International Journal of Advancements in Mechanical and Aeronautical Engineering – IJAMAE
Volume 1 : Issue 1
Publication Date : 09 January 2014

Magnetohydrodynamic flow and heat transfer around a circular cylinder in an unconfined medium

Satish Kumar Gupta, Dipankar Chatterjee, Bittagopal Mondal

***Abstract*—A two dimensional numerical simulation is carried out for analyzing the fluid flow and forced convection heat transfer using a finite volume approach for the hydromagnetic flow around a circular cylinder at low Reynolds numbers. The cylinder is placed in an unconfined medium and acted upon by the magnetohydrodynamic (MHD) flow of a viscous incompressible and electrically conductive fluid. The magnetic field is applied either along the streamwise or transverse directions. Fictitious confining boundaries are considered on the lateral sides of the simulation domain to make the problem computationally feasible. The simulation is carried out for the range of Reynolds number $10 \leq Re \leq 50$ with Hartmann number $0 \leq Ha \leq 10$ and a blockage parameter, $\beta = d/H = 5\%$. In the present study, the results are presented graphically for the Prandtl number, Pr = 0.71 (air).**

Keywords— MHD, Forced Convection, Circular Cylinder, Hartmann Number

I. Introduction

The thermohydrodynamics of obstructed magnetohydrodynamic (MHD) flow is an important subject of research because of its numerous engineering applications in several industrial processes. One of the niche areas is the handling of liquid metal in metallurgical processes, stirring, pumping, casting, etc. that involves the interaction of a conducting fluid with externally applied magnetic field. the objective of the present investigation is to numerically simulate the flow and forced convection heat transfer from a heated cylinder placed in an unconfined medium and subjected to externally applied magnetic fields. Additionally, the suppression of flow separation by imposed magnetic field is also given due consideration and the critical magnetic field strength is computed for which the flow becomes completely suppressed.

Apart from the tremendous engineering importance, the MHD flow and heat transfer problems are fundamentally very interesting as well as challenging, since the addition of Maxwell's laws to ordinary hydrodynamics and the associated action of the Lorentz forces and Joule heating give rise to intriguing physical phenomena. In the context of using heat transfer promoters like cylindrical obstacles to induce vortices and enhance heat transfer rates, experimental studies are performed by several researchers [1-2]. Some numerical studies [3-5] are also available with a major focus on the two and three-dimensional wake dynamics subjected to externally applied magnetic fields and hydrodynamic instabilities for MHD flows past internal obstacles. Recently, Chatterjee and Chatterjee [6] reported numerical studies for the wall bounded MHD flow and heat transfer around a circular obstacle for low Reynolds and Hartmann numbers. They demonstrated that the hydrodynamic instability was suppressed more for transverse magnetic field in comparison to the streamwise magnetic field.

From the above survey, the suppression of flow separation by imposed magnetic field is also given due consideration and the critical magnetic field strength is computed for which the flow becomes completely suppressed. In the present study, the magnetic fields are applied either along the streamwise or transverse directions and the simulations are performed for the Reynolds and Hartmann number ranges of $10 \leq Re \leq 50$ and $0 \leq Ha \leq 10$, respectively for a fixed Prandtl number, Pr = 0.71.

II. Physical problem, governing equations, boundary conditions

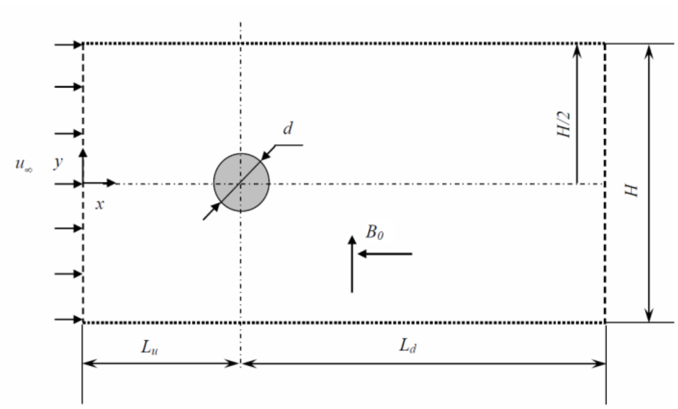


Figure 1. Schematic diagram of the computational domain. The external magnetic field (B) is applied either along streamwise (*x*) or transverse (*y*) directions

Figure 1 shows the schematic diagram of the physical problem considered in the present study. A fixed circular cylinder of diameter d heated to a constant temperature T_w is exposed to a uniform free stream of air with velocity u_∞ and

Satish Kumar Gupta
National Institute of Technology, Durgapur
India
Email: gsatish60@gmail.com

Dipankar Chatterjee (**Corresponding Author**)
CSIR-Central Mechanical Engineering Research Institute, Durgapur
India
Email: d_chatterjee@cmeri.res.in

Bittagopal Mondal
CSIR-Central Mechanical Engineering Research Institute, Durgapur
India
Email: b_mondal@cmeri.res.in

temperature T_∞ ($<T_w$). It should be mentioned that although air is not an electrically conducting fluid, its use is permissible for simulation purpose. Additionally many works are available using air as the working medium which facilitates numerical validation.The flow is considered in an unbounded medium. However, fictitious confining boundaries are assumed on the lateral sides that makes the blockage as $\beta=d/H=5\%$. (where H is the width of the computational domain). The upstream and downstream lengths of the computational domain are fixed as $L_u=12d$ and $L_d=42d$ respectively. One important assumption that goes with the present formulation is that the magnetic Reynolds number is very small ($Re_m=u_\infty d/\eta<<1$, with η being the magnetic diffusivity) such that any induced magnetic field can safely be neglected. Furthermore, the solid cylinders walls are assumed to be electrically insulated.

Assuming a two-dimensional, laminar, incompressible flow of a Newtonian and electrically conductive fluid with constant thermophysical properties, the governing inductionless differential equations in the dimensionless form consisting of the conservation of mass, momentum, energy and charge can be expressed as:

$$\nabla \cdot \mathbf{u} = 0 \quad (1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u}.\nabla)\mathbf{u} = -\nabla p + \frac{1}{Re}\nabla^2 \mathbf{u} + \mathbf{F}_\mathrm{L} \quad (2)$$

$$\frac{\partial \Theta}{\partial t} + (\mathbf{u}.\nabla)\Theta = \frac{1}{Re\,Pr}\nabla^2 \Theta \quad (3)$$

$$\nabla \cdot \mathbf{j} = 0 \quad (4)$$

Here $\mathbf{u}(u_x,u_y) = \bar{\mathbf{u}}/u_\infty$, $t = u_\infty \bar{t}/d$, $p = \bar{p}/\rho u_\infty^2$ and $\Theta = (\bar{T}-T_\infty)/(T_w - T_\infty)$ denote the dimensionless velocity, time, pressure and temperature, $\mathbf{F}_\mathrm{L} = N(\mathbf{j}\times\mathbf{B_0})$ is the Lorentz force with $N = Ha^2/Re$ being the interaction parameter (or Stuart number) and $\mathbf{B_0}$ is the unit vector of the applied magnetic field, $\mathbf{j} = \bar{\mathbf{j}}/\sigma u_\infty B_0$ is the dimensionless induced electric current density, which is given by the Ohm's law as:

$$\mathbf{j} = \mathbf{E} + (\mathbf{u}\times\mathbf{B_0}) \quad (5)$$

The dimensionless electric field $\mathbf{E}$ in Eq. (5) is irrotational and can be expressed in terms of the electrostatic potential as $\mathbf{E} = -\nabla\Phi$. Applying the solenoidal condition (Eq. 4) on Eq. (5), a Poisson equation can be derived for the dimensionless electric potential $\Phi = \bar{\Phi}/u_\infty B_0 d$ as,

$$\nabla^2\Phi = \nabla.(\mathbf{u}\times\mathbf{B_0}) = \mathbf{B_0}\cdot\mathbf{\Omega} \quad (6)$$

where $\mathbf{\Omega}$ is the fluid vorticity.

Three major dimensionless parameters that appear in the governing equations are hydrodynamic Reynolds number, Re ($=u_\infty d/\nu$), Hartmann number, $Ha = B_0 d\sqrt{\sigma/\rho\nu}$ and Prandtl number, $Pr = \nu/\alpha$. The material properties are represented by density ρ, kinematic viscosity ν, specific heat c_p, thermal conductivity κ and electrical conductivity σ. The overbar quantities such as $\bar{\mathbf{u}}$, $\bar{t}$, $\bar{p}$, $\bar{T}$, $\bar{\mathbf{j}}$ and $\bar{\Phi}$ represent the corresponding dimensional values.

The boundary conditions are as follows: at the inlet a uniform flow is prescribed ($u_x = 1, u_y = 0, \Theta = 0$). The exit boundary is located sufficiently far downstream from the region of interest, hence an outflow boundary condition (anticipating a fully developed flow situation) is proposed at the outlet. A symmetry boundary condition considering a frictionless wall ($\partial u_x/\partial y = u_y = 0$) and zero heat flux ($\partial\Theta/\partial y = 0$) are imposed on the artificial confining boundaries and a no-slip boundary condition with a uniform prescribed temperature ($\Theta = 1$) is imposed on the cylinder surfaces .

III. Method of solution and numerical verification.

The numerical simulation is performed by using the commercial CFD package FLUENT [7]. FLUENT uses a control volume based technique to solve the governing system of partial differential equations in a collocated grid system by constructing a set of discrete algebraic equations with conservative properties. The pressure based numerical scheme, which solves the discretized governing equations sequentially, is selected. The two-dimensional double precision laminar viscous model is chosen to account for the low Reynolds number flow consideration. An implicit scheme is applied to obtain the discretized system of equations. The sequence updates the velocity field through the solution of the momentum equations using known values for pressure and velocity. Then, it solves a 'Poisson-type' pressure correction equation obtained by combining the continuity and momentum equations. The QUICK scheme is used for spatial discretization of the convective terms and a central difference scheme is used for the diffusive terms of the momentum and energy equations. SIMPLEC algorithm is selected as the pressure-velocity coupling scheme. Finally, the algebraic equations are solved by using the Gauss-Siedel point-by-point iterative method in conjunction with the Algebraic Multigrid (AMG) method solver. The convergence criteria for the inner (time step) iterations are set as 10^{-8} for the discretized continuity and momentum equations and 10^{-12} for the discretized energy equation.

A non-uniform grid distribution having a close clustering of grid points in the vicinity of the solid cylinder wall is used in the present computation. Grids are generated using the grid generation package GAMBIT. A grid independence study has also been carried out with three different grid sizes of 80000, 93000 and 121000 quadrilateral cells. Finally, a mesh size of 93000 quadrilateral cells is chosen based on the comparison of drag coefficients and surface Nusselt number.

For the purpose of validation of the present numerical method, Fig. 2 is plotted to show the comparison of the present computed results with the available results (8-10) in terms of the surface average Nusselt number for the non-MHD condition (*Ha* = 0) at *Pr* = 0.71 and *Re* in the range $10 \le Re \le 40$.

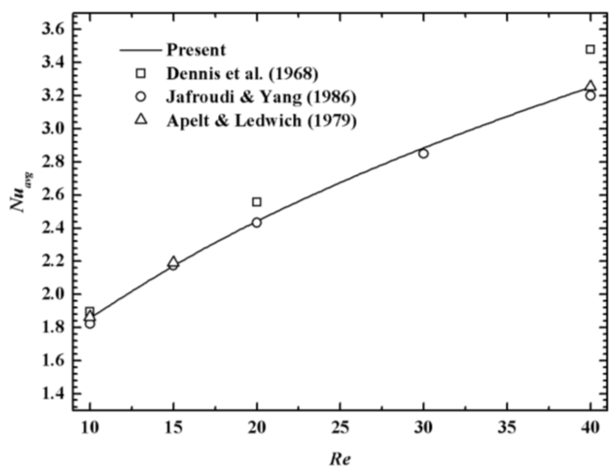


Figure 2.Variation of surface average Nusselt number with Reynolds number (*Re*) for non-MHD flow (*Ha* = 0), *Pr* = 0.71

IV. Results and Discussion

The evolution of the hydrodynamic phenomena after imposition of the external magnetic field along different directions are depicted in Fig. 3. It shows the streamlines for various Hartmann numbers corresponding to streamwise and transversely applied magnetic fields for *Pr* = 0.71 and at a Reynolds number *Re* = 50. Due to page limitation, some results are not presented in this manuscript.

It is well established that the two-dimensional instabilities can be suppressed by the application of the external magnetic field. The externally applied magnetic field i.e Lorentz force in magnetohydrodynamic flow acts like a damping force. This force can suppress any instability in the flow structure. The above fact of suppression of wake region is strongly established for different directions of applied magnetic fields presented in Fig. 3. The wake region progressively diminishes both longitudinally as well as laterally with the increase in the magnetic field strength. However, the nature of wake suppression quantitatively, depends on the direction of application of the external magnetic field. For stronger streamwise magnetic fields, the wake region is not reduced further due to the existence of a very slow moving region formed before the body known as "upstream wake". However, the wake region vanishes at larger Hartmann number for the transversely applied magnetic field.

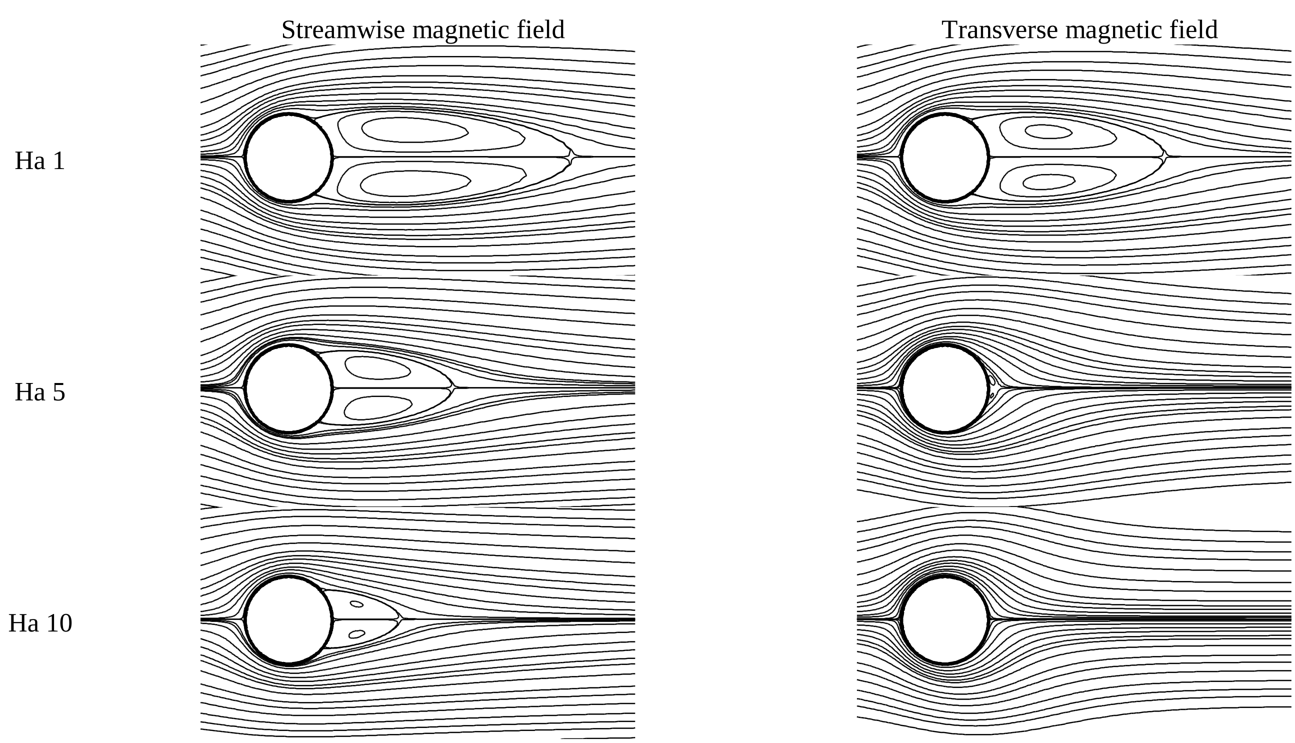


Figure 3. Streamlines around the cylinder for *Re* = 50 at different Hartmann numbers for streamwise magnetic field, B_x (left) and transverse magnetic field, B_y (right)

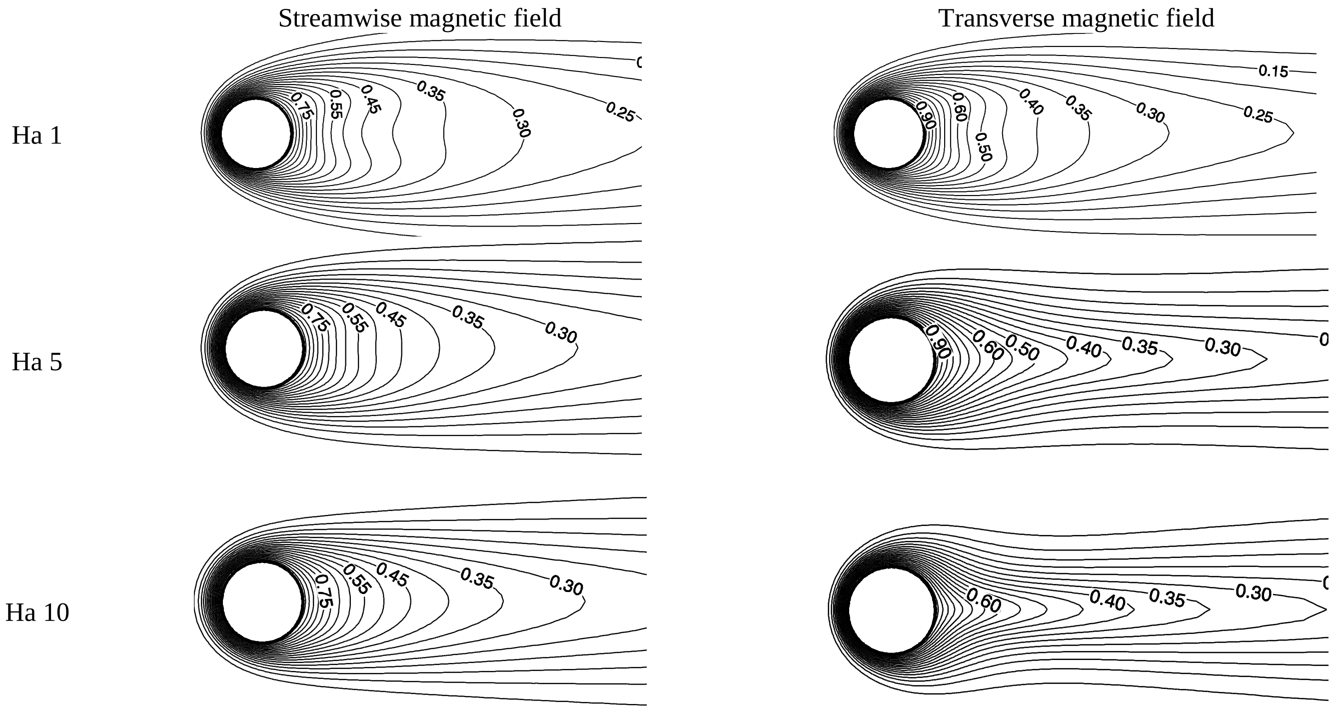


Figure 4. Isotherms around the cylinder for *Re* = 50 at different Hartmann numbers for streamwise magnetic field, B_x (left) and transverse magnetic field, B_y (right)

The isotherm profiles for different Hartmann numbers in Fig. 4a are not much affected by the streamwise magnetic field. However, very less effect of transverse magnetic field on isotherms are observed in Fig. 4b. The thermal transport is affected by the direction of the applied magnetic field. Since the wake region behind the cylinder is not reducing much for the streamwise magnetic field, the temperature contours remain almost similar for different magnetic field strength. For the transverse magnetic field, the major variation in the size of the wake region is observed and the temperature contours are elongated further downstream since the heat is convected away from the body. The thermal characteristics vary only because of the variation of flow pattern due to the application of the external magnetic fields.

The magnetic field strength for which the wake behind the obstacle completely disappears can be considered as the critical magnetic field. The hydrodynamic instability is found to be suppressed completely for transverse magnetic field for all considered Reynolds number. The corresponding critical Hartmann number (Ha_c) is plotted as a function of Reynolds number in Fig. 5 for the case of transversely applied magnetic fields.

Figure 6 shows the variation of surface average Nusselt number with Hartmann number for different directions of applied magnetic fields and Reynolds number. The Nusselt number increases as usual with the Reynolds number for both the cases of applied magnetic fields. Since the wake region decreases at a faster rate for the transversely applied magnetic field, the Nusselt number increases with magnetic field strength (Fig. 6b). However, the variation is not significant for the streamwise magnetic field (Fig. 6a).

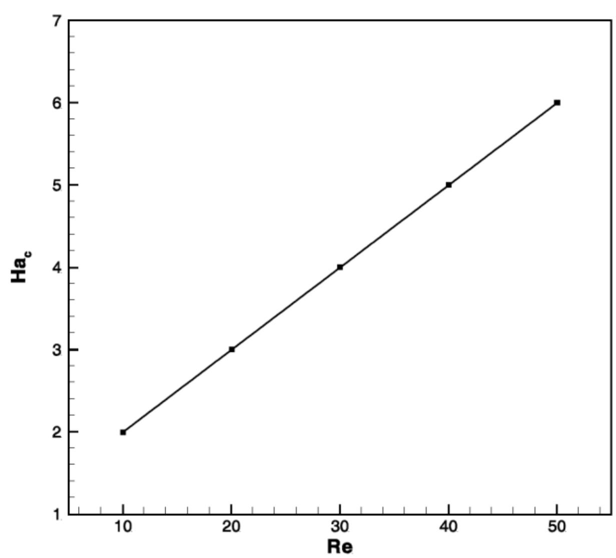
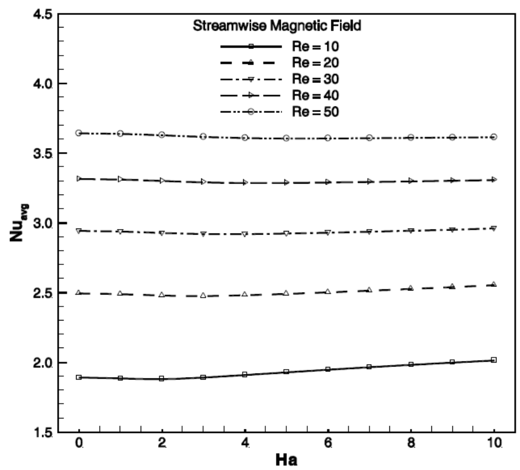
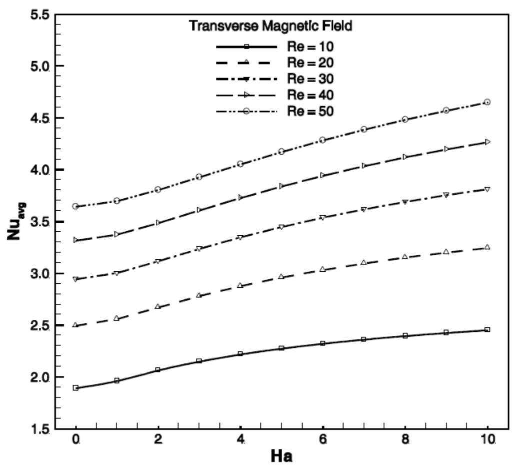


Figure 5 Critical Hartmann number as a function of Reynolds number for transversely applied magnetic field.



(a)



(b)

Figure 6. Variation of surface average Nusselt number with Hartmann number for (a) streamwise and (b) transverse magnetic fields for different Reynolds numbers

V. Conclusion

Numerical simulation is carried out to analyze the hydrodynamic and thermal transport phenomena for flow of an electrically conducting fluid over a circular obstacle in a two dimensional unconfined medium. The magnetic field is imposed uniformly either along the streamwise or transverse directions. The simulations are performed for *Re* range of 10-50 and Hartman number 0-10 considering air (*Pr* = 0.71) as the working fluid. The imposed transverse magnetic field effectively suppresses the hydrodynamic instability compared to streamwise field; because of the faster decay of wake region (bubble length and width) for the transverse magnetic field. The variation of surface average Nusselt number is not significantly changes with the Hartmann number for streamwise applied magnetic field. It increases significantly with Hartmann number for transverse magnetic field. The Nusselt number increases with Reynolds number for both the cases.

References

[1] Y. Kolesnikov and A. Tsinober, *Experimental investigation of two-dimensional turbulence behind a grid*, Fluid Dynamics Research, 1974. **9**: p. 621-624.

[2] G. Mutschke, V. Shatrov, and G. Gerbeth, *Cylinder wake control by magnetic fields in liquid metal flows*, Experimental Thermal and Fluid Science, 1998. **16**: p. 92-99.

[3] T.V.S. Sekhar, R. Sivakumar, H. Kumar, and T.V.R. Kumar, *Effect of aligned magnetic field on the steady viscous flow past a circular cylinder*, Applied Mathematical Modeling, 2007. **31**: p. 130-139.

[4] O.V. Andreev, and Y.B. Kolesnikov, *MHD instabilities at transverse flow around a circular cylinder in an axial magnetic field*, In Third International Conference on Transfer Phenomena in Magnetohydrodynamics and Electroconducting Flows, Aussois, France, 1997. p. 205-210.

[5] S.J. Kim, and C.M. Lee, *Investigation of the flow around a circular cylinder under the influence of an electromagnetic force*, Experiments in Fluids, 2000. **28**: p. 252-260.

[6] D. Chatterjee,, and K. Chatterjee, *Wall bounded flow and heat transfer around a circular cylinder at low Reynolds and Hartmann numbers*, Heat Transfer-Asian Research, 2012. DOI 10.1002/htj.21025.

[7] FLUENT 6.0 User's Guide, Fluent Inc., Lebanon, NH, 2001.

[8] S.C.R. Dennis, J.D. Hudson, and N. Smith, *Steady laminar forced convection from a circular cylinder at low Reynolds number*, Physics of Fluids, 1968. **11**, p. 933-940.

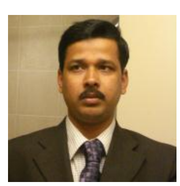
[9] H. Jafroudi, and H.T.Yang, *Steady laminar forced convection from a circular cylinder*, Journal of Computational Physics, 1986. **65**, p. 45-56.

[10] C.J. Apelt, and M.A. Ledwich, *Heat transfer in transient and unsteady flows past a circular cylinder in the range 1 < R < 40*, Journal of Fluid Mechanics, 1979. **95**, p. 761-777.

Mr. Satish Kumar Gupta is a final year under graduate student in the Mechanical Engineering Department of National Institute of Technology, Durgapur, India. His main research interests are computational fluid dynamics, heat transfer and flow over bluff obstacles

.

Dr. Dipankar Chatterjee is a Senior Scientist in CSIR-Central Mechanical Engineering Research Institute, India. Earlier he was associated with LPMI, Arts et Métiers Paris Tech, France as a post doctoral researcher. He received his PhD from the Department of Aerospace Engineering, Indian Institute of Technology Kharagpur, India. He has published 70 international journal and conference papers. His main interests are computational modeling of fluid flow and heat transfer over bluff obstacles, turbulence, phase change and reactive flow process modeling, lattice Boltzmann modeling and electromagneto-hydrodynamic interactions in macro and microflows.



Dr. Bittagopal Mondal is a Scientist in CSIR-Central Mechanical Engineering Research Institute, Durgapur, India. He has received his PhD degree in Mechanical Engineering from IIT Guwahati. Earlier he was associated with Politecnico Di Torino, Italy, University of Pretoria, South Africa, University of Waterloo, Canada. Dr. Mondal has about 38 research publications in peer reviewed journals and conferences. His main research areas are conjugate mode heat transfer, PEM fuel cell, two phase flow, wake dynamics, micro and nano fluids and lattice Boltzmann method.