

Numerical Analysis of Coal Combustion in Circulating Fluidized Bed

K.M.Pandey and Ravindra Kumar

Abstract—The combustion of coal is used in circulating fluidized bed combustion system. This work is concerned about gas–solid two-phase mixtures flowing upwards through the fast beds. The type of information resulting from various ways of analyzing the pressure variations and the effect of superficial velocity on pressure in fluidized beds are discussed. This paper presents the standard $k-\epsilon$ and two-phase turbulence model used to describe the gas–solids flow in a CFB. The analysis of temperature and density in coal combustion is done by discrete phase model (DPM) and non pre mixed combustion in species model through the PDF file creation in DPM model. The flow temperature and density in the system were modeled by solving the continuity and momentum equations of the gas flow by using the CFD analysis. The fluidizing velocity has been taken as 4 m/s and the effects of operating parameters on mole fraction, mean density and mean temperature are discussed in this paper. It is observed that the mean density of minimum and maximum mean mixture fraction along the scaled variance is significant.

Index Terms—Two-phase flow, discrete phase model, turbulence model, non-premixed model

I. INTRODUCTION

Circulating fluidized bed combustors (CFBCs) are considered in some respects to be an improvement over the traditional methods of coal combustion. Operation of industrial CFBCs has confirmed many advantages including fuel flexibility, broad turn-down ratio, high combustion efficiency, low NO_x emissions and high sulphur capture efficiency. These characteristics assure increasing commercialization of CFBC in power generation applications. Although CFBC technology is becoming more common, there are some significant uncertainties in predicting performance in large-scale systems. Technical knowledge about design and operation of CFBC is widely available, but little has been done in the field of mathematical modeling and analysis of combustion in CFBCs. Fluidized beds suspend solid fuels on upward-blowing jets of air during the combustion process. The result is a turbulent mixing of gas and solids. When a fluidized bed is operated above the terminal velocity of the particles, they are carried out of the bed. The fluidization process begins when a bed of inert material (usually sand),

which is a solid granular particle, is suspended by a flow of air or gas (air). This flow is injected into the combustion chamber from the bottom and from the side. Combustion systems for solid fuels FBC reduces the amount of sulfur emitted in the form of SO_x emissions.

II. LITERATURE REVIEW

K. Myöhänen and V. Tanskanen [1] focused on the fluidized bed systems can be simulated by computational fluid dynamics (CFD), but the complicated processes set limits for the modeling. This paper presents the current status of CFD modeling capabilities and modeling examples. The fluid dynamics of solids have a large effect on various phenomena, such as the mixing of different reactants and heat transfer. Thus, as a starting point for a more comprehensive model, first the fluid dynamics of the solids would have to be modeled correctly. The fluid dynamics of the solid phase have a major effect on the different phenomena in fluidized bed systems. For reliable modeling of overall process, including the chemistry and heat transfer, the fluid dynamics of solids would have to be modeled correctly. M. Benes [2] worked on We describe behavior of the air-coal mixture using Navier-Stokes equations for gas and particle phases, accompanied by a turbulence model. The undergoing chemical reactions are described by Arrhenian kinetics (reaction rate proportional to $\exp\left(\frac{-E}{RT}\right)$; where T is temperature). He also considered the heat transfer via conduction and radiation. The system of PDEs is discretized using the Finite Volume Method and an Advection Upstream Splitting Method as the Riemann solver. The resulting ODEs are solved using the 4th order Runge-Kutta method. Sample simulation results for typical power production levels are presented. He has developed a mathematical model, which approximates the combustion process in an industrial furnace, while being affordable from the computational complexity standpoint. As an outlook to the future, mainly the following improvement possibilities are being considered:

1. Further refining of the coal combustion model.
2. Evaluation and enhancements of the NO_x generation model.
3. Thorough evaluation of the 3D model.

Timo Hyppänen [3] presented on Predicting the performance of large scale circulating fluidized bed boilers requires reliable and efficient modelling tools. The furnace should be modelled three-dimensionally to simulate the non-uniform combustion process. He presents semi-empirical steady-state combustion code for three-dimensional modeling of a CFB furnace. The model is a combination of zero-dimensional fractional mass balance

model and three-dimensional modeling of gaseous and solid species. The complexity of the CFB process prevents practical calculations of the whole furnace process with fundamentals-orientated CFD codes. Modeling is possible with simplified, semi-empirical models, but they require reliable and extensive measurement data to validate the empirical correlations.

A. Williams and M. Pourkashanian [4] worked on the some of the key issues currently being debated regarding the combustion of coal and of some biomass materials. It attempts to summarize the present approaches toward the quantification of the fundamental processes of solid fuel combustion for use in computer models. Some aspects of the various chemical and physical processes are included, such as heating-up of particles, devolatilization, and subsequent char formation. Of particular interest is the prediction of char properties, such as composition, surface areas, and morphology, since these impacts on char combustion.

The available models for devolatilization, char formation, and heterogeneous oxidation are considered for application to both coal and biomass, and a description in terms of the fundamental molecular processes of a coal particle is explored. Char combustion models, including global intrinsic reaction rates, are outlined. Konstantin P. Filipov [5] presented on Mathematical model describing two-phase flow in CFB in framework of external model ash was elaborated. It takes into account the all main processes in high-velocity fluidized bed including processes of the coal combustion and the attrition of ash. On base of this model was developed numerical code and carried out some calculations of transient and stationary performances of CFB. Z. Guangbo [6] focused on a steady state model of a coal-fired circulating fluidized-bed boiler, based on hydrodynamics, heat transfer and combustion, is presented. This model predicts the flue gas temperature, the chemical gas species (O_2 , H_2O , CO , CO_2 and SO_2) and char concentration distributions in both the axial and radial locations along the furnace including the bottom and upper portion. The model was validated against experimental data generated in a 35 t/h commercial boiler with low circulation ratio.

AfsinGungor [7] worked on a dynamic two dimensional model is developed considering the hydrodynamic behavior of CFB. In the modeling, the CFB riser is analyzed in two regions: The bottom zone in turbulent fluidization regime is modeled in detail as two-phase flow which is subdivided into a solid-free bubble phase and a solid-laden emulsion phase. In the upper zone core-annulus solids flow structure is established. Simulation model takes into account the axial and radial distribution of voidage, velocity and pressure drop for gas and solid phase, and solids volume fraction and particle size distribution for solid phase. The model results are compared with and validated against atmospheric cold bed CFB units' experimental data given in the literature.

Hideya Nakamura [8] presented on modeling of particle fluidization behaviors in a rotating fluidized bed (RFB) was conducted. The proposed numerical model was based on a DEM (Discrete Element Method)-CFD (Computational Fluid Dynamics) coupling model. Fluid motion was calculated two dimensionally by solving the local averaged

basic equations. Particle motion was calculated two-dimensionally by the DEM. Calculation of fluid motion by the CFD and particle motions by the DEM were simultaneously conducted in the present model. Geldart group B particles (diameter and particle density were 0.5 mm and 918 kg/m³, respectively) were used for both calculation and experiment.

AboozarHadavand [9] presented on a mathematical model of the circulating fluidized bed combustion system based on mass and energy conservation equations were successfully extracted. Using these correlations, a state space dynamical model oriented to bed temperature has been obtained based on subspace method. Bed temperature, which influences boiler overall efficiency and the rate of pollutants emission, is one of the most significant parameters in the operation of these types of systems. Having dynamic and parametric uncertainties in the model, a robust control algorithm based on linear matrix inequalities (LMI) have been applied to control the bed temperature by input parameters, i.e. coal feed rate and fluidization velocity. Circulating fluidized bed (CFB) combustion systems are increasingly used as superior coal burning systems in power generation due to their higher efficiency and lower emissions.

L.X. Kong and P.D. Hodgson[10] worked on To improve the understanding of the heat transfer mechanism and to find a reliable and simple heat-transfer model, the gas flow and heat transfer between fluidized beds and the surfaces of an immersed object is numerically simulated based on a double particle-layer and porous medium model. The velocity field and temperature distribution of the gas and particles are analyzed during the heat transfer process. The results provide sufficient information to improve the understanding of heat transfer processes near the immersed surface. J.C.S.C. Bastos focused on [11] Radial solids velocity profiles were computed on seven axial levels in the riser of a high-flux circulating fluidized bed (HFCFB) using a two-phase 3-D computational fluid dynamics model. The computed solids velocities were compared with experimental data on a riser with an internal diameter of 76 mm and a height of 10 m, at a high solids flux of 300kgm⁻²s⁻¹ and a superficial velocity of 8ms⁻¹. Several hundreds of experimental and numerical studies on CFBs have been carried out at low fluxes of less than 200kgm⁻²s⁻¹, whereas only a few limited useful studies have dealt with high solids flux.

III. CFB MODEL CREATION AND GRID GENERATION IN GAMBIT

For the furnace as explained in a two dimensional model is created in GAMBIT 2.3.16. The 2D view of the furnace after modeling is as shown in Fig.1

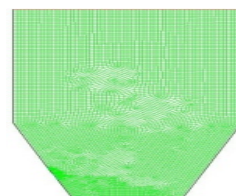


Fig.1. View of the model in 2D after modeling in GAMBIT 2.3.16

GAMBIT 2.3.16 was used for making 2D furnace geometry with width of 3.2m from the lower part and height 15m. Coarse mesh size of 0.01m was taken in order to have 9365 cells (18952 faces) and 9588 nodes for the whole geometry. It was used in order to have better accuracy. But using mesh results in 9365 cells (18952 faces), which requires smaller time steps, more number of iterations per time step and 4 times more calculation per iteration for the solution to converge.

IV. SELECTION OF MODELS FOR ANALYSIS

FLUENT 6.2.16 was used for analysis. 2D segregated 1st order implicit steady solver is used. (The segregated solver must be used for multiphase calculations). Standard k-ε model with standard wall functions were used. The model constants are tabulated as:

TABLE I: MODEL CONSTANTS USED FOR ANALYSIS AS GIVEN BELOW

Cmu	0.09
C1-Epsilon	1.44
Boundary	Condition
Inlet1	Fluidized Velocity inlet
Inlet2	Secondary Velocity inlet
Particle flow	Mass flow rate
outlet	Pressure-outlet
C2-Epsilon	1.92
C3-Epsilon	1.3
TKE Prandtl Number	1
TDR Prandtl Number	1.3
Energy Prandtl Number	0.85

The non-premixed combustion model is used for combustion and in DPM (discrete phase model) single type injection system is used for the solid fuel combustion.

V. BOUNDARY CONDITIONS

The boundary conditions are as equally important as the selection of the proper mathematical model. Initially, solid particle velocity was set at in minimum fluidization and gas velocity was assumed to have the same value everywhere in the bed. The temperature of the primary and secondary air was also set to 503k and 473k respectively. At the inlet, all velocities of all phases were specified. At the outlet, the pressure was assumed to be Atmospheric pressure. The gas tangential normal velocities on the wall were set to zero (no slip condition). The following boundary equation was applied for the tangential velocity of Particle on the wall.

VI. GOVERNING EQUATIONS

When the non-adiabatic non-premixed combustion model is enabled, FLUENT solves the total enthalpy form of the energy equation:

$$\frac{\partial}{\partial t}(\rho H) + \nabla \cdot (\vec{v} H \rho) = \nabla \cdot \left(\frac{k_t}{c_p} \nabla H \right) + S_h \quad (1)$$

Under the assumption that the Lewis number (Le) = 1,

the conduction and species diffusion terms combine to give the first term on the right-hand side of the above equation while the contribution from viscous dissipation appears in the non-conservative form as the second term. The total enthalpy H is defined as

$$H = \sum_j Y_j H_j \quad (2)$$

where Y_j is the mass fraction of species j and

$$H_j = \int_{T_{ref,j}}^T c_{p,j} dT + h_j^0(T_{ref,j}) \quad (3)$$

$h_j^0(T_{ref,j})$ is the formation enthalpy of species j at the reference temperature $T_{ref,j}$.

VII. STANDARD K - ε MODEL

Another class of turbulence models is the two-equation models. The simplest one is the standard $k - \epsilon$ model, which is proposed by *Launder and Spalding*. It is widely used in turbulence simulations because of its general applicability, robustness and economy. The two transport equations for the kinetic energy and dissipation rate are solved to form a characteristic scale for both turbulent velocity and length. These scales represent the turbulent viscosity.

The modeled transport equations for k and ϵ in the realizable $k - \epsilon$ model are

$$\frac{\partial}{\partial t}(\rho k) + \frac{\partial}{\partial x_j}(\rho k u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k + G_b - \rho \epsilon - Y_M + S_k \quad (4)$$

$$\frac{\partial}{\partial t}(\rho \epsilon) + \frac{\partial}{\partial x_j}(\rho \epsilon u_j) = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\epsilon} \right) \frac{\partial \epsilon}{\partial x_j} \right] + \rho C_1 S_\epsilon - \rho C_2 \frac{\epsilon^2}{k + \sqrt{\epsilon \nu}} + C_{1\epsilon} \frac{\epsilon}{k} C_{3\epsilon} G_b + S_\epsilon \quad (5)$$

$$C_1 = \max \left[0.43, \frac{\eta}{\eta + 5} \right], \eta = S_\epsilon \frac{k}{\epsilon}, S = \sqrt{2 S_{ij} S_{ij}} \quad (6)$$

In these equations, G_k represents the generation of turbulence kinetic energy due to the mean velocity gradients. G_b is the generation of turbulence kinetic energy due to buoyancy. Y_M represents the contribution of the fluctuating dilatation in compressible turbulence to the overall dissipation rate. C_2 and $C_{1\epsilon}$ are constants. σ_k and σ_ϵ are the turbulent Prandtl numbers for k and ϵ , respectively. S_k and S_ϵ are user-defined source terms.

VIII. RESULTS AND DISCUSSION MEAN TEMPERATURE PROFILE IN PDF TABLE CREATION

When the non-premixed combustion model in species model is used in coal combustion then table will have to create a PDF table in which, will have to give the fuel composition (ultimate analysis) such as carbon, hydrogen, oxygen, sulfur, nitrogen etc in the form of mole or mass fraction. The mass or mole fraction of fuel composition should be 1. In PDF table creation is self consider the other

values such as mean mixture of fraction points, number of mixture fraction variance points and maximum number of species in fluent software. Fig.2 shows the Mean temperatures with respect to mean mixture fraction in x-y direction. Maximum of Mean Temperature (K) is 2.351846×10^3 and occurs at Mean Mixture Fraction = 1.078125×10^{-1} . Fig.3 shows the Mean temperatures with respect to mean mixture fraction in x-y-z direction. Maximum value of Mean Temperature (K) is 2.351846×10^3 and occurs at mean Mixture Fraction = 1.078125×10^{-1} and Scaled Variance = 0.000000×10^0 . Minimum value of Mean Temperature (K) is 3.000000×10^2 and occurs at Mean Mixture Fraction = 1.000000×10^0 and Scaled Variance = 0.000000×10^0 .

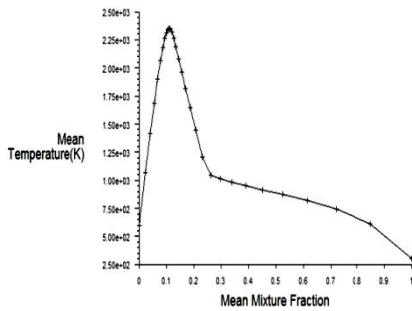


Fig. 2 Mean temperatures along with mean mixture fraction.

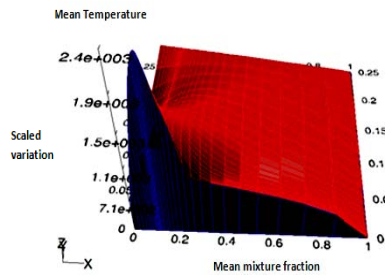


Fig. 3 Mean temperatures along with mean mixture fraction in x-y-z direction.

IX. MEAN DENSITY PROFILE IN PDF TABLE CREATION

The density, or local mass per unit volume, is an intensive property that may vary from point to point. After PDF Table creation in non premixed combustion model can be checked the combustion values though graphically in fluent software such as temperature, density, mole fractions etc. Fig. 4 shows the maximum of Mean Density (kg/m^3) is 7.663372×10^{-1} and occurs at Mean Mixture Fraction = 8.473195×10^{-1} . Fig.5 shows the Mean Density along with Scaled heat loss/Gain in which Maximum of Mean Density (kg/m^3) is 1.212401×10^0 and occurs at Scaled Heat Loss/Gain = -1.000000×10^0 . Fig. 6 shows the mean density along with mean mixture fraction in 3D surface(x-y-z) direction. Maximum value of Mean Density (kg/m^3) is 7.663372×10^{-1} and occurs at Mean Mixture Fraction = 8.473195×10^{-1} and Scaled Variance = 0.000000×10^0 . Minimum value of Mean Density (kg/m^3) is 1.517449×10^{-1} and occurs at Mean Mixture Fraction = 1.125915×10^{-1} and Scaled Variance = 0.000000×10^0 .

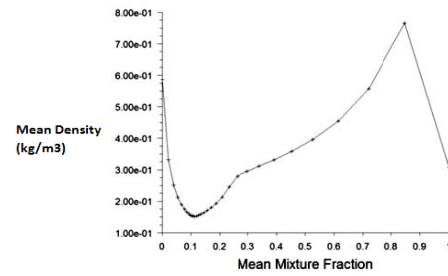


Fig.4 Mean density along with mean mixture fraction.

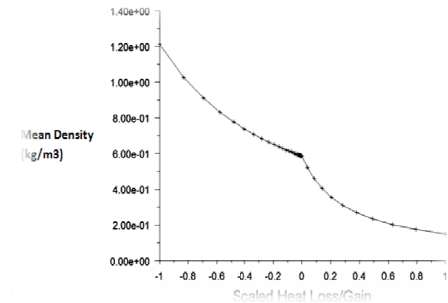


Fig. 5. Mean density along with scaled heat loss/gain.

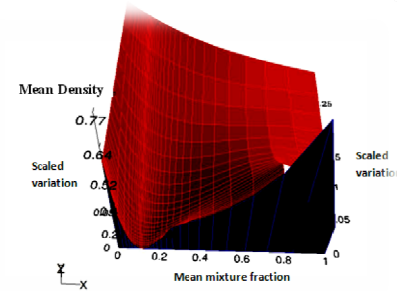


Fig. 6. Mean density along the mean mixture fraction in x-y-z direction..

X. CONCLUSIONS

The following conclusions are drawn on the basis of above mentioned results and discussions.

It is observed that the mean mixture fraction of minimum and maximum mean temperature is significant.

1. It is observed that the scaled variance for different temperature at different mean fraction is constant.
2. Maximum of mean density (kg/m^3) 0.7 occurs at mean mixture fraction 0.8 but the mean density along with scaled heat loss/gain in which maximum of mean density (kg/m^3) is 1.21 occurs at scaled heat loss/gain = -1.0.
3. It is observed that the mean density of minimum and maximum mean mixture fraction along the scaled variance is significant.

NOTATION

ρ = Density
 $\vec{V}$ = Velocity vector
 p = Static pressure
 $\vec{\tau}$ = Stress tensor
 $\rho \vec{g}$ = Gravitational body force
 P = Pressure

μ_{eff} = Effective viscosity
 K = Turbulent kinetic energy
 ε = Dissipation rate of turbulent kinetic energy
 T = Temperature
 g = Acceleration due to gravity
 k_{eff} = Effective conductivity
 k_t = Turbulent thermal conductivity
 μ = molecular viscosity
 I = unit tensor
 h = Sensible enthalpy

XI. ACKNOWLEDGEMENT

The authors acknowledge the financial help provided by aicte from the project aicte: 8023/rid/boiii/ncp(21) 2007-2008 .the project id at iit guwahati is me/p/usd/4.

REFERENCES

- [1] K. Myöhänen and V. Tanskanen, CFD Modelling Of Fluidized Bed Systems, Lappeenranta University of Technology, FI-53851 Lappeenranta, Finland.
- [2] M. Benes, Model of Turbulent Coal Combustion, Kuju Training Center, Oita, Japan, September 15-18, 2005, pp. 95-105.
- [3] T. Hyppänen, Modelling Of Circulating Fluidized Bed Combustion with a SemiEmpirical Three-Dimensional Model, Lappeenranta University of Technology, FI-53851 Lappeenranta, Finland.
- [4] A. Williams and M. Pourkashanian, The Combustion of Coal and Some Other Solid Fuels. 2000; 28: 2141-2162.
- [5] P. Konstantin Filipov Numerical Modeling Of Coal Combustion Processes In Ecological Clear Circulating Fluidized Bed Boiler Units, Novosibirsk State Technical University 630092.
- [6] Z. Guangbo, A coal combustion model for circulating fluidized bed boilers. Fuel 2000; 79: 165-172.
- [7] A. Gungor, Hydrodynamic modeling of a circulating fluidized bed, Powder Technology 2007; 172: 1-13.
- [8] H. Nakamura, Numerical modeling of particle fluidization behavior in a rotating fluidized bed, Powder Technology 2007; 171: 106-117.
- [9] A. Hadavand, An innovative bed temperature-oriented modeling and robust control of a circulating fluidized bed combustor, Chemical Engineering Journal 2008; 140: 497-508.
- [10] L. X. Kong and P. D. Hodgson, Computational simulation of gas flow and heat transfer near an immersed object in fluidized beds, Advances in Engineering Software 2007; 38: 826-834.
- [11] J. C.S.C. Bastos, Modelling and simulation of a gas-solids dispersion flow in a high flux circulating fluidized bed (HFCFB) riser, Catalysis Today 2008; 130: 462-470.
- [12] I. Petersen and J. Werther, Three-dimensional modeling of a circulating fluidized bed gasifier for sewage sludge. Chemical Engineering Science 2005; 60: 4469-4484.
- [13] V. Shilapuram, Comparison of macroscopic flow properties obtained by three methods of operation in a liquid-solid circulating fluidized bed, Chemical Engineering and Processing 2009; 48: 259-267.
- [14] K. M. Pandey and A.P Singh, "Recent Advances in Experimental and Numerical Analysis of Combustor Flow Fields in Supersonic Flow Regime" International Journal of chemical Engineering and Application ,Vol.-1, No. 2, August 2010, ISSN- 2010-0221, pp 132-137
- [15] T. C. Jack ornelissen, CFD modelling of a liquid-solid fluidized bed, Chemical Engineering Science 2007; 62: 6334 - 6348.
- [16] K. M. Pandey and A. P. Singh "Comparative Analysis of Jet Co-Flow and Modified Jet Co-Flow Based Injection at Mach 2 With Les Modeling" International Review of Aerospace Engineering, Praise Worthy publications, October 2011 (Vol. 4 N. 5)
- [17] K. M. Pandey and A.P Singh, "CFD Analysis of Conical Nozzle For Mach 3 At Degrees Of Angle Of Divergence With Fluent Software" International Journal of chemical Engineering and Application ,Vol.-1, No. 2, August 2010, ISSN- 2010-0221, pp 179-185.
- [18] X. B. Qi, Friction between gas-solid flow and circulating fluidized bed downer wall, Chemical Engineering Journal 2008; 142: 318-326.
- [19] H. Zhu, Detailed measurements of flow structure inside a dense gas-solids fluidized bed, Powder Technology 2008; 180: 339-349.
- [20] K. M. Pandey and A. P. Singh " Numerical Analysis of Combustor Flow Fields in Supersonic Flow Regime with Spalart-Allmaras And k-ε Turbulence Model" International Journal of Engineering and Technology. June 2011, pp.-208-214.
- [21] Z. Chen, Comparison of fluidized bed flow regimes for steam methane reforming in membrane reactors: A simulation study. Chemical Engineering Science 2009; 64: 3598 - 3613.
- [22] J. Stephen Goidich, Foster Wheeler Compact CFB Boilers for Utility Scale, Foster Wheeler Energy International, Perryville Corporate Park Clinton, New Jersey, USA 08809-4000.
- [23] A. Almuttahir, Computational fluid dynamics of high density circulating fluidized bed riser: Study of modeling parameters. Powder Technology 2008; 185: 11-23.
- [24] E. U. Hartge, CFD-simulation of a circulating fluidized bed riser, Particology 2009; 7: 283-296.
- [25] K. M. Pandey and A. P. Singh " Transient Analysis of Supersonic Combustion with Jet Co-Flow With Finite Rate Chemistry Model" International Review of Aerospace Engineering, Praise Worthy publications, Vol-4, No.-2, April 2011,.



K.M. Pandey obtained PhD in Mechanical Engineering in 1994 from IIT Kanpur. He has published and presented 200 papers in International & National Conferences and Journals. Currently he is working as Professor of the Mechanical Engineering Department, National Institute of Technology, Silchar, Assam, India..He also served the department in the capacity of head from July 07 to 13 July 2010. He has also worked as faculty consultant in Colombo Plan Staff College, Manila, Philippines as seconded faculty from Government of India. His research interest areas are the following: Combustion, High Speed Flows, Technical Education, Fuzzy Logic and Neural Networks , Heat Transfer, Internal Combustion Engines, Human Resource Management, Gas Dynamics and Numerical Simulations in CFD area from Commercial Software's. (e-mail: kmpandey2001@yahoo.com)