M-2B-3. MIXING ANALYSIS IN A THREE-DIMENSIONAL SERPENTINE MICROCHANNEL

M. A. ANSARI, K. Y. KIM, *Inha University, Korea,* The mixing of the fluids in microdevices is challenge as the flow in these microdevices are predominantly laminar. The small dimension of these microdevices puts a limit on the size of the micromixer. The mixing of the fluid in these devices is due to diffusion which is very slow process. Serpentine channel micromixer has been reported to be effective in mixing the fluids. In the present work mixing and pressure analysis has been performed with two design parameters of the three dimensional serpentine channel. A computational fluid dynamics technique has been applied to understand mixing phenomenon in three dimensional serpentine microchannel. ANSYS ICEM 11.0 was used to make a hexahedral grid for the full model. Numerical analyses of flows in the three-dimensional serpentine channel have been performed using the commercial code ANSYS CFX-11.0. Mixing in the channel has been directly analyzed by using two working fluids water and ethanol. A parameter, mixing index based on the variance of the mass fraction on a cross-section normal to the flow direction was used to evaluate mixing. Dependence of mixing on the various design parameters has been analyzed at four Reynolds numbers i.e., 1, 10, 35 and 70. The mixing index has been reported along channel length at three values of h/w i.e. 0.5, 1.0 and 1.5 for Re=35 and d/w=1.0 and four values of d/w, 0.03, 0.5, 1.0, and 1.5 at a Reynolds number of 10. The pressure drop is remarkably higher for h/w=0.5 as compared to h/w=1.0 or 1.5. The rate of decrease of the pressure drop is almost the same for the three h/w ratios. Pressure drop is not significantly affected by change to the h/w ratio at Re=1 and 10, but becomes pronounced at Re=35 and 70. The pressure drop reaches the highest value at the lowest value of h/w and the highest Reynolds number.

M-2B-4. MIXING AND INTERFACE GEOMETRY IN A SMALL-SCALE CHANNEL

S. NORO, K. KOKUNAI, M. SHIGETA, S. IZAWA, Y. FUKUNISHI, *Tohoku University, Japan,* The experimental and the numerical investigations are carried out to find methods to enhance mixing of two liquids in a small-scale channel. Two liquids, blue and clear water, are alternately injected into the channel. The mixing effect is examined using a simple straight channel and a channel with a cavity. The three-dimensionality of the flow field is examined by comparing the results of the experiment, two-dimensional and three-dimensional computations. The pitfall of focusing only to the interface length is discussed. In the experiment, the flow channel is inscribed into an acrylic resin plate surface using a milling machine. The size of the channels is 100 mm long, 1 mm wide and 0.4 mm deep. The mixing level is evaluated by photographs taken by a digital camera attached to a microscope. In the two-dimensional and three-dimensional simulations, first the flow field is computed using a finite difference method, and afterwards the two-dimensional diffusion-convection equation is solved using the velocity data. The mixing levels in the experiments are compared to those in the simulations. It is found that the mixing levels in the experiments and the simulations show a similar tendency, though the values of the mixing level are different especially between the experiment and two-dimensional simulation. The reason for the discrepancy is explained using the results from the three-dimensional simulation. It is shown that the discrepancy mostly derives from the fact that the flow field is not two-dimensional. The basic aspects of the interface between the two liquids are studied using the two-dimensional numerical simulation. As a result, it is found that the increase in interface length does not directly correspond to the enhancement of mixing level in the downstream region. It is shown that, in order to accomplish rapid mixing, it is important to stretch the interface where the concentration gradient is high.

14:50 ~ 16:10 (Room103)

Turbulence and Instability Control

Session Chair : Prof. C. Lee, Yonsei Univ/Korea

M-2C-1. TIME DEPENDENCE OF TURBULENT DRAG REDUCTION EFFICIENCY FOR POLYISOBUTYLENE-KEROSENE SYSTEM

Ki-Ho LEE, *Department of Polymer Science and Engineering, Inha University & School of Computational Sciences, Korea Institute for Advanced Study, Korea,* Hyoung Jin CHOI, *Department of Polymer Science and Engineering, Inha University, Korea*, Jae Ho KIM, *Korea Institute of Energy Research, Korea,* Phenomenon of turbulent drag reduction has been one of the most interesting issues for reducing energy consumption. A flexible polymeric chain molecule can be a candidate for the agents bringing drag reduction, in which the skin frictional drag of a turbulent flow remarkably decreases from that of turbulent solvent due to the addition of the elastic component. When this event is applied to the transportation of crude oils through long pipelines, many rewards can be obtained in environmental and economical aspects. Even though the turbulent drag reduction in a pipe flow is one of the major applications, a rotating disk system is used in this study to examine time dependence of the drag reduction and mechanical degradation of polymer with an explicit extent of the initial drag reduction, which is hard to be taken in a pipe flow due to the induction period of uniform dispersion of polymer in the flow. Turbulent flow of kerosene, which is driven by a rotating disk in a closed chamber, shows significant drag reduction when tiny amount of polyisobutylene (PIB) was added in the medium. The efficiency of turbulent drag reduction for this PIB-kerosene solution was examined to be decreased with time due to mechanical degradation of the polymer chains induced by severe force of turbulence. The change in drag reduction was fitted with two model equations of Brostow and our model as a function of time. The result shows the drag reduction efficiency fitted well with small deviations for the most of PIB-kerosene solutions of various concentrations at different temperatures. This study is expected to contribute to find optimal conditions, such as temperature and concentration, for obtaining highly effective drag reduction in actual applications.

M-2C-2. LINEAR STABILITY ANALYSIS OF PLANE POISEUILLE FLOW: A PRIMITIVE VARIABLE FORMULATION

H. S. PANDA, *Defence R & D Organization, Chandipur, India,* S. Ghosh MOULIC, *Department of Mechanical Engineering, Indian Institute of Technology Kharagpur, India,* In this paper the results of linear stability analysis of plane Poiseuille flow in primitive variable formulation have been presented. Special to this paper are; a novel boundary condition has been devised and Chebyshev spectral collocation technique is used for the solution of the resulting differential eigenvalue problem for enhanced accuracy. The results compared well with the bench mark results of Orszag [5], who has presented first 32 eigenvalues arranged in descending order of the imaginary part of the complex wave speed and classified into two categories vize symmetric mode and antisymmetric mode and Mack [4], who extended the symmetric mode further. In this study, we also extended the number of eigenvalues up 43 and classified and arranged the eigenvalues following Orszag [5], in addition two new members are inserted into the previously published eigenvalue spectrum. The accuracy of the new boundary condition has also been compared with the normal momentum boundary condition for pressure, which is an obvious boundary condition for the pressure disturbance amplitude. The critical eigenvalue obtained using the proposed boundary condition is 5772.221805. We have formulated a simple diagnostic tool 'β-filtering technique' to identify the good eigenvalues and to eliminate the spurious eigenvalues from the complete eigenvalue spectrum obtained using QZ algorithm. The governing equations are the constant property Navier-Stokes equations. The mean flow whose stability is under study is a steady fully developed flow under a constant pressure gradient. Since primitive variable formulation is adopted in this study, boundary conditions for pressure are required. Conventionally the same is obtained by collocating the linearised disturbance amplitude equation of normal velocity at walls, however in this study we proposed the collocation of normal mode continuity equation at walls, which satisfies the validity of continuity equation throughout the solution domain. To identify the acceptable eigenmodes we devised the *β-filtering* technique, where we sort the β_{qz} in descending order. After that we plot the sorted β_{qz} and identify $n_{\beta*}$ corresponding to β_{qz} where there is sudden drop of the β_{qz} to the neighborhood of zero. Then the eigenvalues corresponding to all β_{qz}s smaller than $\beta_{n\beta*}$ are rejected and the accepted eigenvalues are obtained by dividing the remaining α_{qz} with corresponding β_{qz}. In addition we have also checked the absolute values of residuals ($AX-\lambda BX$) for these accepted eigenvalues and found that in all cases they are well within 10^{-8}. The performance of proposed continuity boundary condition with that of normal momentum boundary conditions is compared. We observed that collocation of continuity boundary condition at wall gives complete spectrum of eigenvalues as obtained via solution of Orr-Sommerfeld equation, however normal momentum boundary condition produces an eigenvalue between mode 1 and 2. The proposed *'β-filtering technique'* to diagnose the spurious eigenvalues has been found to qualify the residual criterion suggested in Dongarra [1]. The critical Reynolds number of the flow is found to be 5772.221815 and the corresponding wave speed is 0.246000126. To demonstrate the accuracy of our computed critical parameters, we have obtained the neutral stability curve in the neighborhood of critical Reynolds number. Finally we suggested the use of continuity boundary condition for primitive variable formulation.