Dr. Dipankar Chatterjee, **PhD**

Senior Scientist & Head
Assistant Professor, AcSIR
Simulation & Modeling Laboratory
CSIR-Central Mechanical Engineering Research Institute

RESEARCH AREAS:

- Computational modeling of fluid flow, heat transfer and species transport in reactive and phase changing systems.
- Numerical simulation of fluid flow and heat transfer over bluff bodies
- Simulation of hydromagnetic flows, electromagnetohydrodynamics and non-Newtonian flows
- Microscale transport and nanofluidics

CORRESPONDENCE DETAILS:

Present Address: Simulation & Modeling Laboratory,
CSIR-Central Mechanical Engineering Research Institute,
MG Avenue, Durgapur-713209, India
Phone: +91-343-6510455
Fax: +91-343-2548204
Mobile: +91 9477261796
E-mail: d_chatterjee@cmeri.res.in, Alternate E-mail: rsdchat@yahoo.co.in

Permanent Address: 23, Ram Chand Ghosh Road,
Post & Vill: Kodalia, Dist: 24 pgs(s)
Kolkata - 700146, India

EDUCATIONAL DETAILS:

Examination	Board/Council/Univ.	% of Marks	Y.O.P.
Ph. D (Engg.)	Indian Institute of Technology, Kharagpur	-	2006
M.E. (Mechanical)	Jadavpur University, Kolkata	85.07%	1999
B.E. (Mechanical)	Jadavpur University, Kolkata.	72.97%	1996
12^{th} Standard	West Bengal Council of Higher Secondary Education	74.30%	1991
10^{th} Standard	West Bengal Board of Secondary Education	78.89%	1989

PROFESSIONAL EXPERIENCES:

Name of the Organization	Designation	Period of Service	Nature of Experience
CSIR-CMERI, Durgapur	Senior Scientist & Head; Assistant Professor at AcSIR	11.11.2009 to date 06 Yrs.	Research
LPMI, Arts et Métiers Paris Tech, France	Post Doctoral Fellow	10.11.2008 to 09.11.2009 01 Yrs.	Research
Indian School of Mines University, Dhanbad	Assistant Professor	04.04.2008 to 31.10.2008 07 Mths.	Teaching/Research
Haldia Institute of Technology, Haldia	Professor	25.07.2007 to 02.04.2008 08 Mths.	Teaching/Research
Future Institute of Engineering and Management, Kolkata	Assistant Professor	14.02.2007 to 24.07.2007 05 Mths.	Teaching/Research
B.P. Poddar Institute of Management and Technology, Kolkata	Assistant Professor	02.08.2005 to 13.02.2007 01 Yrs. 06 Mths.	Teaching/Research
Indian Institute of Technology, Kharagpur	Senior Research Fellow	18.02.2005 to 27.07.2005 05 Mths.	Research
Indian Institute of Technology, Kharagpur	Senior Research Fellow	12.11.1999 to 16.01.2001 01 Yrs. 2 Mths.	Research
Usha Martin Industries Limited (Usha Alloy & Steel Division, Jamshedpur)	Graduate Engineer Trainee	01.07.1996 to 28.06.1997 01 Yrs.	Industrial
Total Experience	13 Yrs.		

MISCELLANEOUS:

- **Software Skills**
  - Operating system –Windows.
  - Programming language –FORTRAN.
  - Application software – ORIGIN, TECPLOT
  - Mesh Generation Package – **GAMBIT**
  - CFD Package – **FLUENT**

- **Date of Birth**
  - 22nd December 1972.

PhD TOPIC: Some aspects of phase change modeling in materials processing

Summary of PhD Work:

Numerical simulation of the phase change materials processing is considered as the central theme of the thesis. While some standard mathematical models in this regard have been well established, certain aspects of phase change modeling are yet to be extensively explored in materials processing applications, mainly because of the associated computational complexity. Primary aim of the work is to investigate some of these relatively unexplored areas, such as turbulence modeling in materials processing and Lattice Boltzmann simulation of melting/solidification problems.

As a first step, a generalized three dimensional transient turbulence modeling framework is developed, which incorporates morphological and thermodynamic features of solid-liquid phase change. The standard k-ε model is chosen as a basis, to start with, for its inherent simplicity and adaptability to common CFD codes. In order to assess the performance of the formulated turbulence model, the same is subsequently utilized to simulate a generic high power laser surface alloying process, where effects of turbulent transport can actually be realized.

In order to explore further possibilities of more sophisticated turbulence modeling for materials processing applications, a large eddy simulation (LES) scheme is formulated for modeling isothermal phase change processes in laser surface melting applications. While satisfactory results are obtained in this respect, and more detailed information are obtained with regard to capturing of larger eddies, it is observed to be computationally too costly to take into account all intrinsic aspects of materials processing applications.

For an order-of-magnitude assessment of predictions from the numerical model, a systematic approach is subsequently developed for scaling analysis of three-dimensional transient turbulent momentum, heat and mass transfer pertaining to the case of a typical laser surface alloying process.

In order to reduce high computational costs involved with the numerical simulation, a highly efficient parallel algorithm based on k-ε model is developed to simulate the three dimensional, transient, turbulent transport phenomena in case of a typical laser surface alloying process.

Concerned with the computational complexities associated with the conventional CFD approach for modeling complicated phase change problems in materials processing mentioned as above, the next effort is devoted to devise an entirely different elegant computational modeling approach based on the lattice-Boltzmann method.

To start with, a hybrid lattice Boltzmann scheme is developed initially to solve a conduction based phase change problem. The model essentially combines the classical Boltzmann transport equation with the conventional enthalpy formulation of phase change. The developed model is subsequently utilized to simulate formation of an industrial ice block and excellent agreement is found with the available results in the literature.

In order to further extend the present modeling philosophy for the case of generalized convection-diffusion phase change situation, a completely new lattice Boltzmann scheme is developed to numerically simulate real-life melting-solidification problems. The model is subsequently applied to test cases reported in benchmark studies of solid-liquid phase change modeling, such as problems involving melting of pure gallium in a rectangular cavity, crystal growth during equiaxed solidification of an undercooled melt and Rayleigh-Benard convection in presence of directional solidification of a freezing substance kept in a top-cooled enclosure and an excellent agreement is found with the numerical and experimental results available in the literature. It is, therefore, found to be an attractive and efficient proposition for numerical modeling of complicated phase change problems, over the years to come.

PUBLICATIONS:

	Published/Accepted	Under Review
Journal	**81**	**10**
Conferences	**33**	**01**
Books/Book Chapters	**03**	**-**
Technical Reports	**05**	**-**

Journal Papers

1. Numerical simulation of convective transport of fly ash-water slurry in horizontal pipe bends, Bibhuti Bhusan Nayak, **Dipankar Chatterjee**, Amar Nath Mullick (**Multiphase Science and Technology, Accepted**)

   [*A three-dimensional numerical study is conducted to analyze the thermo-fluidic transport associated with the flow of water-fly ash slurry in 90˚curved horizontal pipes having different radius ratios (ratio between radius of curvature and radius of pipe) of 2.98 and 5.6. The above phenomena in bends is also compared with straight pipe of same length and diameter for the flow of spherical fly ash particle of sizes 13 and 34 µm at velocities ranging between 1-5 m/s and particle concentrations within 10-50% by volume for each velocity. The simulation is carried out by deploying an Eulerian multiphase model available in the commercial Computational Fluid Dynamics (CFD) code Ansys Fluent. The pipe wall is kept at an isothermal condition of 400 K, whereas the slurry enters the pipeline at a temperature of 300 K. The results indicate that the pipes having bends enhance the heat transfer performance at the expense of the increased pressure drop compared to the straight pipes and also the pressure drop and heat transfer increase with decreasing radius of curvature due to the increase of the secondary flow in the pipe bends. It is also observed that the pressure drop is always greater when the slurry contains larger size particles. On the contrary, the heat transfer coefficient of the slurry having smaller size of particles is found more in comparison to the larger size particle slurry*.]

2. Thermo-magneto-convective transport around a square cylinder in a square duct under strong axial magnetic field, **Dipankar Chatterjee**, Satish Kumar Gupta (**Journal of Applied Fluid Mechanics**, vol. 9(4), **2016**)

   [*A quasi two-dimensional numerical study is performed to analyze the thermo-magneto-convective transport of liquid metal around a square cylinder in a square duct subjected to a strong externally imposed axial magnetic field. The channel bottom wall is considered heated while the top wall is maintained at the free stream temperature keeping the cylinder adiabatic. The Reynolds and Hartmann numbers are kept in the range* $0 < Re \le 6000$ *and* $0 < Ha \le 2160$. *The flow dynamics in the aforementioned range of parameters reveals the existence of four different regimes out of which the first three ones are similar to the classical non-MHD 2D cylinder wakes while the fourth one is characterized by the vortices evolved from the duct side walls due to the boundary layer separation which strongly disturbs the Kármán vortex street. The flow dynamics and heat transfer rate from the heated channel wall are observed to depend on the imposed magnetic field strength. With increasing magnetic field, the flow becomes stabilized resulting in a degradation in heat transfer. A special case at a very high Reynolds number* $Re = 3\times10^4$ *with Ha = 2160 is also considered to show the development of a Kelvin–Helmholtz-type instability that substantially affects the heat transfer rate.*]

3. Magnetoconvective Transport in a Lid-Driven Square Enclosure with Two Rotating Circular Cylinders, **Dipankar Chatterjee**, Pabitra Halder (**Heat Transfer Engineering,** vol. 37(2), pp. 198-209, **2016**)

[*Numerical analysis of hydromagnetic mixed convective transport in a differentially heated vertical lid-driven square enclosure is performed in this article. Two heat conducting solid circular cylinders of identical shape are placed within the enclosure. The cylinders may be either stationary or rotating about their centroidal axes. Two different combinations of cylinder rotations are considered, viz., one when the top cylinder rotates and the bottom one is kept stationary and the other when the bottom cylinder rotates and the top one is kept stationary. Simulations are performed for a range of controlling parameters such as the Richardson number (1 to 10), Hartmann number (0 to 50) and dimensionless rotational speed (0 to 5) keeping the Reynolds number based on the lid velocity fixed as 100. Furthermore, three different Prandtl numbers 0.02, 0.71 and 7 are considered. The flow and thermal fields are analyzed through streamline and isotherm plots for various Hartmann and Richardson numbers and rotational speeds. Additionally, the Nusselt number and the bulk fluid temperature are also computed to understand the effects of Hartmann and Richardson numbers and rotational speeds on them. It is observed that the heat transfer greatly depends on the rotational speed of the cylinder.*]

4. Unconfined hydromagnetic flow and heat transfer around a circular cylinder at low Reynolds numbers, **Dipankar Chatterjee,** Kanchan Chatterjee, Bittagopal Mondal, Nirmal Baran Hui (**CFD Letters, Accepted**).

[*A two-dimensional numerical simulation is carried out to analyze the fluid flow and forced convection heat transfer around a circular obstacle at low Reynolds and Hartmann numbers. The cylinder is placed in an infinite 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. The simulation is carried out for the range of Reynolds number* $10 \le Re \le 50$ *with Hartmann number* $0 \le Ha \le 10$ *and with a fixed Prandtl number, Pr = 0.02 (liquid metal). The flow is steady and stable for the above range of conditions. The magnetic field provides additional stability to the flow as a result of which the wake region behind the cylinder reduces with increasing magnetic field strength. The critical magnetic field strength is also computed for which the separation is completely suppressed for the Reynolds number range in case of transversely applied magnetic field. The rate of heat transfer is found almost invariant at low Reynolds number whereas it increases slightly for higher Reynolds number with the applied magnetic field. The heat transfer increases as usual with the Reynolds number for all Hartmann numbers.*]

5. Effect of Prandtl number and rotation on vortex shedding behind a circular cylinder subjected to cross buoyancy at subcritical Reynolds number, **Dipankar Chatterjee**, Chiranjit Sinha (**International Communications in Heat and Mass Transfer**, **Accepted**).

[*We perform a two-dimensional numerical simulation following a finite volume approach to understand the vortex shedding (VS) phenomena around a circular cylinder subjected to cross thermal buoyancy at a subcritical Reynolds number, Re = 40. The flow is considered in an unbounded medium. The cylinder may either be stationary or rotating about its centroidal axis. At the subcritical Reynolds number the flow and thermal fields are steady without the superimposed thermal buoyancy (i.e. for pure forced flow). However, as the buoyancy parameter (Richardson number, Ri) increases, flow becomes unstable and eventually, at some critical value of Ri, periodic VS is observed to characterize the flow and thermal fields. An extended Stuart-Landau model is used in this work for the accurate quantitative estimation of the critical Richardson number for the onset of VS. The above phenomena of VS with imposed buoyancy is strongly dependent on the type of the fluid being used. We quantify here the minimum heating requirement for the initiation of VS by choosing three different types of fluids having Prandtl numbers, Pr = 0.71, 7 and 100. The dimensionless rotational speed* (Ω) *ranges between 0-4. It is revealed that as Pr increases, heating requirement also increases for the initiation of VS. A possible explanation for the observation is provided.*]

6. Transient mixed convection heat transfer around three isothermal square cylinders in cross-flow at low Reynolds numbers, **Dipankar Chatterjee**, Shyama Prasad Das (**Journal of Energy, Heat and Mass Transfer**, **Accepted**).

[*The objective of this work is to numerically investigate the influence of cross buoyancy on the flow and heat transfer characteristics around three equal isothermal square cylinders arranged in a staggered fashion in an unconfined medium. Transient two-dimensional numerical simulations are performed in this regard with a finite volume code based on the SIMPLEC algorithm in a collocated grid system. The pertinent dimensionless parameters, such as the Reynolds and Richardson numbers are considered in the range of* $50 \le \mathrm{Re} \le 150$, *and* $0 \le \mathrm{Ri} \le 5$. *The representative streamlines, vortex structures and isotherm patterns are presented and discussed. It is observed in general that the introduction of thermal buoyancy causes drastic change in the flow and thermal field patterns. In addition, the overall drag and lift coefficients, dimensionless vortex shedding frequency and average Nusselt numbers are determined to elucidate the effects of Reynolds and Richardson numbers on the flow and heat transfer characteristics. The heat transfer rate increases monotonically with Re and Ri, so also the vortex shedding frequency.*]

7. Influence of Aiding Buoyancy on the Suppression of Flow Separation for Power-Law Fluids around a Circular Object, Satish Kumar Gupta, Sudipta Ray, **Dipankar Chatterjee** (**Heat Transfer Engineering,** vol. 37 (15), **2016**).

[*Numerical computations are performed to analyze the influence of aiding thermal buoyancy on the phenomenon of suppression of flow separation in power-law fluids around a circular object. The idea has been borrowed from some recent similar works in Newtonian fluids. Owing to the contradictory behavior of shear-thinning and shear-thickening fluids in regard to the separation mechanism, we intend to understand the role of superimposed thermal buoyancy on the suppression phenomena in non-Newtonian power-law fluids, for which a range of power-law indices (0.4 to 1.8) is considered. The Reynolds numbers are kept intentionally low, within 10 to 40, such that the isothermal flow remains steady and separated without imposition of thermal buoyancy. The buoyancy causes a delay in the separation, thereby affecting the suppression phenomena. We determine the critical heating parameter (Richardson number) for the complete suppression of the flow separation and from there we construct a bifurcation diagram to show the typical flow regime evolved due to the complex interplay between the aiding thermal buoyancy and fluid rheology. The Richardson number in the simulation lies in the range 0 to 0.35 keeping the Prandtl number fixed at 50. The heat transfer rates from the object are also obtained and important inferences are drawn in regard to the inhibition/augmentation of heat transfer due to fluid rheology.*]

8. Numerical Investigation of the Water Droplet Transport in a PEM Fuel Cell with Serpentine Flow Channel, Bittagopal Mondal, **Dipankar Chatterjee** (**Journal of Applied Fluid Mechanics,** vol. 9(4), **2016**).

[*The serpentine flow channel can be considered as one of the most common and practical channel layouts for a polymer electrolyte membrane fuel cell (PEMFC) since it ensures an effective and efficient removal of water produced in a cell with acceptable parasitic load. Water management is one of the key issues to improve the cell performance since at low operating temperatures in PEMFC, water vapor condensation starts easily and accumulates the liquid water droplet within the flow channels, thus affecting the chemical reactions and reducing the fuel cell performance. In this article, a comprehensive three dimensional numerical simulation is carried out to understand the water droplet mobility in a serpentine gas flow channel for a wide range of surface properties, inlet air velocities, droplet positions (center or off-center, bottom or top) and droplet sizes by deploying a finite volume based methodology. The liquid-gas interface is tracked following the volume-of-fluid (VOF) method. The droplet transport is found to be greatly influenced by the surface wettability properties, inlet velocities, number of droplets emerged and initial droplet positions. Super hydrophobic surface property is not always preferable for designing the gas flow channels. It depends upon the inlet velocity conditions, droplet positions, number of droplets and surface properties.*]

9. Numerical analysis of convective transport of fly ash-water slurry through a horizontal pipe, Bibhuti Bhusan Nayak, Satish Kumar Gupta, **Dipankar Chatterjee**, Amar Nath Mullick (**Journal of Computational Multiphase Flows**, vol. 7(2), pp. 79-96, **2015**).

[*The thermal transport of solid-liquid suspension under turbulent flow condition is not well understood because of the complex interaction between the solid particles and the turbulent carrier fluid. The solid* *particles may enhance or suppress the rate of heat transfer and turbulence depending on their size and concentration. In the present paper, a three-dimensional numerical simulation is carried out in order to study the pressure drop and heat transfer characteristics of a liquid-solid slurry flow in a horizontal pipe. The simulation is performed by using the algebraic slip mixture (ASM) model which is a part of the finite-volume based CFD software Ansys Fluent. The turbulence is handled by the RNG* $k-\varepsilon$ *model. A hexagonal shape and cooper type non-uniform three-dimensional grid is created to discretize the computational domain. Spherical fly ash particles, with mass median diameter of* $13\,\mu\text{m}$ *for an average flow velocity ranging from 1-5 m/s and particle concentrations within 0-40% by volume for each velocity, are considered as the dispersed phase. The results illustrate that higher particle concentration in the flow causes an increase in the heat transfer and pressure drop. Moreover, both heat transfer and pressure drop are seen to show a positive dependence on the mean velocity of the flow.*]

10. MHD flow and heat transfer behind a square cylinder in a duct under strong axial magnetic field, **Dipankar Chatterjee,** Satish Kumar Gupta (**International Journal of Heat and Mass Transfer**, vol. 88, pp. 1-13, **2015**)

[*We capture through numerical simulation the MHD (magnetohydrodynamic) vortex dynamics around a square object in a square duct subjected to a strong externally imposed axial magnetic field. A quasi two-dimensional conditionality allows us to follow a two-dimensional modeling approach. The pertinent MHD control parameters such as the Reynolds and Hartmann numbers are kept in the range* $0 < Re \leq 6000$ *and* $0 < Ha \leq 2160$*, respectively. The various regimes of the MHD wake are found in-line with those obtained by Dousset and Pothérat under similar conditions using a circular object [Phys. Fluids 20 (2008) 017104]. Four different regimes are identified out of which the first three regimes are the classical non-MHD 2D cylinder wakes. The transition from one to another regime is controlled by the friction parameter Re/Ha. The fourth regime is characterized by the vortices evolved from the duct side walls due to the boundary layer separation which strongly disturbs the Kármán vortex street. In order to explore the thermal transport phenomena under the action of the axially imposed magnetic field, the channel bottom wall is considered heated while the top wall is maintained at the free stream temperature keeping the cylinder adiabatic. The heat transfer rate from the heated channel wall strongly depends on the imposed magnetic field strength as well as Reynolds number. Additionally, an enhancement in heat transfer is experienced by placing the square cylinder in the channel over the bare channel, however, simultaneously a degradation in heat transfer would occur if the square object is replaced by a same size circular object.*]

11. Dynamic behavior of flow around rows of square cylinders kept in staggered arrangement, **Dipankar Chatterjee**, Gautam Biswas (**Journal of Wind Engineering and Industrial Aerodynamics,** vol. 136, pp. 1-11, **2015**).

[*A two-dimensional numerical study is carried out to understand the vortex dynamics behind two rows of square cylinders arranged in a staggered fashion in an unbounded medium and at low Reynolds number (Re = 100). A uniform cross flow of an incompressible fluid is considered. The transverse spacing between the cylinders is varied (S/d = 1, 2, 3 and 5; with S and d are the transverse spacing and cylinder size) while the streamwise gap is fixed at L/d = 1 (L being the streamwise gap). Computations are performed by a finite volume based CFD solver using PISO algorithm. An unsteady two-dimensional laminar model is used for all the computations. Effect of transverse spacing on flow characteristics and global fluid dynamic parameters are discussed. Furthermore, the establishment of the low-dimensional chaotic nature of the flow is predicted for relatively small transverse spacing of the cylinders. The transition to chaos is manifested through a quasi-periodic route. The quasi-periodic route to chaos is established through different characterization tools, such as the spectra, autocorrelation function, time-delay reconstruction and the Poincare section. The chaotic behavior of the flow system is quantified through the calculation of the Lyapunov exponent and fractal dimension.*]

12. Investigation of Mixed Convection in a ventilated cavity in the presence of a heat conducting circular cylinder, Satish Kumar Gupta, **Dipankar Chatterjee**, Bittagopal Mondal (**Numerical Heat Transfer A,** vol. 97, pp. 1-23, **2015**).

[*The study is aimed to investigate the mixed convective transport within a ventilated square cavity in presence of a heat conducting circular cylinder. The fluid flow is imposed through an opening at the bottom of the left cavity wall and is taken away by a similar opening at the top of the right cavity wall. The cylinder is placed at the centre of the cavity. Two cases are considered depending on the thermal conditions of the* *cavity walls. In the first case the left and right vertical walls are kept isothermal with different temperatures and the top and bottom horizontal walls are considered as thermally insulated. For the second case, the top and bottom walls are maintained at different constant temperatures and the left and right walls are considered adiabatic. Heat transfer due to forced flow, thermal buoyancy and conduction within the cylinder are taken into account. Effect of the cylinder size (* $0.1 \le D \le 0.5$ *) and the solid-fluid thermal conductivity ratio (* $0.1 \le K \le 10$ *) are explored for various values of Richardson number (* $0 \le Ri \le 5$ *) at fixed Reynolds (Re = 100) and Prandtl (Pr = 0.71) numbers. The fluid dynamic and thermal transport phenomena are depicted through streamline and isotherm plots. Additionally, the global thermal parameters such as the average Nusselt number and average fluid temperature of the cavity are presented. It is found that the aforementioned parameters have significant influences on the fluid flow and heat transfer characteristics in the cavity.*]

13. Effect of thermal buoyancy on fluid flow and heat transfer across a semicircular cylinder in cross flow at low Reynolds numbers, **Dipankar Chatterjee,** Bittagopal Mondal (**Numerical Heat Transfer A,** vol. 67, pp. 436-453, **2015**).

    [*This paper presents a two-dimensional numerical study on the fluid flow and mixed convection heat transfer around two equal isothermal square cylinders placed in a tandem arrangement and subjected to the cross flow of a Newtonian fluid at moderate Reynolds numbers. The spacing between the cylinders is varied by changing the gap to cylinder size ratio as* $S/d = 1, 2, 3, 4, 5, 7$ and 10. *The Reynolds number is considered in the range* $50 \le Re \le 150$. *The mixed convection effect is studied for Richardson number range of 0-2 and the Prandtl number is chosen constant as 0.71. The flow is considered in an unbounded medium; however, fictitious confining boundaries are chosen to make the problem computationally feasible. Numerical calculations are performed by using a PISO algorithm based finite volume solver in a collocated grid system. The effect of superimposed thermal buoyancy on flow and isotherm patterns are presented and discussed. The global flow and heat transfer quantities such as overall drag and lift coefficients, local and surface average Nusselt numbers and Strouhal number are calculated and discussed for various Reynolds, Richardson numbers and spacing ratios. The notable contribution is the quantification of the critical spacing ratio which is observed to decrease with increasing thermal buoyancy effect for a specific Reynolds number.*]

14. Influence of an adiabatic square cylinder on hydrodynamic and thermal characteristics in a two-dimensional backward-facing step channel, **Dipankar Chatterjee**, Amrita Sengupta, Nandini Debnath, Sudipta De (**Heat Transfer Research**, vol. 46, pp. 63-89, **2015**).

    [*The work demonstrates numerically the influence of an adiabatic cylinder of square cross section on the flow and heat transfer characteristics within a two-dimensional backward-facing step channel. The study conducted for a Reynolds number range of 1-200 (in the steady laminar regime) with a fixed Prandtl number 0.71 (air). Additionally, simulations are also conducted for Reynolds numbers of* 10^4 *and* 2×10^4 *(unsteady turbulent regime). The numerical simulation is performed by deploying a finite volume based commercial solver Fluent. Different cross stream positions of the cylinder are considered to show the positional effect of the obstacle on the overall flow and heat transfer pattern. The flow and thermal fields are demonstrated through the streamlines and isotherms profiles for different Reynolds numbers and cylinder positions. The global flow and heat transfer quantities such as the reattachment length, drag coefficient, local and average Nusselt number are computed and presented. A significant heat transfer augmentation due to the presence of the cylinder is observed as compared to the unobstructed case (without a cylinder). Furthermore, it is observed that the heat transfer augmentation is more for the case of a square cylinder in comparison to a circular cylinder introduced within the same channel. Finally, correlations are devised for the drag coefficient and Nusselt number for the above range of conditions.*]

15. Mixed convection heat transfer from an equilateral triangular cylinder at low Reynolds numbers, **Dipankar Chatterjee**, Bittagopal Mondal (**Heat Transfer Engineering**, vol. 36, 123-133, **2015**).

    [*A two dimensional numerical study is undertaken to investigate the influences of cross buoyancy on the vortex shedding phenomena behind a long heated equilateral triangular cylinder for the low Reynolds number laminar regime. The flow is considered in an unbounded medium, however, fictitious confining boundaries are chosen on the lateral sides to make the problem computationally feasible. Numerical calculations are performed by using a finite volume method based on the PISO (Pressure-Implicit with*

Splitting of Operators) algorithm in a collocated grid system. The range of Reynolds number is chosen to be 10-100 with a fixed Prandtl number, 0.71. The mixed convection effect is studied for the Richardson number range of 0-1. The effect of superimposed thermal buoyancy on flow and isotherm patterns are presented and discussed. The global flow and heat transfer quantities such as the overall drag and lift coefficients, local and surface average Nusselt numbers and Strouhal number are calculated and discussed for various Reynolds and Richardson numbers.]

16. Convective transport around a rotating square cylinder at moderate Reynolds numbers, **Dipankar Chatterjee**, Satish Kumar Gupta (**Numerical Heat Transfer A,** vol. 67, pp. 1386-1407, **2015**).

[*Two-dimensional numerical simulation is performed to analyze the thermofluidic transport around a rotating square cylinder in an unconfined medium. The convective transport originates as a consequence of the interaction between a uniform free stream flow and the flow evolving due to the rotation of the sharp edged body. A finite volume based method and a body fitted grid system along with moving boundaries are used to obtain the numerical solution of the incompressible Navier-Stokes and energy equations. The Reynolds number based on the free stream flow is considered in the range* $10 \le Re \le 200$ *and the dimensionless rotational speed of the cylinder is kept* $0 \le \Omega \le 5$*. Depending on the Reynolds number and rotational speed of the cylinder the transport characteristics change. For the range* $10 \le Re < 50$ *the flow remains steady irrespective of the rotational speed. In the range* $50 \le Re \le 200$*, regular low frequency Kármán vortex shedding is observed up to a critical rate of rotation (*Ω_{cr}*). Beyond* Ω_{cr}*, the global convective transport shows steady nature. The rotating circular cylinder also shows likewise degeneration of Kármán vortex shedding at some critical rotational speed. However, the heat transfer behavior varies significantly with a rotating circular cylinder. Such thermofluidic transport around a spinning square in an unconfined free stream flow is reported for the first time.*]

17. Forced convection heat transfer in power-law fluids around a semicircular cylinder at incidence, Satish Kumar Gupta, Sudipta Ray, **Dipankar Chatterjee** (**Numerical Heat Transfer A,** vol. 67, pp. 952-971, **2015**).

[*The two-dimensional steady flow and convective heat transfer of power-law fluids past a semicircular cylinder are investigated in the reported work. The heated semicircular cylinder is placed in an unconfined domain at different angles facing the incoming free stream flow of power-law fluids having a generalized Prandtl number Pr = 100. Particular emphasis is given to study the effect of angle of incidence (*$0 \le \alpha \le 180^\circ$*) on the fluid dynamics and thermal transport around the semicircular object for varying Reynolds number (*$10 \le Re \le 40$*) and power-law index (*$0.4 \le n \le 1.8$*). A finite volume based method is adopted for the numerical computation. The flow and heat transfer phenomena are visualized through the streamline and isotherm profiles at various operating conditions. Also, the pressure coefficient, drag coefficient and Nusselt number on the surface of the object are presented and discussed.*]

18. Convective transport around rows of square cylinders arranged in a staggered fashion at moderate Reynolds number, **Dipankar Chatterjee**, Satish Kumar Gupta (**Numerical Heat Transfer A,** vol. 68, pp. 388-410, **2015**)

[*The thermo-hydrodynamic interactions among multiple bodies fixed in an incident flow are analyzed through two-dimensional numerical computation. The bodies are identical in shape and size with square cross-section and arranged in two rows in a staggered fashion within an unconfined domain. Simulation is carried out using a finite volume based method considering a uniform cross flow of air (Prandtl number, Pr = 0.71) at a moderate Reynolds number (Re = 100). Apart from the Reynolds number, certain geometrical parameters such as the stream-wise and transverse spacings of the objects may significantly influence the wake dynamics, vortex structure formation and the associated thermal transport. Accordingly, both the dimensionless transverse spacing (S/d = 1, 3 and 5; with S and d are the transverse spacing and cylinder size) and the non-dimensional stream-wise gap (L/d = 1, 3 and 5, with L being the stream-wise gap) are varied to elucidate their roles on controlling the hydrodynamic and thermal transports. Even at such a moderate Reynolds number, the flow and thermal fields show chaotic behavior at smaller transverse spacing. The chaotic behavior is established through various chaos characterization tools. However, at larger spacing, the usual unsteady vortex dynamics persists. The shedding in the gap is inhibited at smaller* *stream-wise spacing. Once again, at larger spacing usual shedding characteristics continue. The average heat transfer from the cylinders is found more at smaller stream-wise and transverse spacings.*]

19. Magneto-convective transport of nanofluid in a vertical lid-driven cavity including a heat conducting rotating circular cylinder, Suraj Bansal, **Dipankar Chatterjee** (**Numerical Heat Transfer A,** vol. 68, pp. 411-431, **2015**)

    [*Numerical simulations are performed for the two-dimensional magneto-convective transport of Cu-H_2O nanofluid in a vertical lid-driven square cavity in the presence of a heat conducting and rotating circular cylinder. The left wall of the cavity is allowed to translate at a constant velocity in the vertically upward direction. Both the left and right walls are maintained at isothermal but different temperatures. The top and the bottom walls of the enclosure are made thermally insulated. At the central region of the cavity, there exists a heat conducting circular cylinder which can rotate in either clockwise or counter-clockwise direction. A constant horizontal magnetic field of amplitude B_0 is applied perpendicular to the vertical walls. The nanofluid is electrically conducting, while the solid walls are considered electrically insulated. Simulations are performed for various controlling parameters such as the Richardson number $(0.01 \le Ri \le 10)$, Hartmann Number $(0 \le Ha \le 50)$, dimensionless rotational speed of cylinder $(\Omega = \pm 1)$ and nanoparticle concentration $(0 \le \varphi \le 0.3)$, while the Reynolds number based on the lid velocity is kept fixed to a particular value $(Re = 100)$. The flow and thermal fields are found quite susceptible to the changes in the magnetic field and mixed convective strengths as well as nanoparticle concentrations. However, the direction of cylinder rotation is observed to have a little or no influence quantitatively on the global hydrodynamic and thermal parameters.*]

20. Cold flow simulation in underground coal gasification (UCG) cavities, **Dipankar Chatterjee**, Satish Kumar Gupta, Chebolu Aravind, Rakesh Roshan (**Journal of Advanced Thermal Science Research,** vol. 1, pp. 15-24, **2014**)

    [*Underground coal gasification (UCG), in the recent years, have gathered a significant amount of interest from the researchers because of its advantages over conventional mining and utilization techniques. It is one of the most promising and innovative technology where coal is gasified in-situ by injection of a suitable oxidant for the production of synthetic gas. The simultaneous occurrence of several phenomena such as complex flow patterns, chemical reactions, water influx, thermo-mechanical failure of the coal seam etc. make the mathematical modeling of the entire UCG process very abstruse and computationally challenging. The reaction between the oxidant and the coal in the deep underground seams leads to the formation of combustible gas and subsequently results in a cavity. As the gasification proceeds the cavity grows three dimensionally in a non-linear fashion. The cavity size strongly depends on several parameters like position and orientation of the inlet nozzle, coal properties etc. A comprehensive three-dimensional numerical study is conducted to understand the hydrodynamics within a given cavity size which would give us a relatively quick but reliable insights into the process. Five different cavity sizes are considered inside which the complete turbulent transport is simulated. Apart from the usual vertical and horizontal injection, the effect of inclined injection on the hydrodynamics is also reported here for the first time.*]

21. Control of boundary layer separation over a triangular surface by superimposed thermal buoyancy, **Dipankar Chatterjee**, Sudipta Ray (**International Journal of Heat and Mass Transfer,** vol. 79, pp. 769-782, **2014**).

    [*We endeavour here to elucidate the role of the superimposed thermal buoyancy on the boundary layer separation over a two-dimensional triangular surface. Particular emphasis is given to analyze the response of different orientations of the triangular object with respect to the incoming flow under the action of aiding/opposing thermal buoyancy. The object is placed in a vertical unconfined domain with two different orientations, one when the apex of the object is facing the flow (C1) and the other when one of the bases of the object is exposed to the incoming fluid (C2). A similar study by Chatterjee and Mondal [Int. J. Heat Mass Transfer 72 (2014) 128-138] considering circular and square shaped objects reveals that the steady, laminar and separated flow over the objects at low Reynolds numbers can be degenerated to an attached flow under the action of aiding thermal buoyancy. However, unlike circular/square bodies, the triangular body shows significant deviations in the separation characteristics. The present effort aims at numerically obtaining the critical heating parameters for which the separated boundary layer on the triangular object can be suppressed and analyzing the influence of the object orientation on the thermally induced suppression phenomena. Furthermore, the opposing buoyancy is known to trigger the vortex shedding* *process at low Reynolds numbers which is already established for circular/square objects. This triggering of vortex shedding over different orientations of a triangular object under the action of opposing buoyancy is numerically demonstrated. The Reynolds number is kept in the range* $5 \le Re \le 30$ *keeping the Prandtl number fixed at Pr = 50 with varying Richardson number. The critical Richardson numbers for the onset of flow suppression as well as the complete suppression of flow separation and the critical Richardson number for the onset of vortex shedding are obtained for the two different orientations of the object. Important inferences are drawn on the fluid dynamic and thermal transport characteristics focussing the separation phenomena. It is observed that the configuration C1 needs more heating than C2 for flow suppression. Also, C1 needs more cooling than C2 for the initiation of the vortex shedding. Such quantification in regard to the critical heating parameters for flow suppression and triggering of vortex shedding over a triangular object is reported for the first time.*]

22. Numerical study of the laminar flow past a rotating square cylinder at low spinning rates, **Dipankar Chatterjee,** Satish Kumar Gupta (**Journal of Fluids Engineering Trans ASME,** vol. 137, pp. 021204, **2014**).

[*The fluid dynamic interaction between a uniform free stream flow and the rotation induced flow from a sharp edged body is numerically investigated. A two-dimensional finite volume based computation is performed in this regard to simulate the laminar fluid flow around a rotating square cylinder in an unconfined medium. Body fitted grid system along with moving boundaries is used to obtain the numerical solution of the incompressible Navier-Stokes equations. The Reynolds number based on the free stream flow is kept in the range* $10 \le Re \le 200$ *with a dimensionless rotational speed of the cylinder in the range* $0 \le \Omega \le 5$. *At low* $Re = 10$, *the flow field remains steady irrespective of the rotational speed. For* $50 \le Re \le 200$, *regular low frequency Kármán vortex shedding is observed up to a critical rate of rotation (*Ω_{cr}*). Beyond* Ω_{cr} *, the global flow shows steady nature, although, high frequency oscillations in the aerodynamic coefficients are present. The rotating circular cylinder also shows likewise degeneration of Kármán vortex shedding at some critical rotational speed. However, significant differences can be seen at higher rotation. Such fluid dynamic transport around a spinning square in an unconfined free stream flow is reported for the first time.*]

23. Wall confined flow and heat transfer around a square cylinder at low Reynolds and Hartmann numbers, **Dipankar Chatterjee,** Kanchan Chatterjee, Bittagopal Mondal, Nirmal Baran Hui (**Heat Transfer-Asian Research**, vol. 43, pp. 459-475, **2014**).

[*This paper discusses the results obtained through a two-dimensional numerical simulation following a finite volume approach on the forced convection heat transfer for the hydromagnetic flow around a square cylinder at low Reynolds and Hartmann numbers. The magnetohydrodynamic (MHD) flow of a viscous incompressible and electrically conducting fluid is assumed to take place in a rectangular channel subjected to externally imposed magnetic fields and the cylinder is fixed within the channel. The magnetic fields may be applied either along the streamwise or transverse directions. Simulations are performed for the range of kinetic Reynolds number* $10 \le Re \le 60$ *with Hartmann number* $0 \le Ha \le 15$ *and for different thermal Prandtl numbers, Pr = 0.02 (liquid metal), 0.71 (air) and 7 (water) for a blockage ratio* $\beta = 0.25$. *A steady flow can be expected for the above range of conditions. Besides the channel wall, the magnetic field imparts additional stability to the flow as a consequence of which the recirculation region behind the obstacle reduces with increasing magnetic field strength for a particular Re. The critical Hartmann numbers for the complete suppression of flow separation in the case of a transversely applied magnetic field are computed. The rate of heat transfer is found almost invariant at low Re whereas it increases moderately for higher Re with the applied magnetic field. The heat transfer increases in general with the Reynolds number for all Hartmann numbers. Finally, the influence of obstacle shape on the thermohydrodynamic quantities is noted* .]

24. MHD mixed convection in a lid-driven cavity including heat conducting circular solid object and corner heaters with Joule heating, Sudipta Ray, **Dipankar Chatterjee** (**International Communications in Heat and Mass Transfer,** vol. 56, 152-158, **2014**).

[*A two-dimensional finite-volume based numerical approach is adopted to solve the hydro-magnetic mixed convection in a horizontal lid-driven square cavity with a circular solid object. The object is located centrally within the enclosure. Part of the bottom and the right walls are taken to be constant temperature* *heaters. The top boundary of the cavity is moving with uniform speed and kept at a constant temperature lower than the heaters. All other boundary segments are considered thermally insulated. For the simulation purposes, the corner heater sizes are altered* $(XL = YL = 0.25, 0.50, 0.75)$*, keeping the overall sizes of the cavity and the solid object fixed. The mixed convection is studied for the Richardson number range (*$0.1 \le Ri \le 10$*) keeping the Reynolds number based on the lid velocity fixed as* $Re = 100$*. The magnetic effect is brought in by varying the Hartmann number (*$0 \le Ha \le 50$*) along with the variation of Joule heating parameter (*$0 \le J \le 5$*). The results indicate a major influence of the prevailing convection method and the applied magnetic field on the flow as well as the thermal field, while the effect of Joule heating is found to be of very small significance.*]

25. Dual role of thermal buoyancy in controlling boundary layer separation around bluff obstacles, **Dipankar Chatterjee** (**International Communications in Heat and Mass Transfer,** vol. 56, 152-158, **2014**).

[*We establish through numerical simulation a dual role played by the superimposed thermal buoyancy in controlling the boundary layer separation around bluff obstacles. The work essentially demonstrates the influence of superimposed thermal buoyancy on flow around bluff obstacles of circular and square cross sections in aiding/opposing and cross buoyancy configurations. For the aiding/opposing configuration we show two phenomena such as the suppression of flow separation which occurs at relatively low Reynolds numbers (10-40) and the suppression of vortex shedding at a moderate range of Reynolds numbers (50-150). In the cross buoyancy configuration, the initiation of vortex shedding by the introduction of thermal buoyancy is shown at relatively low Reynolds numbers (10-40). Hence, depending on the direction of interaction with the free stream flow, the buoyancy sometimes stabilizes the flow and sometimes destabilizes the flow. Accordingly, there is a dual role of superimposed thermal buoyancy in controlling the boundary layer separation around bluff obstacles. Such duality cannot be observed in case of other agents such as rotation, magnetic force which also control the boundary layer separation around bluff obstacles.*]

26. Mixed convective transport in a lid-driven cavity containing a nanofluid and a rotating circular cylinder at the center, **Dipankar Chatterjee,** Satish Kumar Gupta, Bittagopal Mondal (**International Communications in Heat and Mass Transfer,** vol. 56, 71-78, **2014**).

[*The mixed convective transport of Cu-*H_2O *nanofluid in a differentially heated and lid-driven square enclosure in presence of a rotating circular cylinder is investigated numerically. The top wall of the enclosure is sliding from left to right at a uniform speed while all other walls are stationary. A thermally insulated circular cylinder is placed centrally within the enclosure. The cylinder can rotate about its centroidal axis. The top and bottom walls are kept isothermal at different temperatures while the side walls are assumed adiabatic. Simulations are performed for, Richardson number* $1 \le Ri \le 10$*, dimensionless rotational speed* $0 \le \Omega \le 5$ *and nanoparticle concentration* $0 \le \phi \le 0.20$ *keeping the Grashof number fixed as* $Gr = 10^4$*. The flow and thermal fields are analyzed through streamline and isotherm plots for various* Ω *and* Ri*. Furthermore, the drag coefficient of the moving lid and Nusselt number of the hot wall are also computed to understand the effects of* Ω *and* Ri *on them. It is observed that the heat transfer greatly depends on the rotational speed of the cylinder, mixed convective strength and the nanoparticle concentration.*]

27. Computational investigation of transport processes during high energy materials processing application using a hybrid lattice Boltzmann model, **Dipankar Chatterjee** (**American Journal of Heat and Mass Transfer,** vol. 1 No. 2 pp. 52-65, **2014**).

[*A three dimensional numerical simulation is performed in order to understand the transport of mass, momentum and heat during high energy materials processing applications using a hybrid lattice Boltzmann (LB) model. The underlying hydrodynamics is addressed through an evolution equation of a single-particle density distribution function following the conventional LB approach, whereas, the macroscopic temperature and compositional fields are obtained by solving auxiliary scalar transport equations. The solid-liquid phase changing aspects are incorporated into the hydrodynamic LB model by inserting appropriate source terms in the evolution equation through the most formal technique following the extended Boltzmann equations along with an adapted enthalpy updating scheme in association with the* *classical enthalpy-porosity technique. A high power laser surface alloying process is simulated using the developed model and excellent agreement with the available experimental results is observed.*]

28. Impact of transverse shear on vortex induced vibrations of a circular cylinder at low Reynolds numbers, Satya Prakash Singh, **Dipankar Chatterjee** (**Computers and Fluids**, vol. 93, pp. 61-73, **2014**).

[*This paper presents a numerical investigation on the vortex induced vibrations (VIV) of an elastically mounted circular cylinder in linear shear flows at low Reynolds numbers with an aim to shed light on a novel aspect of the VIV phenomena i.e., the impact of transverse shear. In this regard, two-dimensional numerical computations are carried out by deploying a stabilized space-time finite-element formulation. The Reynolds number and the shear parameter are considered in the ranges* $70 \le Re \le 500$ *(for a fixed reduced velocity of* $U^* = 4.92$*) and* $0\% \le \beta \le 40\%$*, respectively. The cylinder of low dimensionless mass (*$m^* = 10$*) is allowed to vibrate along both the transverse and in-line directions. The structural damping coefficient is kept zero to maximize the displacement response. Phenomena of hysteresis are observed around* $Re \sim 84$ *and 325. Modes of vortex shedding are 2S, C(2S) and S + P for various values of Re and* β*. However, only one hysteresis is observed for* β *= 40% at* $Re \sim 84$*. It is further observed that the maximum displacement along the transverse direction does not get affected by the shear introduced at the inlet, however, the maximum in-line displacement depends on the shear parameter. The maximum displacement along the in-line direction increases as the shear parameter increases. For the first hysteresis (Re ~ 84), the extent of Re (for maximum in-line displacement) varies as the shear parameter is changed. The range of Re for the second hysteresis (for all response parameters) depends on the shear parameter such as for* $\beta = 0-10\%$ *the range is 300-325, for* β *= 20% and 30% it is 325-340 and 225-325, respectively. Strouhal number variation with Re is similar to that for other variables. Plots of pressure coefficient distribution for all shear parameters for instantaneous flow field indicate that the difference between the maximum and the minimum values of the pressure coefficient can vary significantly depending on the Reynolds number and the shear parameter.*]

29. MHD mixed convection in a lid-driven cavity including heat conducting solid object and corner heaters with Joule heating, Sudipta Ray, **Dipankar Chatterjee** (**Numerical Heat Transfer A**, vol. 66, pp. 530-550, **2014**).

[*The hydromagnetic mixed convection flow and heat transfer in a top sided lid-driven square enclosure is numerically simulated in this paper following a finite volume approach based on the SIMPLEC algorithm. The enclosure is heated by corner heaters which are under isothermal boundary conditions with different lengths in bottom and right vertical walls. The lid is having lower temperature than heaters. The other boundaries of the enclosure are insulated. A uniform magnetic field is applied along the horizontal direction. A heat conducting horizontal solid object (a square cylinder) is placed centrally within the outer enclosure. Shear forces through lid motion, buoyancy forces due to differential heating and magnetic forces within the electrically conducting fluid inside the enclosure act simultaneously. Heat transfer due to forced flow, thermal buoyancy, Joule dissipation and conduction within the solid object are taken into account. Simulations are conducted for various controlling parameters such as the Richardson number (*$0.1 \le Ri \le 10$*), Hartmann number (*$0 \le Ha \le 50$*) and Joule heating parameter (*$0 \le J \le 5$*) keeping the Reynolds number based on lid velocity fixed as* $Re = 100$*. The flow and thermal fields are analyzed through streamline and isotherm plots for various Ha, J and Ri. Furthermore, the pertinent transport quantities such as the drag coefficient, Nusselt number and bulk fluid temperature are also plotted to show the effects of Ha, J and Ri on them.*]

30. Control of flow separation around bluff obstacles by superimposed thermal buoyancy, **Dipankar Chatterjee**, Bittagopal Mondal (**International Journal of Heat and Mass Transfer**, vol. 72, pp. 128-138, **2014**).

[*Two-dimensional numerical simulation is performed to understand the influence of cross buoyancy on the vortex shedding process behind a stationary heated square cylinder at low Reynolds numbers. The flow is considered in an unbounded medium, however, fictitious confining boundaries are chosen to make the problem computationally feasible. Numerical calculations are carried out by using a finite volume method based on the PISO algorithm in a collocated grid system. The range of Reynolds number is chosen to be 5-40. In this range the flow and thermal fields are found to be steady and separated without the superimposed* *thermal buoyancy (i.e. for pure forced convection). However, as the buoyancy parameter (Richardson number) increases flow becomes unstable and subsequently, at some critical value of Richardson number, periodic vortex shedding is observed to characterize the flow and thermal fields. The global flow and heat transfer quantities such as the recirculation length, overall drag coefficient and surface average Nusselt number are computed for the steady separated flow (without buoyancy effect) and are found in good agreement with the available results in the literature. The effect of superimposed thermal buoyancy for the range of Reynolds number is studied for various Richardson numbers. The frequency of vortex shedding (Strouhal number) is calculated for the unsteady periodic flow. The critical Richardson number for the onset of vortex shedding decreases and the corresponding critical Strouhal number increases with increasing Reynolds number in the chosen range.*]

31. MHD Mixed Convective Transport in a Square Enclosure with Two Rotating Circular Cylinders, **Dipankar Chatterjee**, Pabitra Halder (**Numerical Heat Transfer A**, vol. 65, pp. 802-824, **2014**).

[*This study investigates the two dimensional hydromagnetic mixed convective transport in a cooled square enclosure filled with an electrically conducting fluid in presence of two inner heated circular cylinders with identical shape. The centers of the two equidiameter cylinders are placed at those of the lower and upper half of the enclosure, respectively and the gap between the cylinders is kept fixed. The cylinders may be either kept stationary or made to rotate in their own plane about their centroidal axes. Additionally, cylinders may rotate either in the same direction or in the opposite direction. All the walls of the enclosure are kept isothermal with temperatures less than that of the cylinders. A uniform magnetic field is applied along the horizontal direction normal to the vertical wall. All solid walls are assumed electrically insulated. Simulations are performed by deploying a finite volume technique for various controlling parameters such as the Rayleigh number (*$10^3 \le Ra \le 10^5$*) and Hartmann number (*$0 \le Ha \le 50$*) keeping the Reynolds number based on the cylinder rotation fixed as* $Re = 100$*. The flow and thermal fields are analyzed through streamline and isotherm plots for various Ha, Ra and different combinations of cylinder rotation. Furthermore, the effects of Ha,* Ra *and cylinder rotation on the heat transfer characteristics are portrayed through the local and average Nusselt numbers on all the cylinder surfaces and the bulk average fluid temperature plots.*]

32. Hydromagnetic Mixed Convective Transport in a Vertical Lid-Driven Cavity including a Heat Conducting Rotating Circular Cylinder, **Dipankar Chatterjee**, Bittagopal Mondal, Pabitra Halder (**Numerical Heat Transfer A,** vol. 65, pp. 48-65, **2014**).

[*Two dimensional numerical simulation is performed for the hydromagnetic mixed convective transport in a vertical lid-driven square enclosure filled with an electrically conducting fluid in presence of a heat conducting and rotating solid circular cylinder. Both the top and bottom horizontal walls of the enclosure are considered as thermally insulated and the left and right vertical walls are kept isothermal with different temperatures. The left wall is moving in the upward direction at a uniform speed while all other walls are stationary. A uniform magnetic field is applied along the horizontal direction normal to the moving wall. A heat conducting circular cylinder is placed centrally within the outer enclosure. The cylinder is made to rotate in its own plane about its centroidal axis. Both the clockwise and counterclockwise rotations of the cylinder are considered. All solid walls are assumed electrically insulated. Simulations are performed for various controlling parameters such as the Richardson number (*$1 \le Ri \le 10$*), Hartmann number (*$0 \le Ha \le 50$*) and dimensionless rotational speed (*$-5 \le \Omega \le 5$*) keeping the Reynolds number based on lid velocity fixed as* $Re = 100$*. The flow and thermal fields are analyzed through streamline and isotherm plots for various* Ha, Ω *and* Ri*. Furthermore, the pertinent transport quantities such as the drag coefficient, Nusselt number and bulk fluid temperature are also computed to understand the effects of* Ha, Ω *and* Ri *on them. It is observed that the heat transfer greatly depends on the rotational speed of the cylinder.*]

33. Influence of thermal buoyancy on vortex shedding behind a rotating circular cylinder in cross flow at subcritical Reynolds numbers, **Dipankar Chatterjee**, Chiranjit Sinha (**Journal of Heat Transfer-Trans ASME**, vol. 135, pp. 051704, **2014**).

[*The vortex shedding (VS) behind stationary bluff obstacles in cross-flow can be initiated by imposing thermal instability at subcritical Reynolds numbers (Re). We demonstrate here that additional thermal* *instability is required to be imparted in the form of heating for destabilizing the flow around a rotating bluff obstacle. A two-dimensional numerical simulation is performed in this regard to investigate the influences of cross buoyancy on the VS process behind a heated and rotating circular cylinder at subcritical Re. The flow is considered in an unbounded medium. The range of Re is chosen to be 5-45 with a dimensionless rotational speed (Ω) ranging between 0-4. At this subcritical range of Reynolds number the flow and thermal fields are found to be steady without the superimposed thermal buoyancy (i.e. for pure forced flow). However, as the buoyancy parameter (Richardson number, Ri) increases flow becomes unstable and subsequently, at some critical value of Ri, periodic VS is observed to characterize the flow and thermal fields. The rotation of the cylinder is found to have a stabilizing effect and as Ω increases more heating is observed to be required to destabilize the flow.*]

34. Hydromagnetic Mixed Convective Transport in a Non-isothermally Heated Lid-Driven Square Enclosure including a Heat Conducting Circular Cylinder, **Dipankar Chatterjee**, Satish Kumar Gupta (**Industrial and Engineering Chemistry Research,** vol. 53, pp. 19775–19787, **2014**)

[*A two-dimensional numerical study is performed in an effort to understand the fundamental characteristics of the hydromagnetic mixed convective transport in a non-isothermally heated vertical lid-driven square enclosure filled with an electrically conducting fluid in presence of a heat conducting solid circular cylinder. Additionally, the entropy generation due to heat transfer, fluid friction and magnetic effect is also determined. Simulations are performed for various controlling parameters such as the Richardson number ($1 \le Ri \le 10$), Hartmann number ($0 \le Ha \le 50$), Prandtl number ($0.02 \le Pr \le 7$), Reynolds number based on lid velocity, Re = 100, 150 and 200 and amplitude of sinusoidal function (A = 0.25, 0.5 and 1), keeping the solid-fluid thermal conductivity ratio fixed as, K = 5. The flow and thermal fields are analyzed through streamline and isotherm plots for various Ha, Ri, Re and Pr. Furthermore, the drag coefficient on the moving lid and Nusselt numbers on heated surfaces are also computed to understand the effects of Ha, Ri, Re and Pr and n on them. It is observed that the drag on the moving lid decreases with Re and increases with Ri and Ha, however, remains insensitive with Pr. The heat transfer rate from the hot right wall increases as usual with Re, Pr and Ri, but decreases with Ha. The sinusoidally heated bottom wall shows a decrease in heat transfer rate with increasing Pr and at higher Pr it also decreases with Re. Furthermore, increasing magnetic field strength causes an increase in heat transfer rate from the bottom wall. It also decreases with decreasing value of the amplitude of sinusoidal function.*]

35. Magnetohydrodynamic flow and heat transfer around a circular cylinder in an unconfined medium, Satish Kumar Gupta, **Dipankar Chatterjee**, Bittagopal Mondal (**International Journal of Advancements in Mechanical and Aeronautical Engineering**, vol. 1(1), pp. 102-106, **2014**)

[*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 $10 \le Re \le 50$ with Hartmann number $0 \le Ha \le 10$ 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).*]

36. Lattice kinetic simulation of buoyancy induced MHD flows, **Dipankar Chatterjee** (**International Journal of Heat and Mass Transfer,** vol. 65, pp. 533-544, **2013**).

[*A lattice kinetic model is proposed in this article for simulating buoyancy induced classical magnetohydrodynamic (MHD) flow in the low Mach number incompressible limit. The model is derived by coupling the passive scalar approach of He et al. [J. Comput. Phys. 146 (1998) 282-300] for the flow and thermal fields and Dellar formalism [J. Comput. Phys. 179 (2002) 95-126] for the magnetic field. Accordingly, the underlying hydrodynamics is monitored by a conventional single relaxation time lattice Boltzmann (LB) model through a density distribution function (DF), which obeys a scalar kinetic equation (KE) associated with external force fields (Lorentz and buoyancy forces). The magnetic field is represented by a vector DF, which obeys a corresponding vector KE and the thermal field is obtained from a separate* *temperature DF through another scalar KE incorporating the Joule heating effect. The three distribution functions are coupled through the macroscopic density, momentum, magnetic and thermal fields evaluated at lattice points. This allows a reduced lattice to be used for the magnetic distribution function, with a corresponding saving in the storage. Furthermore, the fluid viscosity, magnetic resistivity and thermal diffusivity may be adjusted independently that renders the model to be applicable for a wide variety of non-isothermal MHD problems. The novelty of the work is the computation of the thermal field in conjunction with the hydro-magnetic fields in the LB framework for the buoyancy driven non-isothermal MHD flows. A 9-bit 2D (d2q9) lattice scheme is used for the numerical computation of the hydrodynamic and thermal fields, whereas the magnetic field is simulated by a reduced 5-bit 2D (d2q5) lattice. Simulation of the magnetoconvective buoyancy induced flow (a) past a vertical flat plate, (b) between two differentially heated vertical walls provide excellent agreement with analytical results. Finally, the model is utilized to solve a classical problem of buoyancy driven MHD flow in a square cavity.*]

37. Magnetoconvective transport in a vertical lid-driven cavity including a heat conducting square cylinder with Joule heating, **Dipankar Chatterjee**, Pabitra Halder, Sinchan Mondal, Supratim Bhattacharjee (**Numerical Heat Transfer A,** vol. 64, pp. 1050-1071, **2013**).

[*The hydromagnetic mixed convection flow and heat transfer in a vertical lid-driven square enclosure is numerically simulated following a finite volume approach based on the SIMPLEC algorithm. Both the top and bottom horizontal walls of the enclosure are insulated and the left and right vertical walls are kept isothermal with different temperatures. The left vertical wall is translating in its own plane at a uniform speed while all other walls are stationary. Two cases of translational lid motion, viz. vertically (a) upward and (b) downward are considered. A uniform magnetic field is applied along the horizontal direction normal to the translating wall. A heat conducting horizontal solid square cylinder is placed centrally within the outer enclosure. Simulations are conducted for various controlling parameters such as the Richardson number (*$1 \le Ri \le 10$*), Hartmann number (*$0 \le Ha \le 50$*) and Joule heating parameter (*$0 \le J \le 5$*) keeping the Reynolds number based on lid velocity fixed as* $Re = 100$*. The flow and thermal fields are analyzed through streamline and isotherm plots for various Ha, J and Ri. Furthermore, the pertinent transport quantities such as the drag coefficient, Nusselt number and bulk fluid temperature are also plotted to show the effects of Ha, J and Ri on them.*]

38. Nonlinear Amplification of Electrokinetic Pumping in Nanochannels in Presence of Hydrophobic Interactions, Suman Chakraborty, **Dipankar Chatterjee**, Chirodeep Bakli (**Physical Review Letters,** vol. 110, pp. 184503, **2013**).

[*We discover a nonlinear coupling between the hydrophobicity of a charged substrate and electrokinetic pumping in narrow fluidic confinements. Our analyses demonstrate that the effective electrokinetic transport in nanochannels may get massively amplified over a regime of bare surface potentials and may subsequently get attenuated beyond a threshold surface charging condition, because of a complex interplay between reduced hydrodynamic resistance on account of the spontaneous inception of a less dense interfacial phase and ionic transport within the electrical double layer. We also show that the essential physics delineated by our mesoscopic model, when expressed in terms of a simple mathematical formula, agrees remarkably with that portrayed by molecular dynamics simulations. The non-trivial characteristics of initial increment followed by a decrement of effective zeta potential with bare surface potential may open up the realm of hitherto-unexplored operating regimes of electro-hydrodynamically actuated nanofluidic devices.*]

39. Unconfined flow and heat transfer around a square cylinder at low Reynolds and Hartmann numbers, **Dipankar Chatterjee**, Kanchan Chatterjee (**International Journal of Fluid Mechanics Research**, vol. 40, pp. 71-90, **2013**).

[*The forced convection heat transfer is analyzed through a two-dimensional numerical simulation following a finite volume approach for the hydromagnetic flow around a square 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* $10 \le Re \le 50$ *with Hartmann number* $0 \le Ha \le 10$ *and with a fixed Prandtl* *number, Pr = 0.02 (liquid metal) and a blockage parameter, $\beta = d/H = 5\%$. The flow is steady and stable for the above range of conditions. The magnetic field provides additional stability to the flow as a result of which the wake region behind the cylinder reduces with increasing magnetic field strength at any Reynolds number. The critical magnetic field strength is also computed for which the separation is completely suppressed for the Reynolds number range in case of transversely applied magnetic field. The rate of heat transfer is found almost invariant at low Reynolds number whereas it increases slightly for higher Reynolds number with the applied magnetic field. The heat transfer increases as usual with the Reynolds number for all Hartmann numbers.*]

40. Unsteady mixed convection heat transfer from tandem square cylinders in cross flow at low Reynolds numbers, **Dipankar Chatterjee**, Bittagopal Mondal (**Heat and Mass Transfer,** vol. 49, pp. 907-920, **2013**).

[*A two-dimensional numerical study is carried out to understand the influence of cross buoyancy on the vortex shedding processes behind two equal isothermal square cylinders placed in a tandem arrangement at low Reynolds numbers. The spacing between the cylinders is fixed with five widths of the cylinder dimension. The flow is considered in an unbounded medium, however, fictitious confining boundaries are chosen to make the problem computationally feasible. Numerical calculations are performed by using a finite volume method based on the PISO algorithm in a collocated grid system. The range of Reynolds number is chosen to be 50-150. The flow is unsteady laminar and two-dimensional in this Reynolds number range. The mixed convection effect is studied for Richardson number range of 0-2 and the Prandtl number is chosen constant as 0.71. The effect of superimposed thermal buoyancy on flow and isotherm patterns are presented and discussed. The global flow and heat transfer quantities such as overall drag and lift coefficients, local and surface average Nusselt numbers and Strouhal number are calculated and discussed for various Reynolds and Richardson numbers.*]

41. MHD Mixed Convection in a Lid-Driven Cavity including a Heated Source, **Dipankar Chatterjee** (**Numerical Heat Transfer A,** vol. 64, pp. 235-254, **2013**).

[*Numerical simulations are performed to understand the thermo-magneto-convective transport of fluid and heat in a vertical lid-driven square enclosure following a finite volume approach based on the SIMPLEC algorithm. The enclosure is filled with an electrically conducting fluid and having a heated source on the right vertical wall. Two different types of sources such as a semicircular and a rectangular one are considered. Both the top and bottom horizontal walls and the right vertical wall except the source of the enclosure are assumed insulated and the left vertical wall and the sources are kept isothermal with different temperatures. The left vertical wall is also translating in its own plane at a uniform speed while all other walls are stationary. Two cases of translational lid motion, viz. vertically (a) upward and (b) downward are considered. A uniform magnetic field is applied along the horizontal direction normal to the translating wall. Shear forces due to lid motion, buoyancy forces as a result of differential heating and magnetic forces within the electrically conducting fluid act simultaneously. Heat transfer due to forced flow, natural convection and Joule dissipation are taken into account. Simulations are conducted for various controlling parameters such as the Rayleigh number ($10^3 \le Ra \le 10^5$), Hartmann number ($0 \le Ha \le 100$) and Joule heating parameter ($0 \le J \le 5$) keeping the Reynolds number based on lid velocity fixed as $Re = 100$. The flow and thermal fields are analyzed through streamline and isotherm plots for various Ha and J. Furthermore, the pertinent transport quantities such as the drag coefficient, Nusselt number and bulk fluid temperature are also plotted to show the effects of Ha, J and Ra on them.*]

42. Mixed convection heat transfer past in-line square cylinders in a vertical duct, **Dipankar Chatterjee**, Md. Raja (**Thermal Science,** vol. 17, pp. 567-580, **2013**).

[*The mixed convection heat transfer around five in-line isothermal square cylinders periodically arranged within a vertical duct is numerically investigated in this paper. Spacing between two cylinders (S) is fixed at one width of the cylinder dimension (d) and the flow confinement of various degrees are studied for the blockage ratios of B = 0%, 10%, 25% and 50%. The buoyancy aided/opposed convection is examined for the Richardson number (Ri) ranges from -1 to +1 with a fixed Prandtl number Pr = 0.7 and Reynolds number Re = 100. The transient numerical simulation for this two-dimensional, incompressible, laminar flow and heat transfer problem is carried out by a finite volume based commercial CFD package FLUENT ™. The representative streamlines and isotherm patterns are presented to interpret the flow and thermal transport visualization. Additionally, the time and surface average skin friction coefficient (C_f), drag (C_D)*

and lift (C_L) coefficients as well as the time and surface average Nusselt number (Nu) for representative cylinders are determined to elucidate the effects of Re and Ri on the flow and heat transfer phenomena.]

43. Unsteady forced convection heat transfer over semicircular cylinder at low Reynolds numbers, **Dipankar Chatterjee**, Bittagopal Mondal, Pabitra Halder (**Numerical Heat Transfer A,** vol. 63, pp. 411-429, **2013**).

    [*An unsteady two-dimensional numerical simulation is performed to investigate the laminar forced convection heat transfer for flow past a semicircular cylinder in an unconfined medium. The Reynolds number considered in this study ranges from 50 to 150 with a fixed Prandtl number (Pr = 0.71). Two different configurations of the semicircular cylinder are considered; one when the curved surface facing the flow and the other when the flat surface facing the flow. Fictitious confining boundaries are chosen on the lateral sides of the computational domain that makes the blockage ratio* B *= 5% in order to make the problem computationally feasible. A finite volume based technique is used for the numerical computation. The flow and heat transfer characteristics are analyzed with the streamline and isotherm patterns at various Reynolds numbers. The dimensionless frequency of vortex shedding (Strouhal number), drag coefficient and Nusselt numbers are presented and discussed. Substantial differences in the global flow and heat transfer quantities are observed for the two different configurations of the obstacle chosen in the study. It is observed that the heat transfer rate is enhanced substantially when the curved surface facing the flow in comparison to the case when the flat surface facing the flow.*]

44. Wall bounded flow and heat transfer around a circular cylinder at low Reynolds and Hartmann numbers, **Dipankar Chatterjee**, Kanchan Chatterjee (**Heat Transfer - Asian Research**, vol. 42, pp. 133-150, **2013**).

    [*A two-dimensional numerical simulation is performed following a finite volume approach to analyze the forced convection heat transfer for the hydromagnetic flow around a circular cylinder at low Reynolds numbers. The cylinder is placed within a rectangular channel subjected to externally applied magnetic fields 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. The simulation is carried out for the range of Reynolds number* $10 \le Re \le 80$ *with Hartmann number* $0 \le Ha \le 10$ *and for different Prandtl numbers, Pr = 0.02 (liquid metal), 0.71 (air) and 7 (water) for a blockage ratio* $\beta = 0.25$*. The flow is steady for the above range of conditions. Apart from the channel wall, the magnetic field provides additional stability to the flow as a result of which the recirculation region behind the obstacle reduces with increasing magnetic field strength for a particular Reynolds number. The rate of heat transfer is found almost invariant at low Re whereas it increases slightly for higher Re with the applied magnetic field. The heat transfer increases as usual with the Reynolds number for all Hartmann numbers.*]

45. Mixed convection heat transfer from tandem square cylinders for various spacing ratios, **Dipankar Chatterjee**, Bittagopal Mondal (**Numerical Heat Transfer A,** vol. 63, pp. 101-119, **2013**).

    [*This paper presents a two-dimensional numerical study on the fluid flow and mixed convection heat transfer around two equal isothermal square cylinders placed in a tandem arrangement and subjected to the cross flow of a Newtonian fluid at moderate Reynolds numbers. The spacing between the cylinders is varied by changing the gap to cylinder size ratio as* $S/d = 1, 2, 3, 4, 5, 7$ and 10*. The Reynolds number is considered in the range* $50 \le Re \le 150$*. The mixed convection effect is studied for Richardson number range of 0-2 and the Prandtl number is chosen constant as 0.71. The flow is considered in an unbounded medium; however, fictitious confining boundaries are chosen to make the problem computationally feasible. Numerical calculations are performed by using a PISO algorithm based finite volume solver in a collocated grid system. The effect of superimposed thermal buoyancy on flow and isotherm patterns are presented and discussed. The global flow and heat transfer quantities such as overall drag and lift coefficients, local and surface average Nusselt numbers and Strouhal number are calculated and discussed for various Reynolds, Richardson numbers and spacing ratios. The notable contribution is the quantification of the critical spacing ratio which is observed to decrease with increasing thermal buoyancy effect for a specific Reynolds number.*]

46. Control of flow separation around bluff obstacles by transverse magnetic field, **Dipankar Chatterjee**, Kanchan Chatterjee, Bittagopal Mondal (**Journal of Fluids Engineering-Transactions of ASME**, vol. 134, pp. 091102-1, **2012**).

[*Electromagnetic field may be used to control the flow separation during flow of electrically conducting fluids around bluff obstacles. The steady separated flow around bluff bodies at low Reynolds numbers almost behaves like creeping flow at certain field strength. This phenomena, although, already known is exactly quantified through numerical simulation and the critical field strength of an externally applied magnetic field is obtained for which the flow separation is completely suppressed. The flow of a viscous, incompressible and electrically conducting fluid (preferably liquid metal or electrolyte solution) at Reynolds number range 10-40 and at low magnetic Reynolds number is considered in an unbounded medium subjected to uniform magnetic field strength along the transverse direction. Circular and square cross-sections of the bluff obstacles are considered for simulation purpose. Fictitious confining boundaries are chosen on the lateral sides of the computational domain that makes the blockage ratio (ratio of cylinder size to width of the domain) 5%. The two-dimensional numerical simulation is performed following a finite volume approach based on SIMPLE algorithm. The major contribution is the determination of the critical Hartmann number for the complete suppression of the flow separation around circular and square cylinders for the steady flow in the low Reynolds number laminar regime. The recirculation length and separation angle are computed to substantiate the findings. Additionally, the drag and skin friction coefficients are computed to show the aerodynamic response of the obstacles under imposed magnetic field condition.*]

47. Forced Convection heat transfer from an equilateral triangular cylinder at low Reynolds numbers, **Dipankar Chatterjee**, Bittagopal Mondal (**Heat and Mass Transfer**, vol. 48, pp. 1575-1587, 2012).

[*An unsteady two-dimensional numerical simulation is performed to investigate the forced convection heat transfer for flow past a long heated equilateral triangular cylinder in an unconfined medium for the low Reynolds number laminar regime. The Reynolds number considered in this study ranges from 50 to 250 with a fixed Prandtl number (Pr = 0.71). Fictitious confining boundaries are chosen on the lateral sides of the computational domain that makes the blockage ratio* β *= 5% in order to make the problem computationally feasible. An unstructured triangular mesh is used for the computational domain discretization and the simulation is carried out with the commercial CFD solver Fluent. The flow and heat transfer characteristics are analyzed with the streamline and isotherm patterns at various Reynolds numbers. The dimensionless frequency of vortex shedding (Strouhal number), drag coefficient and Nusselt numbers are presented and discussed. The results obtained are in good agreement with the available results in the literature.*]

48. On the vortex shedding mechanism behind a circular cylinder subjected to cross buoyancy at low Reynolds numbers, Dipankar Chatterjee, Bitta Gopal Mondal (**Computational Thermal Sciences**, vol. 4(1), pp. 23-38, **2012**).

[*A two-dimensional numerical investigation is carried out to understand the influence of thermal buoyancy on the vortex shedding processes behind a fixed heated circular cylinder in cross-flow at low Reynolds numbers. The flow is considered in an infinite medium, however, fictitious confining boundaries are chosen to make the problem computationally feasible. Numerical simulations are performed by using a finite volume method based on the PISO algorithm in a collocated grid system.. The range of Reynolds number for the present study is chosen to be 5-40, such that the flow remains laminar and two-dimensional. In this range the flow and thermal fields are found to be steady without the superimposed thermal buoyancy (i.e. for pure forced convection). However, as the buoyancy parameter (Richardson number, Ri) increases flow becomes unstable and consequently, at some critical value of Ri, periodic vortex shedding is observed to characterize the flow and thermal fields. The global flow and heat transfer quantities such as the recirculation length, overall drag coefficient and surface average Nusselt number are computed for the steady separated flow (without buoyancy effect) and are found in good agreement with the available results in the literature. The effect of superimposed thermal buoyancy for the range of Reynolds number is studied for various Richardson numbers. The dimensionless frequency of vortex shedding (Strouhal number) is computed for the unsteady periodic flow.*]

49. Mixed convection heat transfer from an in-line row of square cylinders in cross-flow at low Reynolds number, **Dipankar Chatterjee**, Gautam Biswas, Sakir Amiroudine (**Numerical Heat Transfer A**, vol. 61, 891-911, **2012**).

[*This paper is to investigate the two-dimensional unsteady laminar mixed convection heat transfer from a row of five in-line and fixed isothermal square cylinders placed in an unconfined medium and subjected to cross-flow of a Newtonian fluid at low Reynolds number (Re = 125). The flow and thermal variables are obtained for the separation ratios (spacing to cylinder size ratio, s/d) of 0.5, 1, 1.5, 2, 3 and 4. The mixed convection heat transfer is studied for Richardson numbers (Ri) ranging from 0 to 3 with a fixed Prandtl number (Pr = 0.71). Numerical simulations are performed with a finite volume code based on the PISO algorithm in a co-located grid system. The instantaneous vorticity fields along with the isotherm patterns are systematically presented and discussed for different separation ratios and Richardson numbers. Depending on the engineering application, the temperature difference between the surface and the free stream could vary to make buoyancy of primary importance entailing major modification of the flow field. Additionally, the instantaneous and mean drag and lift coefficients, Strouhal numbers and mean Nusselt numbers are determined and discussed*.]

50. Triggering vortex shedding by superimposed thermal buoyancy around bluff obstacles in cross-flow at low Reynolds numbers, Dipankar Chatterjee (**Numerical Heat Transfer A**, vol. 61, pp. 800-806, **2012**).

[*Influence of superimposed thermal buoyancy on the initiation of vortex shedding process behind bluff obstacles (such as circular and square cylinders in 2D) in cross-flow at low Reynolds numbers (*$10 \le Re \le 40$*) is discussed. The flow which is steady and separated at this Reynolds number range eventually becomes unsteady periodic with the introduction of thermal buoyancy. The aim here is to numerically predict the critical value of the buoyancy parameter (Richardson number, Ri) for the onset of vortex shedding. The critical Ri is found to have a decreasing tendency for both types of cylinder geometries with increasing Re*.]

51. Forced convection heat transfer from tandem square cylinders for various spacing ratios, Dipankar Chatterjee, Bittagopal Mondal (**Numerical Heat Transfer A**, vol. 61, 381-400, **2012**).

[*This paper presents a two-dimensional numerical study on the fluid flow and forced convection heat transfer around two equal isothermal square cylinders placed in a tandem arrangement and subjected to the cross flow of a Newtonian fluid at low Reynolds numbers. The spacing between the cylinders is varied by changing the gap to cylinder size ratio as* $S/d = 1, 2, 3, 4, 5, 7$ and 10. *The flow is considered in an unbounded medium, however, fictitious confining boundaries are chosen to make the problem computationally feasible. Numerical calculations are performed by using a PISO algorithm based finite volume solver in a collocated grid system. The Reynolds number is considered in the range* $50 \le Re \le 150$ *and the Prandtl number is chosen constant as 0.71. The instantaneous vorticity and isotherm patterns are presented and discussed at various Reynolds numbers and spacing ratios for the flow and thermal transport visualization. Additionally, the global flow and heat transfer quantities such as the overall drag and lift coefficients, local and surface average Nusselt numbers and Strouhal number are calculated and discussed for various Reynolds numbers and spacing ratios*.]

52. Lattice Boltzmann simulation of heat conduction problems in non-isothermally heated enclosures, Bittagopal Mondal, **Dipankar Chatterjee** (**Heat Transfer-Asian Research**, vol. 41(2), pp. 127-144, **2012**).

[*A numerical study following the lattice Boltzmann method (LBM) is performed to solve transient heat conduction problems with and without volumetric heat generation/absorption in 2-D and 3-D Cartesian geometries. Uniform lattices are considered for both geometries. To validate the correctness of LBM, a finite difference method (FDM) is also used to solve the 2-D problem without heat generation/absorption and results are compared with that of LBM. For both 2-D and 3-D geometries one of the walls is heated and cooled with a sinusoidal function and rest of the walls are cooled isothermally. Effects of amplitude of the sinusoidal function and volumetric heat generation/absorption on temperature profiles are analyzed*.]

53. Effect of thermal buoyancy on the upward flow and heat transfer around a square cylinder, **Dipankar Chatterjee**, Bitta Gopal Mondal (**Heat Transfer Engineering,** vol. 33(12), pp. , **2012**).

[*The effect of aiding/opposing buoyancy on the two-dimensional upward flow and heat transfer around a heated/cooled cylinder of square cross section is studied in this work. The commercial CFD software FLUENT ^{TM} is used for the numerical solution. The influence of buoyancy aided/opposed convection is studied for the range of parameters* $-1 \le Ri \le 1$, $50 \le Re \le 150$ *and the blockage parameters of* $B = 0.02$ *and 0.25. The flow shows unsteady periodic nature in the chosen range of Reynolds numbers for the forced convective cases (* $Ri = 0$ *). However, the vortex shedding is observed to stop completely at some critical values of Richardson numbers, below which the shedding of vortices into the stream is quite prominent. Representative vortex structures and isotherm patterns for different blockage parameters are systematically presented and discussed. The critical Richardson and Nusselt numbers are plotted against the Reynolds and Richardson numbers to elucidate the role of thermal buoyancy on flow and heat transfer characteristics. It is observed that the vortex shedding frequencies (Strouhal numbers) increase with increased heating and suddenly reduce to zero at the critical Ri. The critical Ri is again found to increase with Re for a particular blockage ratio and higher the blockage ratio, less is the critical Ri. The results obtained from the present study are finally compared with the available numerical and experimental results and an excellent agreement is observed.*]

54. A lattice Boltzmann model for high energy materials processing application, Dipankar Chatterjee (**International Journal for Multiscale Computational Engineering**, vol. 10(3), pp. 229-247, **2012**).

[*A three-dimensional (3D) lattice Boltzmann (LB) scheme is presented, in this article, to address the incompressible transport phenomena in presence of a continuously evolving phase-change interface typically encountered in high energy materials processing applications. The proposed LB scheme utilizes three separate distribution functions to monitor the underlying hydrodynamic, thermal and compositional fields. Accordingly, the kinematic viscosity, thermal and mass diffusivities can be adjusted independently which makes the model suitable for a wide range of phase change problems in high power materials processing applications. The phase changing aspects are incorporated into the LB model by inserting appropriate source terms in the respective kinetic equations through the most formal technique following the extended Boltzmann equations along with an adapted enthalpy updating scheme in association with the classical enthalpy-porosity technique for solid-liquid phase transition problems. The model is used to simulate a conventional high power laser surface alloying process and excellent agreement with the available experimental results is observed.*]

55. Influences of thermal buoyancy on the vortex shedding mechanisms behind a heated circular cylinder at low Reynolds numbers, **Dipankar Chatterjee**, Bittagopal Mondal, Kanchan Chatterjee (**The Bulletin of Engineering and Science**, vol. 4, No. 1, pp. 40-48, **2012**).

[*The article deals with the effects of thermal buoyancy on the vortex shedding mechanisms behind a heated circular cylinder in an infinite medium for cross-flow at low Reynolds numbers. A two-dimensional numerical simulation is performed in this regard using the commercial CFD software FLUENT. The range of Reynolds number is chosen to be 10-45 with Prandtl number* $Pr = 0.7$ *. In this range the flow field is found to be steady and separated without the superimposed thermal buoyancy. However, as the intensity of buoyancy increases (i.e. Richardson number,* $Ri > 0$ *), the flow becomes unstable and eventually, the periodic vortex shedding is observed to characterize the flow and thermal fields at some critical value of* Ri *. The effect of superimposed thermal buoyancy is studied for* $Ri = 0 - 2$ *. The critical Ri for the onset of vortex shedding is found to decrease (which is in clear contrast with other finding) and the dimensionless frequency of vortex shedding (Strouhal number, St) is found to increase with Re in the chosen range. The major contribution is the quantification and assessment of the characteristic behavior of the critical buoyancy parameter (Ri) for transition from steady to unsteady periodic flow.*]

56. Effect of thermal buoyancy on vortex shedding behind a square cylinder in cross flow at low Reynolds number, Dipankar Chatterjee, Bitta Gopal Mondal (**International Journal of Heat and Mass Transfer**, vol. 54, pp. 5262-5274, **2011**).

[*Two-dimensional numerical simulation is performed to understand the influence of cross buoyancy on the vortex shedding processes behind a fixed heated square cylinder at low Reynolds numbers. The flow is considered in an unbounded medium, however, fictitious confining boundaries are chosen to make the problem computationally feasible. Numerical calculations are carried out by using a finite volume method based on the PISO algorithm in a collocated grid system.. The range of Reynolds number for the present study is chosen to be 5-40, such that the flow remains laminar and two-dimensional. In this range the flow and thermal fields are found to be steady without the superimposed thermal buoyancy (i.e. for pure forced convection). However, as the buoyancy parameter (Richardson number, Ri) increases flow becomes unstable and consequently, at some critical value of Ri, periodic vortex shedding is observed to characterize the flow and thermal fields. The global flow and heat transfer quantities such as the recirculation length, overall drag coefficient and surface average Nusselt number are computed for the steady separated flow (without buoyancy effect) and are found in good agreement with the available results in the literature. The effect of superimposed thermal buoyancy for the range of Reynolds number is studied for various Richardson numbers. The dimensionless frequency of vortex shedding (Strouhal number) is computed for the unsteady periodic flow.*]

57. Effects of Reynolds and Prandtl numbers on flow and heat transfer across tandem square cylinders in the steady flow regime, **Dipankar Chatterjee**, Gautam Biswas (**Numerical Heat Transfer A,** vol. 59, pp. 421-437, **2011**).

[*The effects of Reynolds and Prandtl numbers on the fluid flow and heat transfer characteristics over two equal isothermal square cylinders placed in a tandem arrangement in cross flow have been investigated in this paper. The spacing between the cylinders is fixed with four widths of the cylinder. The numerical results are presented for the range of conditions as:* $1 \le \mathrm{Re} \le 30$ *and* $0.7 \le \mathrm{Pr} \le 1000$ *for three different blockage parameters* $\mathrm{B} = 0.05$*, 0.25 and 0.5. Numerical simulations are performed with a finite volume code based on the PISO algorithm in a collocated grid system. The representative streamlines and isotherm patterns are presented and discussed. In addition, the overall drag coefficient and average Nusselt number are determined to elucidate the role of Reynolds and Prandtl numbers on flow and heat transfer. It is found that the flow is completely steady for the chosen ranges of the parameters.*]

58. Computational modeling of transport phenomena in high energy materials processing application: Large Eddy Simulation and Parallelization, **Dipankar Chatterjee** (**International Journal of Computational Materials Science and Surface Engineering**, vol. 4 (1), pp. 1-22, **2011**).

[*A comprehensive three-dimensional numerical model is presented in order to address the coupled turbulent momentum, heat and species transport during molten metal-pool convection in association with continuous evolution of solid-liquid interface typically encountered in high energy materials processing applications. The turbulent aspect is handled by a large eddy simulation (LES) model and the phase changing phenomena is taken care of by a modified enthalpy-porosity technique. The proposed finite volume based LES model is subsequently parallelized for effective computational economy. To demonstrate the effectiveness of the present model, a systematic analysis is subsequently carried out to simulate a typical high power laser surface alloying process, where the effects of turbulent transport can actually be realized.*]

59. Lattice Boltzmann simulation of thermofluidic transport phenomena in a DC magnetohydrodynamic (MHD) micropump, **Dipankar Chatterjee**, Sakir Amiroudine (**Biomedical Microdevices,** vol. 13, pp. 147-157, **2011**).

[*A comprehensive non-isothermal Lattice Boltzmann (LB) algorithm is proposed in this article to simulate the thermofluidic transport phenomena encountered in a direct-current (DC) magnetohydrodynamic (MHD) micropump. Inside the pump, an electrically conducting fluid is transported through the microchannel by the action of an electromagnetic Lorentz force evolved out as a consequence of the interaction between applied electric and magnetic fields. The fluid flow and thermal characteristics of the MHD micropump depends on several factors such as the channel geometry, electromagnetic field strength and electrical* *property of the conducting fluid. An involved analysis is carried out following the LB technique to understand the significant influences of the aforementioned controlling parameters on the overall transport phenomena. In the LB framework, the hydrodynamics is simulated by a distribution function, which obeys a single scalar kinetic equation associated with an externally imposed electromagnetic force field. The thermal history is subsequently monitored by a separate temperature distribution function through another scalar kinetic equation incorporating the Joule heating effect. Agreement with analytical, experimental and other available numerical results is found to be quantitative.*]

60. Numerical simulation of mixed convection heat transfer past in-line square cylinders, Md. Raja, **D. Chatterjee** (**International Journal of Advances in Thermal Sciences and Engineering**, vol. 2, pp. 35-39, **2011**).

[*The mixed convection heat transfer around five in-line isothermal square cylinders periodically and vertically arranged in an unconfined medium is numerically investigated in this paper. Spacing between two cylinders (S) is fixed at one width of the cylinder dimension (d). The buoyancy aided/opposed convection is examined for the Richardson number (Ri) ranges from -1 to +1 with a fixed Prandtl number Pr = 0.7 and Reynolds number Re = 100. The transient numerical simulation for this two-dimensional, incompressible, laminar flow and heat transfer problem is carried out by a finite volume based commercial CFD package FLUENT. The representative streamlines and isotherm patterns are presented to interpret the flow and thermal transport visualization. Additionally, the time and surface average Nusselt number (Nu) is determined to elucidate the effects of Re and Ri on the flow and heat transfer phenomena*.]

61. Mixed convection heat transfer from three heated square cylinders in cross-flow at low Reynolds numbers, **Dipankar Chatterjee**, Shyama Prasad Das (**Heat and Mass Transfer**, vol. 46 (11), pp. 1239-1251, **2010**).

[*The paper presents the effects of cross buoyancy and Prandtl number on the flow and heat transfer characteristics around three equal isothermal square cylinders arranged in a staggered configuration within an unconfined medium. Transient two-dimensional numerical simulations are performed with a finite volume code based on the SIMPLEC algorithm in a collocated grid system. The pertinent dimensionless parameters, such as Reynolds, Prandtl and Richardson numbers are considered in the range of* $1 \le \mathrm{Re} \le 30$, $0.7 \le \mathrm{Pr} \le 100$ *and* $0 \le \mathrm{Ri} \le 1$. *The representative streamlines, vortex structures and isotherm patterns are presented and discussed. In addition, the overall drag and lift coefficients and average Nusselt numbers are determined to elucidate the effects of Reynolds, Prandtl and Richardson numbers on flow and heat transfer. The flow is observed to be steady for all the ranges of parameters considered. The drag coefficient is found to decrease with Re (for Ri = 0) and Ri at low Pr, whereas it increases with Pr at higher Ri. The lift coefficient decreases with Ri at low Pr and increases with Pr at higher Ri. The time and surface average cylinder Nusselt number is found to increase monotonically with Re as well as Pr while it remains almost insensitive to Ri at low Pr.*]

62. Mixed convection heat transfer from tandem square cylinders in a vertical channel at low Reynolds numbers, **Dipankar Chatterjee** (**Numerical Heat Transfer A**, vol. 58: 9, pp. 740-755, **2010**).

[*The fluid flow and heat transfer characteristics around two isothermal square cylinders arranged in a tandem configuration with respect to the incoming flow within an insulated vertical channel at low Reynolds number range (*$1 \le \mathrm{Re} \le 30$*) are estimated in this paper. Spacing between the cylinders (S) is fixed at four widths of the cylinder dimension (d) and the blockage parameter (B) is set to 0.25. The buoyancy aided/opposed convection is examined for the Richardson number (Ri) ranges from -1 to 1 with a fixed Prandtl number (Pr) of 0.7. The transient numerical simulation for this two-dimensional, incompressible, laminar flow and heat transfer problem is carried out by a finite volume code based on the PISO algorithm in a collocated grid system. The results suggest that the flow remains steady for the entire ranges of parameters chosen in this study. The representative streamlines, vorticity and isotherm patterns are presented to interpret the flow and thermal transport visualization. Additionally, the time average drag coefficient (*C_D*) as well as time and surface average Nusselt number (Nu) for the upstream and downstream cylinders are determined to elucidate the effects of Re and Ri on flow and heat transfer phenomena*.]

63. Lattice Boltzmann simulation of incompressible transport phenomena in macroscopic solidification processes, **Dipankar Chatterjee** (**Numerical Heat Transfer B**, vol. 58, pp. 55-72, **2010**).

[*A lattice Boltzmann (LB) simulation strategy is proposed, in this article, for the incompressible transport phenomena occurring in case of macroscopic solidification of pure substances. The proposed model is derived by coupling a passive scalar based thermal LB model with the classical enthalpy-porosity technique for solid-liquid phase transition problems. The underlying hydrodynamics is monitored by a conventional single particle density distribution function (DF) through a kinetic equation, whereas the thermal field is obtained from another kinetic equation which is governed by a separate temperature DF. The phase changing aspects are incorporated into the LB model by inserting appropriate source terms in the respective kinetic equations through the most formal technique following the extended Boltzmann equations along with an appropriate enthalpy updating scheme. The proposed model is validated extensively with one and two-dimensional solidification problems for which analytical and numerical results are available in the literature and finally it is used for solving a benchmark problem, viz. the Bridgman crystal growth in a square crucible.*]

64. Lattice kinetic simulation of non-isothermal magnetohydrodynamics, **Dipankar Chatterjee**, Sakir Amiroudine (**Physical Review E**, vol. 81, pp. 066703-1-6, **2010**).

[*In this paper, a lattice kinetic algorithm is presented to simulate non-isothermal magnetohydrodynamics in the low Mach number incompressible limit. The flow and thermal fields are described by two separate distribution functions through respective scalar kinetic equations and the magnetic field is governed by a vector distribution function through a vector kinetic equation. The distribution functions are only coupled via the macroscopic density, momentum, magnetic field and temperature computed at the lattice points. The novelty of the work is the computation of the thermal field in conjunction with the hydromagnetic fields in the lattice Boltzmann framework. A 9-bit 2D lattice scheme is used for the numerical computation of the hydrodynamic and thermal fields, whereas the magnetic field is simulated in a 5-bit 2D lattice. Simulation of Hartmann flow in a channel provides excellent agreement with corresponding analytical results.*]

65. Effect of aiding/opposing buoyancy on two-dimensional laminar flow and heat transfer across a circular cylinder, Gurunath Gandikota V.S., Sakir Amiroudine, **Dipankar Chatterjee**, Gautam Biswas (**Numerical Heat Transfer A**, vol. 58: 5, pp. 385-402, **2010**).

[*The effect of thermal buoyancy on the upward flow and heat transfer characteristics around a heated/ cooled circular cylinder is studied in this work. A two-dimensional numerical model is deployed for the analysis. The numerical model uses a collocated finite volume code based on the SIMPLEC algorithm. The influence of buoyancy aided/opposed convection is studied for the range of parameters* $-0.5 \le Ri \le 0.5$, $50 \le Re \le 150$ *and the blockage parameters of* $B = 0.02$ *and 0.25. The flow shows unsteady periodic nature in the chosen range of Reynolds numbers for the forced convective cases (* $Ri = 0$ *). However, the vortex shedding is observed to stop completely at some critical values of Richardson numbers, below which the shedding of vortices into the stream is quite prominent. Representative vortex structures and isotherm patterns for different blockage parameters are systematically presented and discussed. The critical Richardson and Nusselt numbers are plotted against the Reynolds and Richardson numbers to elucidate the role of thermal buoyancy on flow and heat transfer characteristics. It is observed that the vortex shedding frequencies (Strouhal numbers) increase with increased heating and suddenly reduce to zero at the critical Ri. The critical Ri is again found to increase with Re for a particular blockage ratio and higher the blockage ratio, less is the critical Ri. The results obtained from the present study are finally compared with the available numerical and experimental results and an excellent agreement is observed.*]

66. Two-dimensional mixed convection heat transfer from confined tandem square cylinders in cross-flow at low Reynolds numbers, **Dipankar Chatterjee**, Sakir Amiroudine (**International Communications in Heat and Mass Transfer**, vol. 37, pp. 7-16, **2010**).

[*A two-dimensional numerical simulation is carried out to understand the effects of thermal buoyancy and Prandtl number on flow characteristics and mixed convection heat transfer over two equal isothermal square cylinders placed in a tandem arrangement within a channel at low Reynolds numbers. The spacing between the cylinders is fixed with four widths of the cylinder. The numerical results are presented for the* *range of conditions as:* $1 \le \mathrm{Re} \le 30$, $0.7 \le \mathrm{Pr} \le 100$ *(the maximum value of Peclet number being 3000) and* $0 \le \mathrm{Ri} \le 1$ *for a fixed blockage parameter* $\mathrm{B} = 10\%$. *The unsteady numerical simulations are performed with a finite volume code based on the PISO algorithm in a collocated grid system. The representative streamlines, vortex structures and isotherm patterns are presented and discussed. In addition, the overall drag and lift coefficients, recirculation length and average Nusselt numbers are determined to elucidate the role of Reynolds, Prandtl and Richardson numbers on flow and heat transfer. It is found that the flow is completely steady for the chosen ranges of the parameters.*]

67. Numerical simulation of flow past row of square cylinders for various separation ratios, **Dipankar Chatterjee**, Gautam Biswas, Sakir Amiroudine (**Computers and Fluids**, vol. 39, pp. 49-59, **2010**).

[*A systematic study for the flow around a row of five square cylinders placed in a side-by-side arrangement and normal to the oncoming flow at a Reynolds number of 150 is carried out through the numerical solution of the two-dimensional unsteady incompressible Navier-Stokes equations. Special attention is paid to investigate the effect of the spacing between the five cylinders on the wake structure and vortex shedding mechanism. The simulations are performed for the separation ratios (spacing to size ratio) of 1.2, 2, 3 and 4. Depending on the separation ratio the following flow patterns are observed: a flip-flopping pattern, in-phase and antiphase synchronized pattern and non-synchronized pattern. These flow patterns are supposed to be a consequence of the interaction between two types of frequencies viz. the vortex shedding (primary) and the cylinder interaction (secondary) frequencies. At small separation ratio the flow is predominantly characterized by the jet in the gaps between successive cylinders and the secondary frequencies play a role in the resulting chaotic flow. On the contrary, at higher separation ratio the secondary frequencies almost disappear and the resulting flow becomes more synchronized dominated by the primary frequency.*]

68. Numerical investigation of forced convection heat transfer in unsteady flow past a row of square cylinders, **Dipankar Chatterjee**, Gautam Biswas, Sakir Amiroudine (**International Journal of Heat and Fluid Flow**, vol. 30, pp. 1114-1128, **2009**).

[*This paper presents the unsteady laminar forced convection heat transfer from a row of five isothermal square cylinders placed in a side-by-side arrangement at a Reynolds number of 150. The numerical simulations are performed using a finite volume code based on the PISO algorithm in a collocated grid system. Special attention is paid to investigate the effect of the spacing between the cylinders on the overall transport processes for the separation ratios (spacing to size ratio) between 0.2 and 10. No significant interaction between the wakes is observed for spacing greater than four times the diameter at this Reynolds number. However, at smaller spacing, the wakes interact in a complicated manner resulting different thermo-hydrodynamic regimes. The vortex structures and isotherm patterns obtained are systematically presented and discussed for different separation ratios. In addition, the mean and instantaneous drag and lift coefficients, mean and local Nusselt number and Strouhal number are determined and discussed for various separation ratios. A new correlation is derived for mean Nusselt number as a function of separation ratio for such flows.*]

69. An enthalpy-based thermal lattice Boltzmann model for non-isothermal systems, **Dipankar Chatterjee** (**Euro Physics Letters**, vol. 86 (1), pp. 14004, **2009**).

[*An enthalpy based thermal lattice Boltzmann model is introduced for simulating a class of strongly coupled thermo-hydrodynamic problems. Novelty of the model lies in the formulation of an enthalpy density distribution function to simulate the temperature field, in place of existing internal energy density distribution function. The proposed model has a clear advantage over the earlier internal energy density distribution function based thermal lattice Boltzmann model, in a sense that it can simulate certain classes of thermofluidic transport problems without facing mathematical difficulties in handling with additional energy source terms.*]

70. Entropy generation analysis for the free surface turbulent flow during laser material processing, **Dipankar Chatterjee**, Suman Chakraborty (**International Journal of Numerical Methods for Heat and Fluid Flow**, vol. 19, pp. 303-328, **2009**).

[*Lasers have been widely used in many industrial applications. Modelling and energy analysis of the laser-oriented processes are essential for process optimization and control. In the present analysis, a three-*

dimensional transient macroscopic numerical model is developed to describe the turbulent transport phenomena during a typical laser surface alloying process and subsequently, the energy analysis is carried out to predict the entropy generation as well as the first and second law efficiencies. A modified k-ε model is used to address turbulent molten metal-pool convection in the presence of a continuously evolving phase-change interface. The phase change aspects are addressed using a modified enthalpy-porosity technique. A kinetic theory approach is adopted for modelling evaporation from the top surface of the molten pool. The numerical results are compared with available experimental results and a very good agreement is found.]

71. An enthalpy-source based lattice Boltzmann model for conduction dominated phase change of pure substances, **Dipankar Chatterjee**, Suman Chakraborty (**International Journal of Thermal Sciences**, vol. 47, pp. 552-559, **2008**).

[*An enthalpy-source based novel lattice Boltzmann technique is formulated for numerical simulation of conduction-dominated phase change processes of single-component systems. The proposed model is based on a classical lattice Boltzmann scheme for description of internal energy evolution with a fixed-grid enthalpy-based formulation for capturing the phase boundary evolution in an implicit fashion. A single particle density distribution function is used for calculating the thermal variable. The macroscopic energy equation is found to be recovered following the Chapman-Enskog multiscale expansion procedure. It is also found that predictions from the present model agree excellently with results obtained from established analytical/numerical models.*]

72. An enthalpy based hybrid lattice Boltzmann method for modeling solid-liquid phase transition in presence of convective transport, Suman Chakraborty, **Dipankar Chatterjee**, (**Journal of Fluid Mechanics**, vol. 592, pp. 155-175, **2007**).

[*An extended lattice Boltzmann model is developed for simulating the convection–diffusion phenomena associated with solid–liquid phase transition processes. Macroscopic hydrodynamic variables are obtained through the solution of an evolution equation of a single-particle density distribution function, whereas, the macroscopic temperature field is obtained by solving auxiliary scalar transport equations. The novelty of the present methodology lies in the formulation of an enthalpy-based approach for phase-change modelling within a lattice-Boltzmann framework, in a thermodynamically consistent manner. Thermofluidic aspects of phase transition are handled by means of a modified enthalpy–porosity formulation, in conjunction with an appropriate enthalpy-updating closure scheme. Lattice-Boltzmann simulations of melting of pure gallium in a rectangular enclosure, Rayleigh–Benard convection in the presence of directional solidification in a top-cooled cavity, and crystal growth during solidification of an undercooled melt agree well with the numerical and experimental results available in the literature, and provide substantial evidence regarding the upscaled computational economy provided by the present methodology.*]

73. A hybrid lattice Boltzmann model for solid-liquid phase transition in presence of fluid flow, **Dipankar Chatterjee**, Suman Chakraborty (**Physics Letters A**, vol. 351, pp. 359-367, **2006**).

[*In the present study, a hybrid lattice Boltzmann methodology is developed for simulating convection-diffusion transport processes pertinent to melting/solidification problems. The model is derived by coupling a modified thermal lattice Boltzmann model with an adapted enthalpy-porosity technique. The hydrodynamic variables are simulated using a single particle density distribution function, whereas the thermodynamic variables are obtained from an enthalpy density distribution function. An exhaustive comparison with other standard methods reveals that the proposed technique is computationally more convenient to be implemented for solving topologically-complicated phase boundary evolution problems during solidification of pure materials, such as the growth of an equiaxed dendrite in an undercooled melt.*]

74. Effect of process parameters on turbulent transport in a laser surface alloying process, **Dipankar Chatterjee**, Nilanjan Chakraborty, Suman Chakraborty (**Journal of Laser Application**, vol. 18, pp. 138-150, **2006**).

[*A three-dimensional, transient numerical model is used for analysing turbulent momentum, heat and mass transport in typical laser surface alloying processes. The Reynolds averaged conservation equations are solved using a finite volume approach. The turbulent transport is accounted for by a suitably modified high*

Reynolds number k-ε model in presence of a continuously evolving phase-change interface. Phase change aspects of the problem are addressed using a modified enthalpy-porosity technique. Subsequently, the developed turbulent transport model is used to simulate single pass laser surface alloying processes with different sets of processing parameters, such as laser power, scanning speed, and powder federate, in order to assess their influences on molten pool geometry and dynamics, cooling rates, as well as on species concentration distribution inside the substrate. In order to investigate the effects of turbulence, the parametric studies for the present turbulent simulation are compared with corresponding laminar simulation results. Significant differences are observed on comparing the laminar and turbulent simulation results, which provide valuable insights for controlling the process parameters based on the manufacturing needs.]

75. An entropy generation analysis of turbulent transport in a laser surface alloying process, **Dipankar Chatterjee**, Suman Chakraborty (**Material Science and Technology**, vol. 22, pp. 627-633, **2006**).

[*A numerical formulation involving second law of Thermodynamics is examined for analysis of three-dimensional surface tension driven turbulent transport during laser materials processing. A modified k-ε model is used to address turbulent melt-pool convection in the presence of a continuously evolving phase-change interface. Phase change aspects are addressed using a modified enthalpy-porosity technique. It is revealed that both energetic and exergetic efficiencies increase with increase in laser power, and remain insensitive to changes in powder feedrate, over the range of parameters chosen. However, the energetic efficiency is found to increase with the laser scanning speed, whereas the exergetic efficiency shows a reverse trend. Accordingly, a trade-off in the laser scanning speed can be sought for, while designing the process parameters, in order to ensure a maximum utilization of the thermodynamic potential offered by the intense laser heat source.*]

76. Parallel simulation study of a laser surface alloying process, **Dipankar Chatterjee**, Supriya Sarkar, Suman Chakraborty (**Numerical Heat Transfer A**, vol. 49, pp. 905-922, **2006**).

[*Numerical simulation of transient phase change processes is massively time consuming, and can benefit from parallel processing to reduce the computational costs. In the present work, an implicit finite volume based three dimensional, transient, turbulent model is developed to describe the transport phenomena for a typical laser surface alloying process, and the code is subsequently parallelized for effective computational economy. Parallelization of the code is done following the pipelining method, which uses temporal parallelism in a distributed computing environment and runs on a cluster of workstations (COW) connected with local area networking (LAN). Equal partitioning of the spatial domain into subdomains along the longer geometric dimension is performed, in order to decompose the computational domain. The message passing interface (MPI) utility is used for communications among the subdomains on each processor. To demonstrate the parallel performance, a systematic analysis is carried out, with upto 16 nodes and various grid sizes. It is found that the overall parallel performance improves up to a certain number of processors, and then degrades with an increase in number of processors, due to communication delays in the LAN and pipeline hazards.*]

77. An enthalpy based lattice Boltzmann model for diffusion dominated solid-liquid phase transformation, **Dipankar Chatterjee**, Suman Chakraborty (**Physics Letters A**, vol. 341, pp. 320-330, **2005**).

[*An extended lattice Boltzmann (LB) formulation is developed for simulation of three-dimensional heat diffusion, coupled with solid-liquid phase transition aspects. Thermodynamic variables are obtained through the evolution equation of a single particle distribution function. Phase change aspects are tackled by an enthalpy-based approach. The proposed model is subsequently utilized to simulate a generic laser surface melting process and compared with the continuum model results obtained by solving the governing equation for energy transport. The comparison shows a potentiality of the developed LB method to solve phase change problems in a computationally effective and convenient manner.*]

78. Large eddy simulation of laser induced surface tension driven flow, **Dipankar Chatterjee**, Suman Chakraborty (**Metallurgical Transactions B**, vol. 36 B, pp. 743-754, **2005**).

[*The turbulent flow inside a laser generated molten pool is investigated by an adapted large eddy simulation (LES) model that incorporates physical considerations pertaining to solid-liquid phase change. A single domain fixed-grid enthalpy-porosity approach is utilized to model the phase change phenomena in presence of a continuously evolving solid-liquid interface. The governing transport equations are simultaneously solved by employing a control volume formulation, in conjunction with an appropriate enthalpy updating closure scheme. To demonstrate the performance of the present model in the context of phase change materials processing, simulation of a typical high power laser melting process is carried out, where effects of turbulent transport can actually be realized. It is found that the present LES based model is more successful in capturing the experimental trends, in comparison to the k-ε based turbulence models often employed to solve similar problems in contemporary research investigations.*]

79. A scaling analysis of turbulent transport in laser surface alloying process, Nilanjan Chakraborty, **Dipankar Chatterjee**, Suman Chakraborty (**Journal of Applied Physics**, vol. 96 (8), pp. 4569-4577, **2004**).

[*A systematic procedure for scaling analysis of three-dimensional transient turbulent momentum, heat and mass transfer in a typical laser surface alloying process is presented. With suitable choices of non-dimensional parameters, the governing equations coupled with appropriate boundary conditions are first scaled, and the relative significance of various terms appearing in them are accordingly analyzed. The analysis is then utilized to predict the orders of magnitude of some important quantities, such as the velocity scale at the top surface, velocity boundary layer thickness, temperature rise in the pool, turbulent kinetic energy and its dissipation rate at the top surface. The scaling predictions are also assessed by comparison with numerical and experimental results, and an excellent agreement is observed in order-of-magnitude sense. Using the scaling predictions, the influence of various processing parameters on the system variables can be well recognized, which enables us to develop a deeper insight into the physical problem of interest. Moreover, some of the quantities predicted from the scaling analysis can be utilized for optimized selection of appropriate grid-size for full numerical simulation of the process.*]

80. Modeling of turbulent transport in laser surface alloying, Nilanjan Chakraborty, **Dipankar Chatterjee**, Suman Chakraborty (**Numerical Heat Transfer A**, vol. 46, pp. 1009-1032, **2004**).

[*A three-dimensional, transient model is developed in order to address the turbulent transport in a typical laser surface alloying process. Turbulent melt-pool convection is taken into account by using a suitably modified high Reynolds number k-ε model in presence of a continuously evolving phase-change interface. The phase change aspects of the problem are addressed using a modified enthalpy-porosity technique. This newly developed mathematical model is subsequently utilized to simulate a typical high power laser surface alloying process, where effects of turbulent transport can actually be realized. In order to investigate the effects of turbulence on laser molten pool convection, simulations with laminar and turbulent transport are carried out for same problem parameters. Significant differences in the molten pool morphology are observed on comparing the laminar and turbulent simulation results. It is also revealed that turbulent simulations yield a much better matching with the experimentally obtained species concentration distribution within the alloyed layer, than corresponding laminar simulations.*]

81. Effects of inlet air swirl and spray cone angle on combustion and emission performance of a liquid fuel spray in a gas turbine combustor, **Dipankar Chatterjee**, A. Datta, A.K. Ghosh, S.K. Som (**Journal of The Institution of Engineers (India): Series C**, vol. 85, pp. 41-46, 2004).

[*An extensive study is carried out to investigate the effect of inlet air swirl and fuel injection angle on the flow and combustion phenomena of a typical diffusion controlled spray combustion process in a can-type gas turbine combustion chamber. A Eulerian-Lagrangian formulation is used for the two phase (gas-droplet) flow, in which the coupling between the two phases is taken care of through interactive source terms. The standard k- ε model with standard wall function treatment is used to model turbulence. The initial spray parameters are specified by a suitable size distribution and a given spray cone angle. The gas phase chemical reaction is modeled using eddy dissipation model. Radiation model for the gas phase based on the discrete ordinate method is adopted in consideration of the gas phase as a gray absorbing emitting* *medium. Thermal and prompt NOx are modeled following extended Zeldovich mechanism and Fenimore reaction.*]

- **(Under Review):**

82. Steady mixed convection in power-law fluids from a heated triangular cylinder, Satish Kumar Gupta, Sudipta Ray, **Dipankar Chatterjee** (**Heat Transfer Research**).

    [*The steady mixed convective heat transfer from a heated triangular cylinder immersed in power-law fluids in an unconfined vertical domain is investigated numerically in this work. The specific aim is to analyze the effect of orientation of the cylinder with respect to the incoming flow on the hydrodynamic and thermal transport of non-Newtonian power-law fluids. Accordingly, two different configurations of the cylinder are chosen; one when the base of the cylinder is facing the flow (C1) and the other when the apex of the triangle is facing the flow (C2). The simulation is performed for the following ranges of physical parameters: Reynolds number* $10 \le Re \le 35$, *Richardson number* $0 \le Ri \le 2$, *power law index* $0.4 \le n \le 1.8$ *and Prandtl number* $Pr = 50$. *The flow and thermal fields are visualized through the streamline and isotherm contours at the close proximity of the heated object for various Re, Ri and n. The distribution of surface pressure coefficient and local Nusselt number provide further insight of the hydrodynamic and thermal characteristics. Finally, the total drag coefficient and average Nusselt numbers on the surface of the cylinder are computed to explore the overall macroscopic behavior of the involved thermo-hydrodynamics. The drag decreases with Re and increases with Ri for both the configurations. The total drag decreases with n for pure forced flow; it remains insensitive to n when the free and forced convection are equally important, whereas, it increases with n when the natural convection dominates. The average Nusselt number shows positive dependence on Re and Ri. The shear thinning behavior (* $n < 1$*) augments the heat transfer, whereas the shear thickening behavior (* $n > 1$*) somewhat impedes it. The flow separation is observed to be more when the apex of the cylinder is facing the flow (C2). The average heat transfer, measured in terms of the Nusselt number, and the total drag on the cylinder are also found higher for that configuration (C2).*]

83. Unsteady CFD simulation of 3D AUV hull at different angles of attack, Sudipta Ray, **Dipankar Chatterjee**, Sambhunath Nandy (**Journal of Naval Architecture and Marine Engineering**).

    [*An unsteady, three-dimensional flow simulation is carried out over the bare hull of the autonomous underwater vehicle currently being developed by CSIR-CMERI, Durgapur, India at various angles of attack with the help of a Finite Volume-based CFD software. The purpose of the study is to provide estimation of various hydrodynamic forces acting on the bare hull at different angles of operation. The operating range of velocity of the vehicle is 0-6 knot (0-3 m/s), considering up to 2 knots of upstream current. For the purpose of the CFD simulation, the widely-implemented RANS approach is used, wherein the turbulent transport equations are solved using the low-Re version of the SST κ-ω turbulence model. The motion of the vehicle is considered within a range of the pitch angle* $\left(0^0 \le \alpha \le 20^0\right)$. *The results are presented in terms of variations of the relevant hydrodynamic parameters. The effects of the angle of attack on the drag and pressure coefficients are discussed in detail.*]

84. Rotation induced flow suppression around a circular cylinder, **Dipankar Chatterjee**, Satish Kumar Gupta, Sudipta Ray (**CFD Letters**).

    [*We demonstrate through numerical computation the phenomena of flow suppression around a circular cylinder subjected to uniform rotation. The Reynolds numbers based on the free stream flow are kept in the range 10-40. In this range of Reynolds number, the free stream flow around a stationary circular cylinder remains steady and separated with formation of counter-rotating vortices behind the body.. Rotation causes stabilization of the flow field resulting in flow suppression. The critical rotation rates for which the flow behind the cylinder is completely suppressed in the said range of Reynolds number are computed to establish the rotation induced flow suppression.*]

85. An effective numerical strategy for modeling high energy laser surface alloying process, **Dipankar Chatterjee** (**International Communications in Heat and Mass Transfer**).

[*We perform a comparative study on the effectiveness of some macroscopic and mesoscopic models deployed for addressing the thermofluidic transport during high power laser surface alloying process. The macroscopic models include laminar, k-ε turbulence and large eddy simulation (LES) models, whereas a lattice Boltzmann (LB) model is invoked under the mesoscopic paradigm. Results show that LB and LES models are more successful in capturing the experimental trends, in comparison to laminar and k-ε turbulence models.*]

86. Mixed convective transport around staggered rows of square cylinders, **Dipankar Chatterjee**, Saugata Patra, Bittagopal Mondal (**Journal of Energy, Heat and Mass Transfer**)

[*The unsteady mixed convective transport around multiple bluff objects placed in a staggered configuration with respect to a uniform free stream flow is analyzed through two-dimensional numerical computation. The bluff objects are identical in shape and size with square cross-section and arranged in two different rows in a staggered fashion within an unconfined domain. A small temperature difference between the objects and the free stream results in the free convection in addition to the forced flow. Simulation is carried out using a finite volume based method considering a uniform cross flow of air (Prandtl number = 0.71) at a moderate Reynolds number (= 100). The transverse spacing between the cylinders may anticipated to influence significantly the wake dynamics, which in turn affects the thermal transport. Simultaneously, the mixed convective strength also influences the wake dynamics and vortex structure formation. An interplay between these two effects aptly dictates the resulting flow dynamics and associated thermal transport. Accordingly, the dimensionless transverse spacing is varied (= 1, 3 and 5) along with the mixed convective strength (Richardson number = 0-2). It is observed that even at such a moderate Reynolds number, the flow and thermal fields show chaotic nature at smaller transverse spacing. However, at larger spacing, the usual unsteady vortex dynamics persists. Very interestingly it is observed that the chaotic flow at smaller transverse spacing reduces its instability to become unsteady periodic at larger strength of the thermal buoyancy. The average heat transfer from the cylinders is found more at smaller transverse spacings and it increases with increasing mixed convective strength.*]

87. MHD Natural Convection in a Square Enclosure with Four Circular Cylinders Positioned at Different Rectangular Locations, **Dipankar Chatterjee**, Satis Kumar Gupta (**Heat Transfer Engineering**)

[*We deploy a finite volume numerical computation to investigate the two dimensional hydromagnetic natural convection in a cooled square enclosure in presence of four inner heated circular cylinders with identical shape. The inner circular cylinders are placed in a rectangular array with equal distance away from each other within the enclosure and moving along the diagonals of the enclosure. All the walls of the enclosure are kept isothermal with temperatures less than that of the cylinders. A uniform magnetic field is applied along the horizontal direction normal to the vertical wall. All solid walls are assumed electrically insulated. Simulations are performed for a range of the controlling parameters such as the Rayleigh number* 10^3 *to* 10^6*, Hartmann number 0 to 50 and the dimensionless horizontal and vertical distance from the centre of a cylinder to centre of another cylinder 0.3 to 0.7. The study specifically aims to understand the effects of the location of the cylinders in the enclosure on the magnetoconvective transport, when they moved along the diagonals of the enclosure. It is observed that the unsteady behavior of the flow and thermal fields at relatively larger Rayleigh numbers and for some cylinder position are suppressed by imposition of the magnetic field. The heat transfer strongly depends on the position of the cylinders and the strength of the magnetic field. Hence, by controlling the position of the objects and the magnetic field strength, a significant control on the hydrodynamic and thermal transport can be achieved*.]

88. Computational Modeling of Gas-Bubble Formation through a Single Submerged Orifice, Vijay Kumar Prasad, Satya Prakash Singh, **Dipankar Chatterjee** (**The Canadian Journal of Chemical Engineering**)

[*A two-dimensional numerical simulation is carried out to analyze the dynamics of gas bubble formation from a single submerged orifice in an immiscible Newtonian liquid under the condition of constant gas inflow rate using a finite volume based CFD technique. Two conditions for the ambient liquid are considered, namely the liquid in quiescent condition and the liquid as a co-flowing stream with the gas. The full cycle from bubble formation to its detachment and the corresponding dynamics are simulated by using both the Volume of Fluid method (VOF) and Coupled Level Set and Volume of Fluid method (CLSVOF). Although, both are front capturing techniques of Eulerian family, they possess some distinct properties in themselves. The CLSVOF method combines the advantages of Level Set method with that of Volume of Fluid method. It is observed that the CLSVOF method is more successful in predicting the interface sharpness in comparison to the VOF method only. The study includes :- (i) time sequence profile of bubble formation to clearly represent bubble growth, neck formation, and bubble breakup at given Weber (We), Reynolds (Re), Bond (Bo) numbers and liquid to gas mean velocity ratio (v_r); (ii) bubble growth history for different v_r and at constant Re, We and Bo; (iii) comparison between results obtained by VOF and CLSVOF at given v_r, We, Re, and Bo; and (iv) Bubble size and bubble formation time, and finally bubble coalescence phenomenon and technique for its inhibition.*]

89. Computational investigation of convective heat transfer in pipeline flow of multi-sized mono dispersed fly ash-water slurry, Bibhuti Bhusan Nayak, **Dipankar Chatterjee**, Amar Nath Mullick (**The Canadian Journal of Chemical Engineering**)

[*A three-dimensional numerical investigation is performed to understand the influence of dispersed particles on the thermo-fluidic transport of liquid-solid slurry in a horizontal pipe. The simulation is carried out by deploying an Eulerian multiphase model of a commercial Computational Fluid Dynamics (CFD) code Ansys FLUENT. Spherical coal fly ash particles having mass median diameters of 4, 8, 13, 34 and 78 μm suspended in water for a mean flow velocity ranging from 1-5 m/s and particle concentrations within 0-50% by volume for each velocity are considered as the dispersed phase. The pipe wall is kept at an isothermal condition of 400 K whereas the slurry enters the pipeline at a temperature of 300 K. The results illustrate that for all particle sizes, heat transfer ratio is found to increase with particle concentration up to 3% and then gradually decrease with increased particle concentration and mean velocity of flow. Moreover, the heat transfer ratio and the relative pressure drop increase with the particle size at higher concentrations and mean velocities.*]

90. Computational Modeling of Gas-Bubble Dynamics through Two Adjacent Submerged Orifices, Vijay Kumar Prasad, Satya Prakash Singh, **Dipankar Chatterjee** (Numerical Simulation of Gas-Bubble Formation through Two Adjacent Submerged Orifices, Vijay Kumar Prasad, Satya Prakash Singh, **Dipankar Chatterjee** (**European Journal of Mechanics - B/Fluids**)

[*In this work a two-dimensional numerical simulation is carried out to analyze the dynamics of gas bubble formation at two adjacent orifices submerged in an immiscible Newtonian liquid under the condition of constant gas inflow rate using a finite volume based commercial CFD solver Ansys Fluent. Two conditions for ambient liquid are considered, namely the liquid in quiescent condition and the liquid as a co-flowing stream with the gas. The full cycle from the bubble formation to its detachment and the corresponding dynamics are simulated by using the Volume of Fluid (VOF) method, which is one of the efficient front capturing techniques of Eulerian family. The study includes :- (i) Time sequence profile of bubble formation to clearly represent the bubble growth, neck formation, and bubble breakup at given Weber number (We), Reynolds number (Re), Bond number (Bo) and Liquid to gas mean velocity ratio (V_r); (ii) Bubble growth history for different V_r and at constant Re, We and Bo; (iii) Effect of orifice spacing on bubbling synchronicity and frequency at given V_r, We, Re and Bo ; and (iv) Synchronous bubbling regime, bubble size and formation time, bubble coalescence phenomenon and finally the technique for its inhibition.*]

91. Rotation induced flow suppression around two tandem circular cylinders at low Reynolds number, **Dipankar Chatterjee**, Krishan Gupta, Virendra Kumar, Sachin Abraham Varghese, Biswajit Ghosh (**Computers and Fluids**)

92. Thermohydrodynamics around rotating tandem circular cylinders at low Reynolds numbers, **Dipankar Chatterjee**, Krishan Gupta, Virendra Kumar, Sachin Abraham Varghese, Biswajit Ghosh (**International Journal of Heat and Mass Transfer**)

CONFERENCE PUBLICATIONS

- **2015 (Total No: 3)**

1. Numerical simulation of turbulent forced convection heat transfer and particle distribution of Fly ash-water slurry flow in horizontal pipe bends and straight pipes, Bibhuti Bhusan Nayak, **Dipankar Chatterjee**, Amar Nath Mullick, **60th Congress of the Indian Society of Theoretical and Applied Mechanics (ISTAM) (An International Conference), December 16-19, 2015, IIT Kharagpur, India**.

2. भूमिगत कोयला गैसीकरण (अंडरग्राउंड कोल गैसिफिकेशन-यूसीजी) में कैविटी वृद्धि की मॉडलिंग, **दीपांकर चटर्जी**, सत्य प्रकाश सिंह, **भारत में कोयला आधारित गॅस उद्योग की चुनोतियाँ एवं विकल्प राजभसा हिन्दी मे आयोजित एक दिवसया राष्ट्रीय संगोष्ठी, 11 मई 2015, सीएसआईआर- केंद्रीय खनन एवं ईंधन अनुसंधान संस्थान, धनबाद, भारत**

3. Numerical analysis of Goldschmied geometry with boundary-layer suction, Sudipta Ray, Soumen Sen, **Dipankar Chatterjee**, **International Symposium on Underwater Technology (UT15), February 23-25, 2015, National Institute of Ocean Technology, Chennai, India**

- **2014 (Total No: 2)**

4. Numerical investigations of heat transfer between wall and water-fly ash slurry flow in horizontal pipes, Bibhuti Bhusan Nayak, Amar Nath Mullick, Satish Kumar Gupta, **Dipankar Chatterjee**, **5th International and 41st National Conference on Fluid Mechanics and Fluid Power (FMFP 2014), 12-14 December, 2014, IIT Kanpur, India**.

5. Modelling turbulent transport in high energy materials processing application: large eddy simulation and parallelization, **Dipankar Chatterjee**, **National Conference on Emerging Trends in Physics of Fluids & Solids (NCETPFS – 2014), March 06-07, 2014, Jadavpur University, Kolkata, India.**

- **2013 (Total No: 7)**

6. Low-dimensional chaos for flow past staggered rows of square cylinders, **Dipankar Chatterjee**, Bittagopal Mondal, Gautam Biswas, **Energy System Modeling and Optimization Conference (ESMOC 2013), December 9-11, 2013, NIT Durgapur, India.**

7. Magnetoconvective transport in a vertical lid-driven cavity in presence of a heat conducting rotating circular cylinder, Kanchan Chatterjee, **Dipankar Chatterjee**, Nirmal Baran Hui, Bittagopal Mondal, **Energy System Modeling and Optimization Conference (ESMOC 2013), December 9-11, 2013, NIT Durgapur, India.**

8. Magnetohydrodynamic flow and heat transfer around a circular cylinder in an unconfined medium, Satish Kumar Gupta, **Dipankar Chatterjee**, Bittagopal Mondal, **International Conference On Future Trends In Structural, Civil, Environmental and Mechanical Engineering - FTSCEM, July 13-14 , 2013, Bangkok, Thailand.**

9. Multiscale Modeling of Transport Phenomena during High Energy Materials Processing Applications, **Dipankar Chatterjee**, Shantanu Dey**, 61st Indian Foundry Congress, January 27-29, 2013, Kolkata, India.**

10. Numerical Simulation of Flow around a Three-Wheeled Vehicle, Ajoy Kuchlyan, Santanu Dey, **Dipankar Chatterjee,** Palash Kumar Maji, **National Conference on Mechanical Engineering : Retrospect and Prospect (NCMERP - 2013), February 2-3, Birbhum Institute of Engineering & Technology (BIET), Suri, Birbhum, India.**

11. Hydromagnetic Mixed Convection in a Vertical Lid-Driven Cavity including a Heat Conducting Rotating Circular Cylinder, Bittagopal Mondal, Pabitra Halder, **Dipankar Chatterjee**, **National Conference on Mechanical Engineering : Retrospect and Prospect (NCMERP - 2013), February 2-3, Birbhum Institute of Engineering & Technology (BIET), Suri, Birbhum, India.**

12. Flow and heat transfer around a square cylinder at low Reynolds and Hartmann numbers, **Dipankar Chatterjee**, Kanchan Chatterjee, Chiranjit Sinha, **National Conference on Mechanical Engineering : Retrospect and Prospect (NCMERP - 2013), February 2-3, Birbhum Institute of Engineering & Technology (BIET), Suri, Birbhum, India.**

- **2012 (Total No: 1)**

13. Thermohydrodynamic simulation of a DC (direct current) magnetohydrodynamic (MHD) micropump, **Dipankar Chatterjee**, **International Conference on Microactuators and Micromechanisms, January 19-20, 2012, CSIR-CMERI, Durgapur, India.**

- **2011 (Total No: 6)**

14. Numerical Simulation of Mixed Convection Heat Transfer past In-Line Square Cylinders, Md. Raja**, Dipankar Chatterjee**, **5th International Conference on Advances in Mechanical Engineering, June 06-08, 2011, Sardar Vallabhbhai National Institute of Technology, Surat, India.**

15. Water Droplet Transport Simulation in Serpentine Gas Flow Channel of PEM Fuel Cell**,** Bittagopal Mondal, **Dipankar Chatterjee**, Md. Raja**, 11th Asian International Conference on Fluid Machinery, Nov. 21-23, 2011, IIT Madras, Chennai, India.**

16. MHD flow and heat transfer around a square cylinder at low Reynolds numbers, **Dipankar Chatterjee,** Kanchan Chatterjee, Nirmal Baran Hui, **The International Congresses on Theoretical and Applied Mechanics, December 12-16, 2011, IIT Kanpur, India.**

17. Vortex degeneration due to thermal buoyancy effect for upward flow around bluff obstacles, **Dipankar Chatterjee,** Bittagopal Mondal, Gautam Biswas**, 21st National & 10th International ISHMT-ASME Heat and Mass Transfer Conference, Dec. 27-30, 2011, IIT Madras, Chennai, India.**

18. A computational study on pollutant generation in a gas turbine combustor, **Dipankar Chatterjee,** Abhishek Mandal, Pradip Kumar Chatterjee, **21st National & 10th International ISHMT-ASME Heat and Mass Transfer Conference, Dec. 27-30, 2011, IIT Madras, Chennai, India.**

19. Forced convection heat transfer around a circular cylinder subjected to streamwise and transverse magnetic fields at low Reynolds numbers, **Dipankar Chatterjee,** Kanchan Chatterjee, Nirmal Baran Hui, **21st National & 10th International ISHMT-ASME Heat and Mass Transfer Conference, Dec. 27-30, 2011, IIT Madras, Chennai, India.**

- **2010 (Total No: 3)**

20. Vibrational instabilities in supercritical fluids, **D. Chatterjee**, S. Amiroudine, D. Beysens, **20th National & 9th International ISHMT-ASME Heat and Mass Transfer Conference, January 4-6, 2010, IIT Bombay, Mumbai, India.**

21. Lattice Boltzmann Simulation of Laser Surface Melting Process, **Dipankar Chatterjee**, **Workshop on Application of Laser in Mechanical Industries and Seminar on Application of Laser in Materials Processing, 7-9 January, 2010, Jadavpur University, India.**

22. Lattice Boltzmann simulation of transport phenomena in high power materials processing application, **Dipankar Chatterjee**, **5th International Conference on Theoretical, Applied, Computational and Experimental Mechanics, 27-29 December, 2010, IIT Kharagpur, India.**

- **2008 (Total No: 2)**

23. A novel thermal lattice Boltzmann model for non-isothermal systems, **Dipankar Chatterjee**, Suman Chakraborty**, 19th National & 8th International ISHMT-ASME Heat and Mass Transfer Conference, January 3 - 5, 2008, Jawaharlal Nehru Technological University (JNTU) College of Engineering, Hyderabad, India.**

24. Numerical Investigations of Laser-Induced Surface-tension Driven Flows by an Axisymmetric Lattice Boltzmann Model, **Dipankar Chatterjee**, Suman Chakraborty, **Workshop on Advanced Laser Manufacturing, 17-18 January, 2008, Jadavpur University, India.**

- **2006 (Total No: 3)**

25. A lattice Boltzmann method for solid-liquid phase transition modeling, **Dipankar Chatterjee**, Suman Chakraborty; **18th National & 7th International ISHMT-ASME Heat and Mass Transfer Conference, January 4 - 6, 2006, IIT Guwahati, India.**

26. Turbulence modeling strategies for molten pool convection in laser aided surface processing of materials, S. Chakraborty, N. Chakraborty, **D. Chatterjee**, S. Sarkar, **Workshop on Advanced Laser Manufacturing, February, 2006, Jadavpur University, India.**

27. A Novel Thermal Lattice Boltzmann Model For Fluid Dynamic Application, **Dipankar Chatterjee**, **in Recent Trends in Emerging Technologies, February, 2006, B.P. Poddar Institute of Management & Technology, Kolkata, India.**

- **2004 (Total No: 2)**

28. Large eddy simulation in a laser surface melting process, **Dipankar Chatterjee**, Amit Kumar Ghosh, Suman Chakraborty; **Workshop on Advanced Laser Manufacturing, February, 2004, Jadavpur University, India.**

29. Mathematical Model of NO_x Formation for the Turbulent Diffusion Flame in a Gas Turbine Combustor, **Dipankar Chatterjee**, Swasti S. Mondal, **31st National Conference on Fluid Mechanics and Fluid Power, December, 2004, Jadavpur University, Kolkata, India.**

- **2003 (Total No: 3)**

30. Effect of inlet air swirl and spray cone angle on combustion and emission performance of a spray combustion process in gas turbine combustor, **Dipankar Chatterjee**, A. Datta, S.K. Som, A.K. Ghosh; **Seventeenth National Convention of Aerospace Engineers and National Seminar on Indian Aerospace Vehicles; November, 2003, BIT Meshra, Ranchi, India.**

31. Effects of various turbulence models on the overall solidification behavior of a pure substance in a top cooled rectangular cavity, **Dipankar Chatterjee**, Suman Chakraborty; **Fluent User Conference; November, 2003, Pune, India.**

32. Modeling of turbulent transport in laser surface alloying, Nilanjan Chakraborty, **Dipankar Chatterjee**, Suman Chakraborty; **ASME International Mechanical Engineering Conference and R&D Expo; November, 2003, Washington D.C., USA.**

- **2002 (Total No: 1)**

33. Effect of inlet air swirl on flow and NO_x formation in a gas turbine combustor, **Dipankar Chatterjee**, A. Datta, S.K. Som, A.K. Ghosh; **5th Annual Symposium on CFD; August, 2002, Bangalore, India**.

BOOKS AUTHORED

1. "Mechanical Sciences – II (Thermodynamics & Fluid Mechanics)", **Dr. Dipankar Chatterjee**, Mr. Subhajit Datta, **SCI Tech Publications, Chennai, 2010**.

2. Lattice Boltzmann Modeling for Melting/Solidification Processes, **Dr. Dipankar Chatterjee,** in "Hydrodynamics" Ed. Harry Edmar Schulz, **INTECH, Croatia, 2011.**

3. Lattice-Boltzmann Methods for Phase-Changing Flow, **Dr. Dipankar Chatterjee, in "Microfluidics and Microscale Transport Processes" Ed. S. Chakraborty, Taylor and Francis, 2012.**

REVIEWS

1. Reviewer of the book "**Gas Turbines**" by V. Ganesan.

2. Reviewer of the book chapter "Lattice Boltzmann Method and Its Applications in Microfluidics" by Junfeng Zhang of **Microfluidics Nanofluidics Handbook** from CRC Press/Taylor & Francis Group, LLC.

3. **Journal**:

- Advances in Space Research;
- Aerospace Science and Technology;
- AIChE;
- Ain Shams Engineering Journal;
- American Journal of Heat and Mass Transfer;
- Applied Bionics and Biomechanics
- Applied Mathematical Modelling;
- Applied Thermal Engineering;
- Biomicrofluidics;
- Chemical Engineering & Technology;
- Chemical Engineering Science;
- Computers and Fluids;
- Defence Science Journal;
- Energy;
- Experimental Thermal and Fluid Science;
- Fusion Engineering and Design;
- Heat and Mass Transfer;
- Heat Transfer - Asian Research;
- Heat Transfer Engineering;
- Heat Transfer Research
- Industrial and Engineering Chemistry Research;
- International Communications in Heat and Mass Transfer;
- International Journal of Heat and Mass Transfer;
- International Journal of Numerical Methods for Heat and Fluid Flow;
- International Journal of Thermal Sciences;
- Iranian Journal of Science and Technology;
- Journal of Applied Fluid Mechanics;
- Journal of Applied Mathematics;
- Journal of Applied Mechanical Engineering;
- Journal of Computational Physics;
- Journal of Fluids and Structures;
- Journal of Fluids Engineering - Transactions of ASME;
- Journal of Heat Transfer - Transactions of ASME;
- Journal of Institution of Engineers (C);
- Journal of Mechanical Engineering Science: Part C;
- Journal of the Taiwan Institute of Chemical Engineers;
- Journal of Thermophysics and Heat Transfer;
- Korea-Australia Rheology Journal;
- Korean Journal of Chemical Engineering;
- Meccanica;
- Numerical Heat Transfer A;

- Physics Letters A;
- PLOS ONE;
- Powder Technology;
- Proceedings of the National Academy of Sciences, India Section A: Physical Sciences;
- Progress in Computational Fluid Dynamics;
- Scientica Iranica;
- Scientific Reports;
- The Scientific World Journal;
- Thermal Science;

4. **MTech Thesis Review:**

- "Surface reconstruction using neural network B-spline method", Ahmad Najam saquib, National Institute of Technology Durgapur, Durgapur-713209, 2013.
- "Numerical study of shock/shock interaction in hypersonic double wedge flows", Rahul Bandlas, Indian Institute of Engineering Science and Technology, Shibpur, 2015.
- "Numerical simulation of convective heat transfer in square cavity utilizing nanofluids", Saurabh Gupta, Indian Institute of Engineering Science and Technology, Shibpur, 2015.

5. **PhD Thesis Review**:

- "Methodology for Improvement of Fin Performance and Optimum design Parameters for Electronic Cooling", Dipankar Bhanja, Jadavpur University, Kolkata-32, 2013.
- "Investigation on Some Aspects of Through Transmission Laser Welding of Plastics", John Deb Barma, Jadavpur University, Kolkata-32, 2013.
- "Some flow problems in Newtonian or non-Newtonian fluids having heat and mass transfer phenomena", Raju Haldar, Jadavpur University, Kolkata-32, 2013.

JOURNAL EDITORSHIP

- ✓ **Associate Editor** of "**Frontiers in Mechanical Engineering (Thermal and Mass Transport)**", 2015.
- ✓ **Associate Editor** of "**Journal of Advances in Mechanical Engineering and Sicence**", 2015.
- ✓ **Editorial Board Member** of "**Journal of Advanced Thermal Science Research**", 2014.
- ✓ **Editorial Board Member** of "**American Journal of Heat and Mass Transfer**", 2013.

M-TECH GUIDANCE

1. Debabrata Das, "**Designing a Simulation Model for Hydro-pneumatic Suspension System**", National Institute of Technology, Durgapur, May, 2013.

2. Ramgopal Mishra, "**Numerical Investigation of Transient Mixed Covective Transport in a Square Enclosure Containing Heated Cylinders**", National Institute of Technology, Durgapur, May, 2016.

PhD GUIDANCE

1. Kanchan Chatterjee, "**Hydromagnetic Flow and Heat Transfer around Bluff Obstacles at Low Reynolds Numbers**", National Institute of Technology, Durgapur, October, 2013.

2. Bibhuti Bhusan Nayak, "**Influence of Dispersed Particles on Hydrodynamic and Thermal Behaviors of Internal Multiphase Transportation**", National Institute of Technology, Durgapur (Undergoing)

3. Santanu Dey, "**Flow Characteristics over Double Delta Wing**", AcSIR (Undergoing)

4. Piyush Pant, "**Design and development of multimaterial deposition**", Jadavpur University (Undergoing)

5. Vijay Kumar Prasad, "Understanding the efficacy of using micro-bubbles for targeted drug delivery, AcSIR (Undergoing)

AWARDS & HONOURS:

1. Recipient of the Merit Certificate of **All India Science Talent Search Examination** (AISTSE)-1986.
2. Recipient of the **National Merit Scholarship,** 1989.
3. Recipient of the **GATE** (**Graduate Aptitude Test in Engineering**) Fellowship-1996.
4. Recipient of the **Post Doctoral Fellowship** from Arts et Métiers Paris Tech, France-2008.
5. Selected in the **Marquis Who's Who in the World** 2010.
6. Recipient of the **Metallurgist for the year 2011-12** by **Indian Institute of Metals Durgapur Chapter** for the Contribution in "Computational Fluid Dynamics in Materials Processing Applications", 2012.
7. **Visiting Faculty** at National Institute of Technology Durgapur 2015.

SPONSORED PROJECTS:

Sl. No.	Title of the Project	Budget	Duration	Funding Agency	Current Status
1.	Two-layer electroosmotic flows in microchannels under time periodic electrical fields **Role: Principal Investigator**	36.56 Lacs	3 years	CSIR-CMERI	**Successfully Completed** Start date: 15/06/2010 End date: 14/06/2013
2.	Computational modelling of transport phenomena in high energy materials processing application: large eddy simulation and parallelization **Role: Principal Investigator**	19.709 Lacs	3 years	DST	**Successfully Completed**, Start date: 19/09/2011 End date: 31/03/2014
3.	Development of underground coal gasification technology in India(CoalGasUrja) (Mathematical Modelling and Simulation Studies of Underground Coal Gasification (UCG) Process) **Role: Project Leader from CSI-CMERI**	85.90 Lacs	5 Years	CSIR (Under 12th Five Year Plan) Nodal Lab: CSIR-CIMFR	**Ongoing** (Start Date: 01/04/2012, End Date: 31/03/2017)
4.	Study of the dynamics of gas-bubble in a flowing liquid environment **Role: Co-Investigator**	34.20 Lacs	3 Years	CSIR-CMERI	**Successfully Completed** Start date: 15/06/2010 End date: 14/06/2013
5.	Numerical study of water droplet mobility in serpentine gas flow channel of PEM fuel cell **Role: Co-Investigator**	24.00 Lacs	3 Years	CSIR-CMERI	**Successfully Completed** Start date: 01/04/ 2011 End date: 31/03/2014
6.	Computational modelling of fluid flow and heat transfer around fixed and/or moving objects **Role: Co-Investigator**	28.95 Lacs	3 Years	CSIR-CMERI	**Successfully Completed** Start date: 10/03/ 2011 End date: 09/03/2014

7.	Modelling Fish Locomotion in Turbulent Vortices **Role: Co-Investigator**	18.00 Lacs	3 years	CSIR-CMERI	**Ongoing** (Start Date: 26/09/2013, End Date: 25.09.2016)
8.	Rheo Pressure Die casting of JIS ADC 12 Aluminium alloy **Role: Co-Project Leader**	26.10 Lacs	15 Months	M/S Sona Koyo Steering Systems Ltd., Gurgaon	**Ongoing** (Start Date: Nov. 2014)
9.	Autonomous underwater robotics (UnWar) **Role: Team Member**	4000.93 Lacs	5 Years	CSIR (Mega Project Under 12th Five Year Plan) Nodal Lab: CSIR-CMERI	**Ongoing** (Start Date: 01/04/2012, End Date: 31.03.2017)

1. Two-layer electroosmotic flows in microchannels under time periodic electrical fields:

   [***Project summary****: Interfacial characteristics of two-layer electroosmotic micro-flows occurring under the action of time periodic electrical fields will be investigated in details in this work. The upper fluid layer may considered to be electrically non-conducting, so that the electroosmosis in the bottom layer transmits the necessary shear to the upper layer for inducing a relative motion between the two layers, as a consequence of an applied electrical field. In addition, pressure-driven flow actuating mechanisms may also be operative in tandem. A comprehensive numerical solution strategy will be developed for determining the interfacial evolution and instability characteristics as a function of the applied electric field as well as the other pertinent system parameters. The numerical results will also be compared with the asymptotic analytical solutions for certain limiting cases. Particular attention will be focused on assessing the role of the time-varying electrical fields towards influencing the rupture of thin liquid films amidst unstable interfaces*.]

2. Computational modeling of transport phenomena in high energy materials processing application: large eddy simulation and parallelization:

   [***Project summary****: A comprehensive three-dimensional numerical study is proposed to be carried out in order to predict the coupled turbulent momentum, heat and species transport during molten metal-pool convection in association with continuous evolution of solid-liquid interface typically encountered in high energy materials processing applications. The turbulent aspect will be handled by a large eddy simulation (LES) model and the phase changing phenomena is going to be taken care of by a modified enthalpy-porosity technique along with an adapted enthalpy updating scheme. The proposed finite volume based LES model will subsequently be parallelized for effective computational economy. To demonstrate the effectiveness of the present model, a systematic analysis will be carried out to simulate a typical high power laser surface alloying process, where the effects of turbulent transport can actually be realized. The results obtained will be of extreme significance for accurate prediction of thermo-mechanical characteristics of the resolidified microstructure*.]

3. Mathematical Modeling and Simulation Studies of Underground Coal Gasification (UCG) Process:

   [***Project summary****: A comprehensive computational fluid dynamics (CFD) based model of UCG process is proposed to be developed that incorporates many complex behaviors like oxidation, pyrolysis, reduction, subsidence, water influx, cavity growth and so on. The model is essentially a combination of the reactive heat and mass transport together with thermo-mechanical failure behavior. Fig. 2 demonstrates the mathematical modeling strategies that will be adopted for modeling the UCG process.*

A simple one-dimensional process model based on kinetics will be developed initially and will subsequently be validated with lab-scale experimental data.
A detailed flow patterns study (two-dimensional) using CFD will be conducted and the results will be used to develop an equivalent reactor network.
The model will further be extended in three-dimension that can: (a) describe the transition from the early stage to a fully developed gasifier, and (b) predict the areal sweep efficiency. In the model the essential features of the underground gasification process will be captured. Furthermore, the model will address the influence of ash permeability on overall cavity development, as well as the thermo-mechanically induced spalling of both roof rock and coal.]

4. Study of the dynamics of gas-bubble in a flowing liquid environment:

   [***Project summary****: The work will be focused on the behaviour of bubble formation in a gas-liquid co-flowing system in which a gas is injected in from a submerged orifice under constant flow rate and the co-flowing outer fluid is the flowing liquid in a concentric cylindrical tube. Here fluid will be considered to be viscous. CLSVOF will be applied. Detached bubble volume, frequency related phenomena will be investigated in detail*.]

5. Numerical study of water droplet mobility in serpentine gas flow channel of PEM fuel cell:

   [***Project summary****: Modeling and simulation of water droplet movement in serpentine gas flow channels can help manufacturers to design the bipolar plate with precise dimension to get higher current density as well as higher efficiency of the PEM fuel cell. This, eventually, can lead to more valuable design of bipolar plate, resulting in an ultimate benefit to the industry. The aim of the proposed research is to develop a rigorous mathematical model for effective simulation of mobility of water droplet within gas flow channels of PEM fuel cell. The major objectives targeted to be achieved out of this proposed project are: simulation of the water droplet movement using the commercial CFD software FLUENT, study on affect of different dimensions and/ or geometry of channels, detail study on effects of surface wettability and inlet air velocity and simulating a real life water/air flow in gas flow channel of PEM fuel cell having considerable industrial applications in automobile section as well as domestic applications*.]

6. Computational modeling of fluid flow and heat transfer around fixed and/or moving objects:

   [***Project summary****: The objective of the project work is to carry out comprehensive computational study and develop a generalized numerical framework for the general fluid flow and heat transfer over single and/or multiple fixed and/or moving objects. The evolved data will be of immense importance for the designer for indigenous product development in the related fields*.]

7. Modelling Fish Locomotion in Turbulent Vortices:

   [***Project summary****: The objective of the work is to propose a novel mechanism, by which swimming fishes can use environmental vortices, which is likely to reduce the cost of locomotion. In a turbulent environment, use of the Karman gait allows the body of a fish to act as a self-correcting hydrofoil. Design of foil propulsors for biomimetic autonomous underwater vehicles could benefit from the reduced power costs associated with the self adjusting motions of the Karman gait. Demonstrating a link between vortex interaction and reduced muscle activity may also provide insight into the dynamics and evolution of aggregation in moving animals. In addition, a precise understanding of the interactions between muscle activity, fin movement and vortices across species promises to aid in the design and implementation of fish passageways.]*

8. Rheo Pressure Die casting of JIS ADC 12 Aluminium alloy:

   [***Project summary****: The objective of the project is the Rheo Pressure Die Casting of JIS ADC12 Aluminium Alloy for automobile application. The specific role of the Co-PI here is to perform numerical simulation of the die casting process using CFD technique. The simulation is required to understand the transport of semi-solid material within the mould cavity. According to the simulation results the die will be designed and subsequently the component will be manufactured*.]

9. Autonomous underwater robotics (UnWar):

[***Project summary***: *The aim of this nationally important mega project is to develop indigenous unmanned underwater vehicle for deep sea surveillance. The 12th five year plan project is an extension of the 11th plan where the vehicle was developed to operate at a depth of 150m. Now in the 12th plan it is proposed to increase the depth from 150m to 500m and subsequently to 1000m. The project is under the supervision of the Robotics laboratory of CSIR-CMERI. The specific role is to perform numerical simulation to understand the hydrodynamics of the underwater vehicle in operation. The main objective is to compute the hydrodynamic forces acting on the surfaces of the vehicle and provide the appropriate design for optimal hydrodynamic coefficients that would produce the maximum propulsive thrust.*]

TRAINING/COURSES TAUGHT:

Theoretical Courses:

Sl. No.	Course Title	Level	No. of times taught
1.	Thermodynamics	UG	3
2.	Fluid Mechanics	UG	2
3.	Engineering Mechanics	UG	2
4.	Fluid Machinery	UG	1
5.	Heat Transfer	UG	1
6.	Material Science	UG	2
7.	Computational Heat Transfer	UG	1

Practical/Sessional courses:

Sl. No.	Course Title	Level	No. of times taught
1.	Engineering Graphics	UG	2
2.	Workshop practical	UG	2
3.	Fluid Mechanics and Fluid Machinery Laboratory	UG	1
4.	Thermal Engineering Laboratory	UG	2

TRAINING/COURSES ATTENDED:

Sl. No.	Course Title	Organized by	Period	No. of Participants
1.	Finite Volume Methods for Solving Transport Equations	CMERI, Durgapur	May 31-June 3, 2010	30
2.	General course on CFD	Director, CMERI	Nov. 2010-Dec. 2010	30
3.	Soft elastic films and nanotechnology	CMERI, Durgapur	Dec. 08-09, 2010	20

WORKSHOP/CONFERENCE/INVITED SEMINAR/SHORT-TERM COURSE ORGANIZED:

Sl. No.	Type	Invited speaker/ course coordinator/ Instructor	Topic/Subject	Date
1.	Seminar Invited	Prof. Sakir Amiroudine & Prof. Michael Bouderon, University of Bordeaux 1, France	Thermovibrational instabilities in supercritical fluids	Dec 17-20, 2011
2.	Short Course	Dr. SK. Chakraborty, Ex Sct, CSIR-NAL	Introduction to CFD with aerospace applications	May 14-18, 2012
3.	Seminar Invited	Prof. R.P. Chhabra, IIT Kanpur	Effect of shape and orientation on laminar natural convection in power-law fluids	Feb 13, 2013

COLLABORATION:

National:

- ✓ **Department of Mechanical Engineering, IIT Madras** (Prof. S.P. Das): Joint research works in thermal-fluid science are undergoing
- ✓ **Department of Mechanical Engineering, IIT Kharagpur** (Prof. Suman Chakraborty): One project is undergoing entitled "Modelling fish locomotion in turbulent vortices" sponsored by CSIR-CMERI. Additionally, other research works in the field of micro and nano-fluidics are also undergoing
- ✓ **Department of Mechanical Engineering, NIT Durgapur** (Prof. A.N. Mullick, Prof. N.B. Hui, Prof. Santanu Pramanik): One joint PhD is completed and one undergoing
- ✓ **Department of Mechanical Engineering, IIT Guwahati** (Prof. Gautam Biswas): Research works in thermal-fluid science are undergoing
- ✓ **Department of Aerospace Engineering and Applied Mechanics, IIEST Shibpur** (Prof. Pabitra Halder): Research works in thermal-fluid science are undergoing
- ✓ **Department of Mechanical Engineering, IIT Dhanbad**: Collaborative works initiated
- ✓ **CSIR-CIMFR, Dhanbad** (Dr. A.K. Singh): A collaborative project of high national importance on "Underground coal gasification technology in India" is undergoing. The project is under the 12th Five Year Plan of India funded by CSIR.

International:

- ✓ **Laboratoire TREFLE UMR CNRS, Esplanade des Arts et Métiers, University of Bordeaux, France** (Prof. Sakir Amiroudine): Joint research works in thermal-fluid science are undergoing

- ✓ **Department of Mechanical Engineering, University of Maryland, USA** (Prof. Siddhartha Das): Joint works in the field of micro/nano scale transport have been initiated

ADMINISTRATIVE/ INSTITUTE SUPPORT WORK:

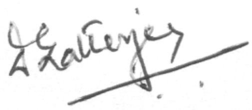
Sl. No.	Section/office/ Institute level committee	From	To	Position held	Responsibilities
1.	Simulation & Modeling Laboratory	17.10.11	Till Date	Head	• Creation of infrastructural facilities for the Department, • Earning high value projects, • Monitoring research and academic activities • Other administrative works
2.	Works Committee	15.04.13	14.04.14	Safety Officer	• Look after safety issues
3.	Payment Committee	23.06.14	Till Date	Member	• Check final payment files and recommend for payment
4.	Bank Settlement Committee	03.03.15	Till Date	Member	Settlement of pending bank reconciliation items and to ensure settlement of all debits raised, credit afforded by bank
5.	Foundation Day Celebration Committee	Sep. 2012	-	Chairman, Hospitality	• Providing hospitality to the special guests and other guests invited to the foundation day celebration of the institute

REFERENCES:

1. **Dr. Suman Chakraborty**
   Professor, Department of Mechanical Engineering,
   IIT Kharagpur-721302, India
   Ph: +91-3222-282990 (O)
   +91-3222-282991 (H)
   Fax: +91-3222-282278
   Email: suman@mech.iitkgp.ernet.in

2. **Dr. Gautam Biswas**
   Professor, Department of Mechanical Engineering,
   IIT Kanpur-721302, India
   Presently on deputation at CSIR-CMERI, Durgapur-713209, India
   Ph: +91-343-2546749/ 2546401 (O)
   +91-343-2564030/6510430 (H)
   Fax: +91-343-2546745
   Email: gtm@iitk.ac.in, gtm@cmeri.res.in

3. **Dr. Sakir Amiroudine**
   Professor, Laboratoire TREFLE UMR CNRS,
   8508 Esplanade des Arts et Métiers,
   33405 Talence Cedex France
   Ph: +33(0)556845403 (O)
   +33(0)626243716 (M)
   Email: sakir.amiroudine@trefle.u-bordeaux.fr



Date: 30.09.2015
Place: Durgapur, India

(Dr. Dipankar Chatterjee)