International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 10 Issue: 11 | Nov 2023 www.irjet.net p-ISSN: 2395-0072

FORCED CONVECTIVE HEAT TRANSFER IN A LID-DRIVEN CAVITY

Anjan Bhunia[1], Souvik Maity[1], Anupam Santra[1], Sribas Samanta[1]

[1]*Dept. of Mechanical Engineering, College of Engineering & Management, Kolaghat, India*

***

Abstract - *This work deals with the processes of forced convective heat transfer within a lid-driven cavity, using numerical simulations to assess the thermal behavior under varied flow conditions. The lid-driven cavity is a traditional structure with several technical and industrial uses. The goal is to comprehend the complex interplay between fluid flow patterns and heat transmission within the hollow. The numerical simulations are carried out using a proven computational fluid dynamics (CFD) model, with the effects of Reynolds number and aspect ratio taken into account. Basically, this work consists Reynolds number (Re), Nusselt number (Nu) and Prandtl number (Pr). We are taking a rectangular channel whose top wall is moving and the working fluid of Prandtl number (Pr) equal to 0.71 and 5.83 is utilized for this study.*

***Key Words*: Forced convection; Rectangular channel; Heat generation; Lid-driven cavity; Reynolds number; Prandtl number**

NOMENCLATURE

Nu	Nusselt number
Pr	Prandtl Number
Re	Reynolds number

1. INTRODUCTION

Forced convection within rectangular channel holds a great relevance in engineering applications such as heat exchangers, cooling systems, and other thermal management equipment. The practical ramifications of this research include the design and optimization of rectangular channel heat exchangers, cooling systems, and other thermal management equipment. By utilizing forced convection inside rectangular geometries, the findings of this study help to the creation of efficient and sustainable engineering solutions. Prior studies have looked into heat transfer enhancement in an inclined square cavity filled with nanofluid, heat transfer in a two-sided Lid-driven cavity filled with volumetrically heat-generating porous media, etc. Basically, this work deals with Nusselt number (Nu) and Prandtl number (Pr). Nusselt number (Nu) is the ratio of heat flow rate by convection and heat flow rate by conduction.

$$Nu = \frac{convective\ heat\ transfer}{conductive\ heat\ transfer} = \frac{hl}{k}$$

For forced convection, Nu is independent of Grashof number.

$$Nu = f(Re, Pr)$$

And the Prandtl number (Pr) is the ratio of kinematic and dynamic viscosity.

$$Pr = \frac{Kinematic\ viscosity}{Dynamic\ viscosity} = \frac{v}{\alpha} = \frac{C_p\mu}{k}$$

For air, Pr =0.71

For water, Pr =5.83

2. PROBLEM FORMULATION

This study deals into captivating domain of computational fluid dynamics (CFD) within a rectangular channel. A rectangular channel with the dimension of height (H) 1m and length (L) of 1m. The top wall of the channel is maintaining the no slip condition while traveling continuously in the positive x-direction. The other walls are stationary. The upper and lower wall of the rectangular channel is insulated and other two side wall is one heating wall and another is cooling wall. The working fluid of Prandtl number (Pr) equals to 0.71 and 5.83 are utilized for this investigation.

The walls are filled with water and air, and the walls are at a specified reference temperature. Except for the mass density, which varies according to the Boussinesq approximation, the thermo-physical parameters of the working fluid are considered to remain constant. The fluid is taken to be Newtonian and incompressible.

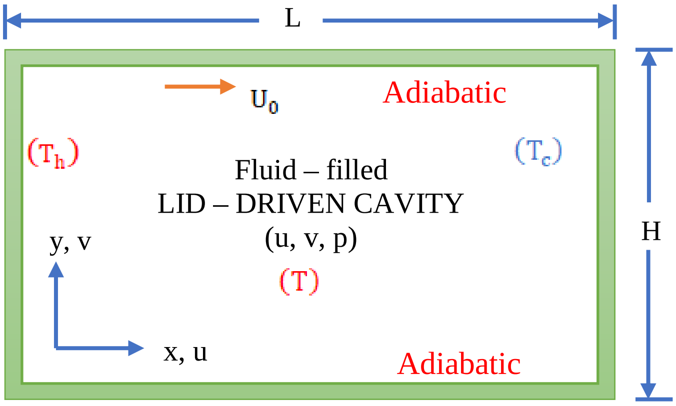


Fig. 1: Problem Formulation

3. MATHEMATICAL FORMULATION

Assumptions are two – dimensional steady incompressible Newtonian laminar flow. The gravity force is neglected. Applying the conservations principles of mass, momentum and energy, the continuity, momentum and energy equations are written as,

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{1}$$

$$\rho\left(u.\frac{\partial u}{\partial x} + v.\frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2}\right) \tag{2}$$

$$\rho\left(u.\frac{\partial v}{\partial x} + v.\frac{\partial v}{\partial y}\right) = -\frac{\partial p}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2}\right) \tag{3}$$

$$\rho C_p\left(u.\frac{\partial T}{\partial x} + v.\frac{\partial T}{\partial y}\right) = k\left(\frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2}\right) \tag{4}$$

The above – mentioned dimensional governing equations are transformed into the non – dimensional equations as expressed below,

$$\frac{\partial U}{\partial X} + \frac{\partial V}{\partial Y} = 0 \tag{5}$$

$$\left(U.\frac{\partial U}{\partial X} + V.\frac{\partial U}{\partial Y}\right) = -\frac{\partial P}{\partial X} + \frac{1}{Re}\left(\frac{\partial^2 U}{\partial X^2} + \frac{\partial^2 U}{\partial Y^2}\right) \tag{6}$$

$$\left(U.\frac{\partial V}{\partial X} + V.\frac{\partial V}{\partial Y}\right) = -\frac{\partial P}{\partial Y} + \frac{1}{Re}\left(\frac{\partial^2 V}{\partial X^2} + \frac{\partial^2 V}{\partial Y^2}\right) \tag{7}$$

$$\left(U.\frac{\partial \theta}{\partial X} + V.\frac{\partial \theta}{\partial Y}\right) = \frac{1}{Re\ Pr}\left(\frac{\partial^2 \theta}{\partial X^2} + \frac{\partial^2 \theta}{\partial Y^2}\right) \tag{8}$$

Here, the transformation of primitive variables (*u, v, p, T)* is made on the basis of certain scales as given by,

$$(X,Y) = \frac{(x,y)}{H}, (U,V) = \frac{(u,v)}{U_0}$$

$$P = \frac{p}{\rho {U_0}^2}, \theta = \frac{T - T_c}{T_h - T_c} \tag{9}$$

$$Pr = \frac{v}{\alpha}, Re = \frac{U_0\ H}{v}$$

4. NUMERICAL ASPECTS

The acronym for computational fluid dynamics is CFD. This area of fluid mechanics uses numerical techniques and algorithms to investigate and solve problems involving fluid flows. CFD has evolved into a strong tool in engineering and scientific research for modeling fluid (liquids and gases) behavior in a variety of applications. Here we are using the CFD domain for fluid flow and heat transfer enhancement in a rectangular channel.

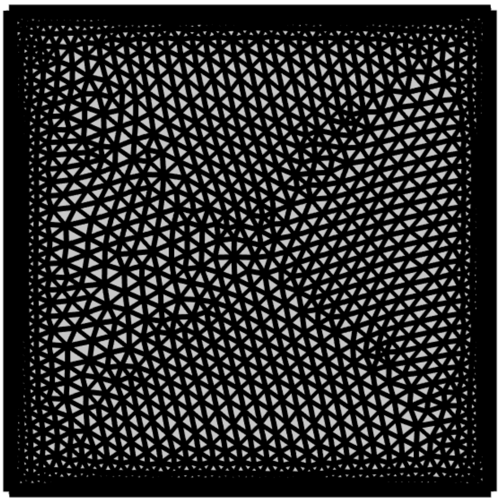


Fig. 2: Meshing Model

For analysis purpose we are using the COMSOL Multiphysics 5.6 version and choosing the domain of computational fluid dynamics. The model we are using has a physics-controlled mesh sequence with fine element sizes.

5. RESULTS AND DISCUSSION

Under the result section the color filled magnitude plot, absolute velocity & streamlines contour plot are given.

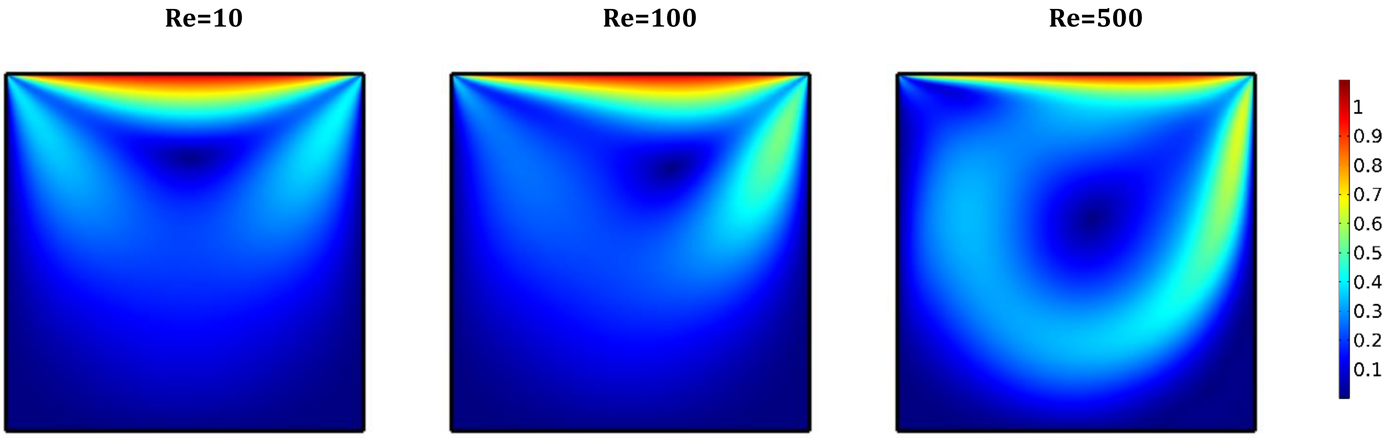


Fig.3: Color filled magnitude plot: Absolute velocity (**Pr=0.71, 5.83**)

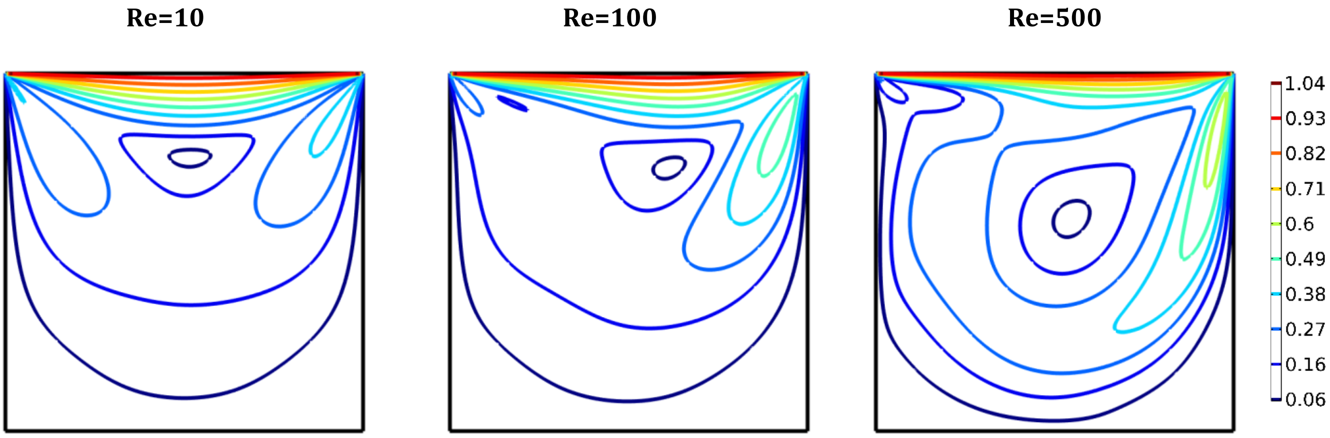


Fig.4: Absolute velocity contour plot

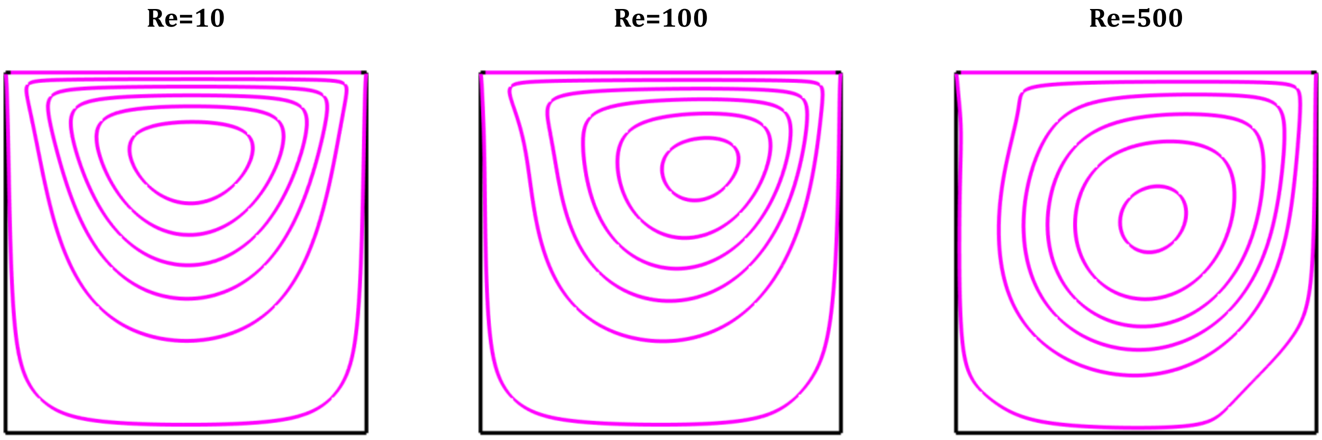


Fig.5: Streamlines contour plot

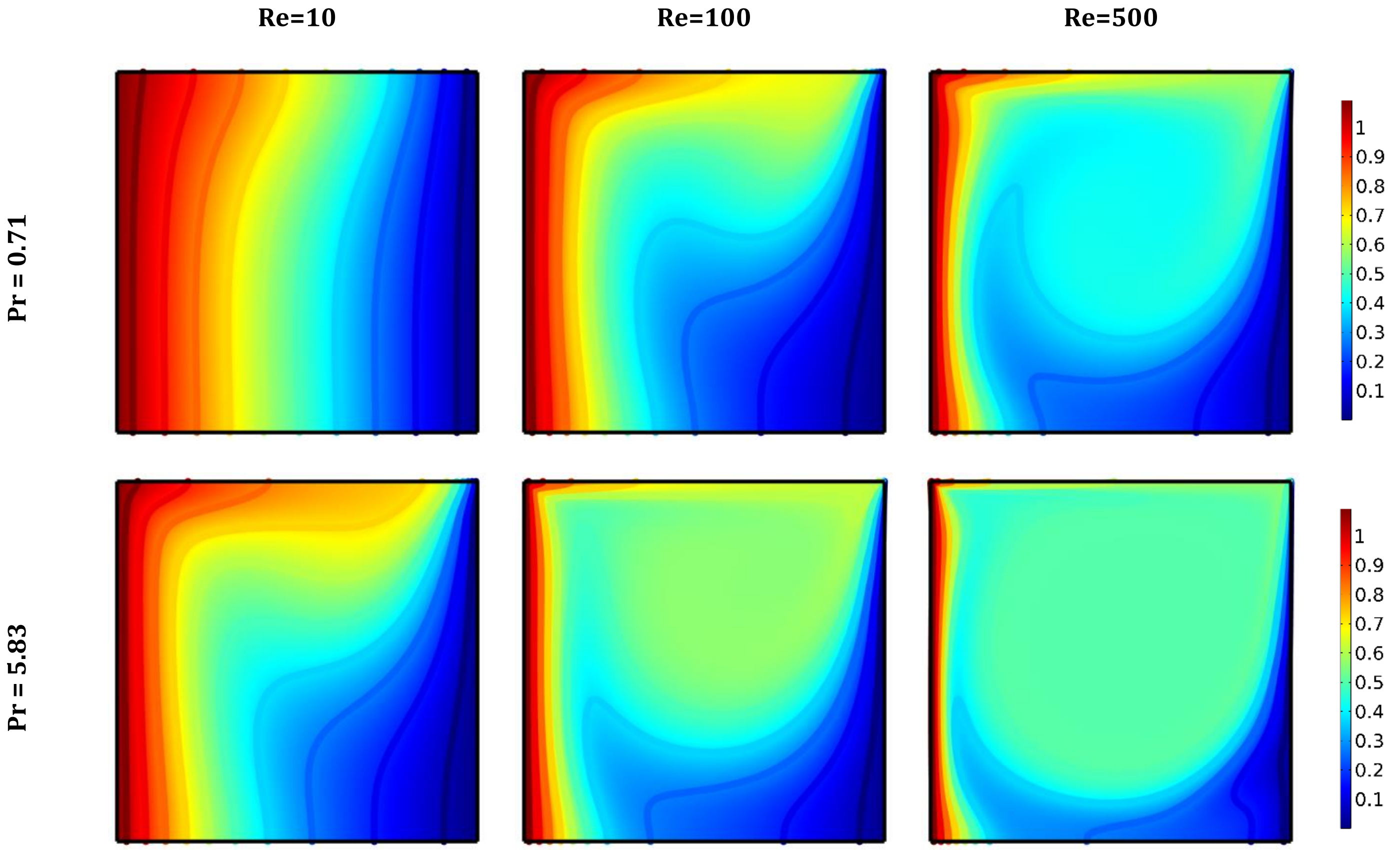
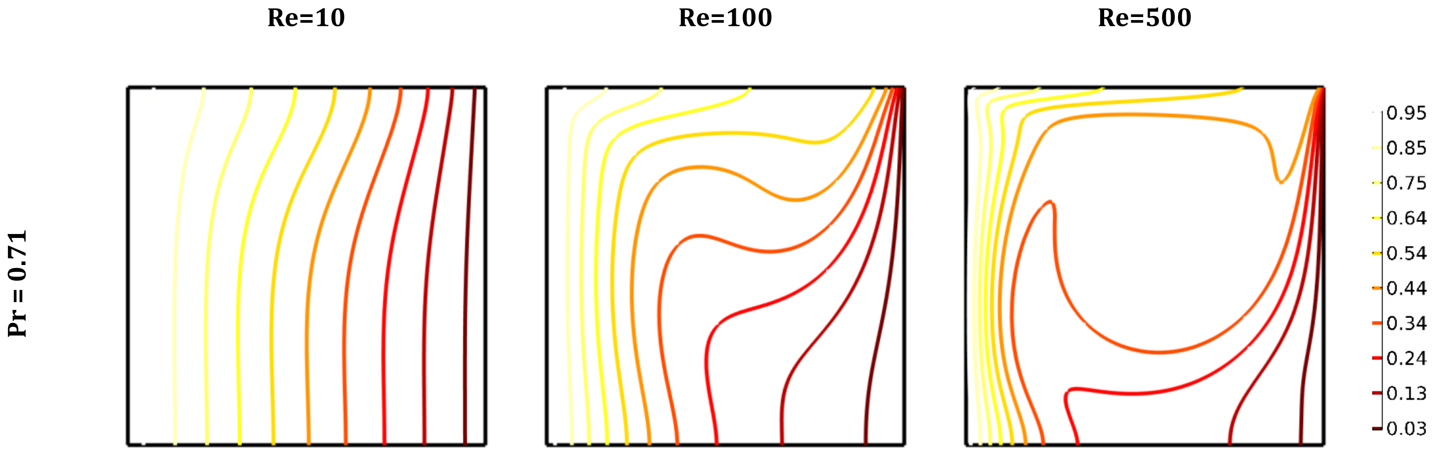


Fig.6: Temperature surface plot



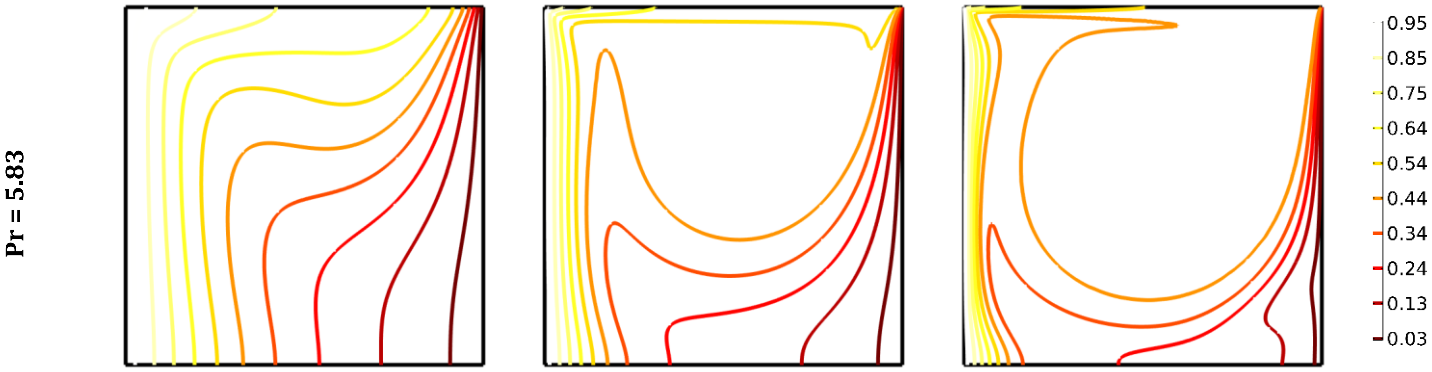


Fig.7: Isotherm plot (**Pr =0.71, 5.83**)

Figure 3 shows the color filled magnitude plot of absolute velocity where the plot of absolute velocity is same for both air (Pr= 0.71) and water (Pr= 5.83) Prandtl number. For the whole study we are taking basically three types of Reynolds number (Re= 10, 100 & 500). In the figure 4, it shows the absolute velocity contour plot and figure 5 shows the Streamlines contour plot. In figure 6 and 7 it shows the temperature surface plot and isotherm plot. In figure 6, we fixed the Reynolds number but for Prandtl number variation it shows the different contour.

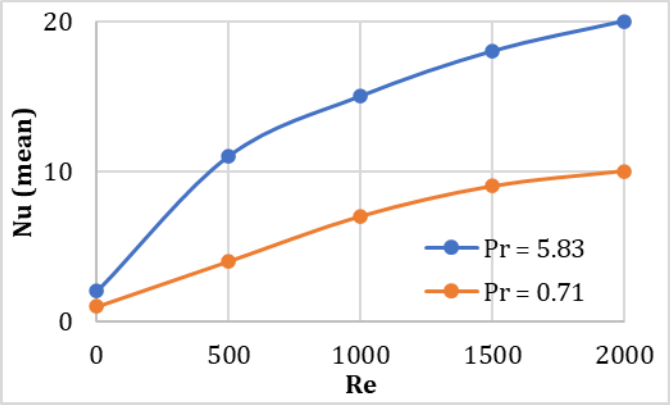


Fig.8: Trend of average Nusselt number (Nu) at the hot wall

6. CONCLUSIONS

We investigate the forced convection and heat transfer in a rectangular channel using computational fluid dynamics. Analyzing laminar fluid flow and transient results provides valuable insights into CFD phenomena. The results shows that fluid flow patterns are independent of Prandtl number (as Pr is absence in the momentum equations). And the thermal aspects significantly affected by Re as well as Pr. Enhance convective flow has profound impact on the enhancement of heat transfer and nonlinear static temperature distribution.

7. REFERENCES

[1] J. V. Indukuri and R. Maniyeri, "Numerical study of forced convection heat transfer in an oscillating lid driven cavity with heated top wall," *Int. J. Heat Technol.*, vol. 36, no. 4, pp. 1378–1387, 2018, doi: 10.18280/ijht.360429.

[2] A. Boutra, K. Ragui, N. Labsi, and Y. K. Benkahla, "Lid-driven and inclined square cavity filled with a nanofluid: Optimum heat transfer," *Open Eng.*, vol. 5, no. 1, pp. 248–255, 2015, doi: 10.1515/eng-2015-0028.

[3] M. Manchanda and K. M. Gangawane, "Mixed convection in a two-sided lid-driven cavity containing heated triangular block for non-Newtonian power-law fluids," *Int. J. Mech. Sci.*, vol. 144, no. June, pp. 235–248, 2018, doi: 10.1016/j.ijmecsci.2018.06.005.

[4] Heat and Mass Transfer, R.K. Rajput, year 2008

[5] Heat and Mass Transfer, P.K. Nag, year 2018

BIOGRAPHIES

Anjan Bhunia
B. Tech (Pursuing)
Dept. of Mechanical Engineering
College of Engineering & Management, Kolaghat

Souvik Maity
B. Tech (Pursuing)
Dept. of Mechanical Engineering
College of Engineering & Management, Kolaghat

Anupam Santra
B. Tech (Pursuing)
Dept. of Mechanical Engineering
College of Engineering & Management, Kolaghat

Sribas Samanta
B. Tech (Pursuing)
Dept. of Mechanical Engineering
College of Engineering & Management, Kolaghat