Through a Rotating System Undergoing Uniform Injection/Suction with Heat Generation”. *BioNanoScience* 9.3 (2019): 740–748. 10.1007/s12668-019-00653-9.

Alizadeh, Rasool, et al. “Artificial intelligence prediction of natural convection of heat in an oscillating cavity filled by CuO nanofluid”. *Journal of the Taiwan Institute of Chemical Engineers* 124 (2021): 75–90. 10.1016/j.jtice.2021.04.067.

Azad, Mohammad, Dominic Groulx, and Adam Donaldson. “Natural convection onset during melting of phase change materials: Part I – Effects of the geometry, Stefan number, and degree of subcooling”. *International Journal of Thermal Sciences* 170 (2021): 107180–107180. 10.1016/j.ijthermalsci.2021.107180.

Chen, Yanjun, et al. “Numerical simulation and analysis of natural convective flow and heat transfer of nanofluid under electric field”. *International Communications in Heat and Mass Transfer* 120 (2021): 105053–105053. 10.1016/j.icheatmasstransfer.2020.105053.

Ho, C. J., C. Jang, and Chi-Ming Lai. “Natural convection heat transfer in PCM suspensions in a square enclosure with bottom heating and top cooling". *Thermal Science and Engineering Progress* 25 (2021): 101037–101037. 10.1016/j.tsep.2021.101037.

Kumar, Pradeep, G. Chanakya, and Naman Bartwal. "Investigations of non-gray/gray radiative heat transfer effect on natural convection in tall cavities at low operating temperature". *International Communications in Heat and Mass Transfer* 125 (2021): 105288–105288. 10.1016/j.icheatmasstransfer.2021.105288.

Loenko, Darya S., Aroon Shenoy, and Mikhail A. Sheremet. "Effect of time-dependent wall temperature on natural convection of a non-Newtonian fluid in an enclosure". *International Journal of Thermal Sciences* 166 (2021): 106973–106973. 10.1016/j.ijthermalsci.2021.106973.

Ma, Yuan, et al. "MHD forced convection of MWCNT–Fe3O4/water hybrid nanofluid in a partially heated τ-shaped channel using LBM". *Journal of Thermal Analysis and Calorimetry* 136.4 (2019): 1723–1735. 10.1007/s10973-018-7788-4.

Mahdi, Mustafa S., et al. "Natural convection improvement of PCM melting in partition latent heat energy storage: Numerical study with experimental validation". *International Communications in Heat and Mass Transfer* 126 (2021): 105463–105463. 10.1016/j.icheatmasstransfer.2021.105463.

Okabe, Takahiro, et al. "Spatio-temporal measurement of natural convective heat transfer on melting process using infrared thermography". *International Journal of Heat and Mass Transfer* 181 (2021): 121882–121882. 10.1016/j.ijheatmasstransfer.2021.121882.

Pranowo and Agung Tri Wijayanta. "Numerical solution strategy for natural convection problems in a triangular cavity using a direct meshless local Petrov-Galerkin method combined with an implicit artificial-compressibility model". *Engineering Analysis with Boundary Elements* 126 (2021): 13–29. 10.1016/j.enganabound.2021.02.006.

Susantez, Çiğdem, Aldélio B. Caldeira, and Bruna R. Loiola. "Natural convection effects on TNT solidification inside a shaped charge mold". *Defence Technology* (2021). 10.1016/j.dt.2021.07.010.

Taghilou, Mohammad and Ehsan Khavasi. "Thermal behavior of a PCM filled heat sink: The contrast between ambient heat convection and heat thermal storage". *Applied Thermal Engineering* 174 (2020): 115273–115273. 10.1016/j.applthermaleng.2020.115273.

Venkateshwar, K., et al. "Influence of Natural Convection on Design of Thermal Diodes". *International Communications in Heat and Mass Transfer* 127 (2021): 105574–105574. 10.1016/j.icheatmasstransfer.2021.105574.

Yang, Dezhi, et al. "Thermal transport of natural convection flow of second grade bio-nanofluid in a vertical channel". *Case Studies in Thermal Engineering* 28 (2021): 101377–101377. 10.1016/j.csite.2021.101377.

Yang, Pengying, et al. "Simulating natural convection with high Rayleigh numbers using the Smoothed Particle Hydrodynamics method". *International Journal of Heat and Mass Transfer* 166 (2021): 120758–120758. 10.1016/j.ijheatmasstransfer.2020.120758.

Embargo period: The article has no embargo period.

To cite this Article: Trehan, Ajay, and Shivani Srivastava. "Solving **Natural Convection by Navier-Stokes Equation in a Closed Cavity using MATLAB**." International Research Journal on Advanced Science Hub 03.06 (2021): 124–130. http://dx.doi.org/10.47392/irjash.2021.151