downstream of the critical nozzle exit on the flow field close to the nozzle throat were clarified numerically. The results obtained from the steady computations were in close agreement with the previous experimental data. In order to simulate the effects of back-pressure fluctuations on the critical nozzle flow, an excited pressure oscillation was assumed downstream of the exit of the critical nozzle. The computed results showed that, for Reynolds numbers below Re = 3740, the unsteady effect of the pressure fluctuations can propagate upstream of the nozzle throat and it has effects on fluctuations in mass flow rate through the critical nozzle, even in chocked-flow conditions. In cases of the pressure disturbance with large amplitude for Re = 7470, the pressure fluctuations can propagate upstream of the nozzle throat. However, it is found that the mass flowrate does not fluctuate with time.

T-2G-3. NUMERICAL INSTABILITY ANALYSIS OF THE LATTICE BOLTZMANN EQUATION METHODS USING DIFFERENT SCHEMES

A. R. RAHMATI, *Isfahan University of Technology, Iran*, M. ASHRAFIZAADEH, *Isfahan University of Technology, Iran*, E. SHIRANI, *Isfahan University of Technology, Iran,* The lattice Boltzmann equation (LBE) method has been recently developed into an alternative effective tool to simulate fluid flows. Although it has significant advantages over the conventional CFD methods, there are still some restrictions in the utilization of LBE models. One of the shortcomings is that the physical boundary conditions for macroscopic variables cannot be implemented directly since the dependent variable in the LBE model is the density distribution function. Furthermore, for the Single-Relaxation–Time Lattice Boltzmann Equation (SRT-LBE) model when the Reynolds number is large, the relaxation parameter approaches to the stability margin if the number of mesh points is not large enough. The instability problems may be compounded in three-dimensional flows when physics may not be adequately resolved owing to computational constraints. Much progress has been made in this direction in recent years and several approaches have been proposed to increase numerical stability. Some of such approaches are Entropic Lattice Boltzmann Equation (ELBE) method, Multi-Relaxation-Time Lattice Boltzmann Equation (MRT-LBE) method and Fractional Volumetric Lattice Boltzmann Equation (FV-LBE) method. In the present work, numerical stability of the ELBE, MRT-LBE, FV-LBE models and a combination of the two last methods, i.e., FV-MRT-LBE model are compared to that of the SRT-LBE method for the simulation of the lid-driven cavity flow at different values of the Reynolds number, ranging from 1000 to 10000, on a 257× 257 grid. The FV-MRT-LBE model is a new approach that is presented in this work and its results are compared with the results obtained by other methods. Results show that the stability and accuracy of all methods are comparable with that of the SRT-LBE method at lower Reynolds numbers. However, as the Reynolds number is increased, the stability of all methods turns out to be better than that of the SRT-LBE method. Furthermore, the MRT-LBE method produces more stable and slightly more accurate results compared to ELBE, FV-LBE, and FV-MRT-LBE methods, especially at higher Reynolds numbers.

T-2G-4. ABILITY OF MIXING TWO IMMISCIBLE LIQUIDS IN A KENICS STATIC MIXER

J. Y. C. LEONG, *Monash University, Malaysia*, C. F. THAN, *Monash University, Malaysia*, Y. W. OOI, *Monash University, Malaysia,* The ability of the Kenics static mixer, a commercially available mixer that does not involve the use of mechanical agitation, in mixing of two immiscible liquids has been investigated. The model object is a length of pipe containing the Kenics static mixer elements each resembling a short helix. The two modeled phases were palm oil triglyceride in the continuous phase and methanol in the discrete phase. This process mimics the pre-mixing process before the transesterification process which takes place in the biodiesel production. The flow field was simulated numerically using the commercially available Computational Fluid Dynamics software package, Fluent. In the preprocessing phase, the modeling of the model object incorporated both tetrahedral and hexahedral meshing schemes to discretize the model geometry. Solving the flow fields involved choosing the appropriate models. These included choosing between the various laminar and turbulent models, an appropriate multiphase model as well as an appropriate drop breakage model. Postprocessing was concerned with extracting relevant information from the flow field to evaluate the performance of the Kenics static mixer and involved assessing its effectiveness in mixing the two fluids by statistical means and estimating the pressure drop across the mixer. Simulated data predicted that the Kenics mixer with an RL (successive elements were installed with an alternate twist) configuration is the most effective static mixer. On the other hand, the Kenics RR (successive elements were installed in the same direction) mixer is more efficient as it achieves mixing with less power requirement. A better understanding of the Kenics static mixer will enable its incorporation into the production cycles that necessitates the dispersion of two immiscible liquids to affect the kinetics of reactions as seen in the transesterification process in biodiesel production.

16:00 ~ 17:20 (Room 101)

Flows with Heat Transfer (I)

Session Chair : Prof. J. S. Park, Halla Univ/Korea

T-3A-1. HEAT TRANSFER ENHANCEMENT OF RECTANGULAR RIBS WITH CONSTANT HEAT FLUX LOCATED IN THE FLOOR OF A 3D TURBULENT DUCT FLOW

E. ESMAEILZADEH, A. ALAMGHOLILOU, H. MIRZAIE, A. KHOSHNEVIS, *Department of Mechanical Engineering, the University of Tabriz, Tabriz, 51666-14766, Iran,* In this paper numerical investigation of hydrodynamic and forced convection heat transfer in a rectangular horizontal duct has been undertaken. Heat sources are cross rectangular ribs with small aspect ratio and uniform heat flux under turbulent regime. The purpose of this study is to apply a passive method for increasing rate of heat transfer from the ribs. Geometry and the physic of the problem are roughly similar to cooling of electronic boards. Therefore three rectangular ribs established along the width of the channel with specified distance on the floor. Between ribs some vortexes are appearing which in general act like a heat traps and reduce the heat transfer rate. This thermal resistance should be neutralized by applying heat transfer enhancement methods. Establishing holes between the ribs because of that the interior pressure is much less than environment is an advantageous method which causes distortion of vortexes and finally augmentations of heat transfer by producing a secondary flow without any outsource energy. This method classifies as passive methods. Numerical simulation for assumed geometry is performed by solving governing equation in finite volume with Phoenics software. Obtained results were compared with available experimental results of literature and indicate good agreement. Comparison between plain case and passive case shows effectiveness (PEC) is related to geometry parameters significantly specially to number of holes and their arrangements. In this investigation 9 arrangements was analyzed so results compared and discussed completely.

T-3A-2. STUDY OF THE EFFECT OF PARTICLE SIZE ON THE HEAT TRANSFER IN A FLUIDIZED BED HEAT EXCHANGER

Y. D. JUN, *Kongju National University, Korea*, K. B. LEE, *Kongju National University, Korea*, S. Z. ISLAM, *Kongju National University, Korea*, S. B. KO, *Kongju National University, Korea*, and M. F. KADER, *Kongju National University, Korea,* Heat recovery from flue gas from industrial furnaces, boilers and incinerators for better use of energy resources is a nation-wide concern in Korea. To overcome the fouling of fly ash on the heat transfer surface and erosion and periodical cleaning which are the major drawbacks in conventional heat exchangers for flue gas heat recovery, a single riser no-distributor-fluidized-bed (NDFB) heat exchanger is devised. Compared to the existing ones, the present heat exchanger system is featured in the particle fluidization method which does not depend on conventionally used baffle plate with holes and the multiple down comer tubes for heat extraction from the heated particles. The heat transfer performance and pressure drop, effect of suspension density and particle size is studied for this no-distributor-fluidized-bed (NDFB) heat exchanger system. It was observed that the effect of particle size on the heat transfer is more significant for smaller particles and larger suspension densities.

T-3A-3. RE-CIRCULATION BEHAVIOUR IN THE FLOW FIELD OF A LOW ASPECT RATIO DUMP COMBUSTOR

N. P. YADAV, *IIT Kanpur, India*, A. KUSHARI, *IIT Kanpur, India,* This paper reports an experimental investigation of the recirculation behaviour inside a low aspect ratio dump combustor. The length of the combustor studied was less than the reattachment length for the separated flow. The exit of the combustor is tapered which supports the flow reversal from the exit section. The recirculation behaviour in the combustor is evaluated from the cold flow visualization and pressure measurement studies. This recirculation was induced by the geometry of the combustor, therefore, it was an unforced recirculation. In this study, a dissimilar thickening of the recirculating flow was observed inside the combustor that happened due to the flow reversal from the exit section. The recirculation and flow reversal from the tapered exit section encourage a cyclic behaviour in the vortex. The findings of the visual study were corroborated by measuring the wall pressure distribution and the results were found to be in a good qualitative agreement with each other. The findings of this study can have applications