10dB. Compressed dry air is stored in a high pressure tank that has a capacity of $5m^3$, and is supplied to the plenum chamber, in which a honeycomb system reduces flow turbulence. A convergent-divergent nozzle with a design Mach number of 2.0 is installed in the end wall of plenum chamber. The nozzle has a throat height of 9.6mm, an exit height (*D*) 16.2mm, and a width 30mm. A Schlieren optical system is employed to visualize the qualitative structures of supersonic jet. Acoustic measurements are made using a condenser microphone that has a diameter of 6mm. The microphone is located at a 96° from the jet direction, and the radial distance of 34*D* (550mm) from the exit of the nozzle. The acoustic signals are analyzed by using a FFT analyzer. A FFT analysis provides the noise spectra, and providing the spectral data in the range from 0 to 40 kHz, with a frequency band width of 25 Hz. The temperature in the plenum chamber is measured by using a thermocouple (approximately 288K), and it maintains constant at room temperature during test. The pressure is measured by a pressure transducer flush mounted on the top wall of the plenum chamber and nozzle wall. The results obtained show that generation of transonic resonance is closely related to the pressure oscillations in the nozzle, and the large area of Mach stem increases the amplitude of transonic tone.

W-1E-2. COMPUTATIONAL ANALYSIS OF JET IN SUPERSONIC REGIME WITH VARIABLE AMBIENT CONDITIONS

Ajmal BAIG, S. BILAL and S. ZAHIR, *National Engineering & Scientific Commission, Islamabad. Pakistan,* Computational study of the behaviour of the supersonic jet in supersonic regime has been done for fully-expanded and under-expanded jets and compared with the still ambience. The supersonic jet is produced with a convergent-divergent nozzle. Initially the centre-line pressure distribution with an ambience of still air is calculated for fully and under-expanded jets and then the pressure distributions for fully and under-expanded jets are calculated when ambient flow is supersonic. The centerline pressure distribution in still air shows that the oscillations in fully expanded jet are very fast as compared to the under-expanded jet. These pressure oscillations are reduced in the supersonic regimes as a result of supersonic pressure relief effect. The length of shock cells for underexpanded jet is greater than the full-expanded jet both in still air and supersonic freestream. The computational analysis showed that the jet pressure adjust to the ambient pressure very quickly in the supersonic regime as compared to the still ambience. The problem is solved using CFD software based on the finite control volume technique. Air is taken as working fluid and solved as ideal gas. Viscosity is calculated using Sutherland law. For 2D axisymmetric geometry, the continuity and momentum conservation equations in axial and radial direction are solved. The energy equation is also solved. Density based solver with implicit formulation and 2nd order upwind flow discritization is used. Two equations k epsilon turbulence model with standard wall functions is used. Symmetry boundary condition is applied along the axis of symmetry for nozzle flow simulation. At the inlet of nozzle, reservoir stagnation conditions are employed. Pressure farfield condition is used for the outer boundaries. At a very high Reynolds number, the boundary layer developing at the nozzle wall is very thin, and its influence on the mean core of the jet is very weak.

W-1E-3. DETAILED ANALYSIS OF THRUST PLUME AND SATELLITE BASE REGION INTERACTION

J. G. KIM, *KAIST, Korea*, O. J. KWON, *KAIST, Korea*, M. J. YU, *KARI, Korea*, K. H. LEE, *KARI, Korea* and S. K. KIM, *KARI, Korea,* The interaction between thrust plume and satellite base region for the Korea multipurpose satellite-Ⅲ configuration was investigated by using direct simulate Monte Carlo calculations. For the accurate simulation of H2 collisions and rotation-translation transition, a variable soft-sphere model and a recent rotational relaxation model of N2 and H2 were used. For the investigation of the interaction between thrust plume and base region, the number density distribution for each species, translational and rotational temperature distributions, heat flux, and pressure were examined by DSMC calculations. It was found that most of the surface properties are affected by H2 collisions and a strong non-equilibrium state is observed on the S-band antenna and at the base region. It was demonstrated that an accurate model is needed to simulate H2 collisions and the rotation-translation transition. The results by the present calculation are more accurate than previous DSMC calculations because more accurate models were used in simulating elastic and inelastic collisions in the present study.

W-1E-4. EXPERIMENTAL INVESTIGATION ON HYSTERESIS PHENOMENA OF MACH DISK IN UNDER-EXPANDED AXISYMMETRIC JET

S. MATSUO, *Saga University, Japan*, T. HIGASHI, *Saga University, Japan*, T. SETOGUCHI, *Saga University, Japan*, H. D. KIM, *Andong National University, Korea,* When the high-pressure gas is exhausted to atmosphere from the nozzle exit, the under-expanded supersonic jet with the Mach disk is formed at a specific condition. The jet structure has been known as a fundamental phenomenon of the supersonic fluid mechanics. The jet is very important for some industrial devices and there exists many papers. These papers are on the steady jet as the ratio of reservoir stagnation pressure to back pressure is fixed. In two-dimensional under-expanded supersonic jet, it is reported that the hysteresis phenomenon for the reflection type of shock wave in the jet is occurred under the quasi-steady flow in the recent studies and the transitional pressure ratio between regular reflection and Mach reflection in the jet is affected by this phenomenon. The purpose of this study is to clarify the hysteresis phenomena for the reflection type of shock wave at the under-expanded axi-symmetric jet experimentally and to discuss the relationship between hysteresis phenomenon and rate of the change of pressure ratio with time. Furthermore, the effect of Mach number at the nozzle exit on hysteresis loop was investigated for two kinds of nozzle. The design Mach numbers of Nozzles A and B are M_e= 1.0 and 1.5, respectively. The Mach disk diameter and the distance from nozzle exit to Mach disk were obtained from shadowgraph pictures. As a result, in the under-expanded axi-symmetric jet, hysteresis phenomena for the configuration of shock wave were investigated experimentally. The phenomena were confirmed in the supersonic axi-symmetric jet at the present experimental condition. Furthermore, the range of hysteresis loop extended with an increase of Mach number at the nozzle exit and this phenomenon had the same characteristics as that in the transition between regular and Mach reflection of shock waves in two-dimensional flow field.

09:00 ~ 10:20 (Room106)

General Fluid Dynamics

Session Chair : Prof. M. Alam JNCASR/India

W-1F-1. THERMOMAGNETIC CONVECTION IN A MAGNETIC NANOFLUID LAYER

I. S. SHIVAKUMARA, *UGC-Centre for Advanced Studies in Fluid Mechanics, Department of Mathematics, Bangalore University, India,* Magnetic nanofluids are commercially manufactured stable colloidal liquids formed by suspending fine magnetic monodomain nanoparticles in non-conducting carrier liquids like water, heptane, kerosene, etc. Thermal convection in magnetic nanofluids has become a topic of current technical importance because magnetic forces can be used to create circulation of coolant in small passages where natural convection is either absent or ineffective. In the present paper, the linear stability of an initially quiescent magnetic nanofluid layer under the influence of an external magnetic field is investigated. The lower rigid-ferromagnetic boundary is heated by a constant heat flux, while at the upper rigid-ferromagnetic boundary a general thermal condition which encompasses fixed temperature and constant heat flux as particular cases is invoked. The resulting eigenvalue problem is solved numerically using the Galerkin technique and it is observed that the instability is always onsets into steady convection. Realizing this fact, closed form analytical solutions are also obtained using regular perturbation technique for constant heat flux thermal boundary conditions with wave number a as a perturbation parameter. It is found that the results obtained from these two techniques are in excellent agreement suggesting the analytical solution obtained for the constant heat flux thermal conditions is exact. It is also found that an increase in the magnetic number M_1, and the nonlinearity of fluid magnetization M_3, as well as decrease in the Biot number Bi, is to hasten the onset of thermomagnetic convection. To the contrary, the nonlinearity of fluid magnetization is found to have no effect on the threshold values for the onset of thermomagnetic convective instability in the case of constant heat flux thermal boundary conditions. Besides, comparison of results with earlier work is also made under the limiting case and good agreement is found.

W-1F-2. FRICTION FACTOR IN A V-SHAPED OPEN CHANNEL

Mirali MOHAMMADI, *Department of Civil Engineering,Urmia University, Urmia,* Alireza MOGHADDAMNIA, *Faculty of Natural Resources, University of Zabol, Iran,* The experimental study on the distribution of friction factor, f, of a V-shaped channel is examined as it occurs in sewers and culverts. Several series of experiments were conducted for measuring velocity and boundary shear stress. It may be seen that the Darcy-Weisbach friction factor, f, is more sensitive than the other resistance coefficients such as the Manning n. The contour plots of 2D isovels shows that the isovels are parallel to the channel boundary in a region close to the bed, and almost symmetric about the centerline. A strong surface phenomenon is observable throughout the isovels plots. Consequently, it may be said that the presence of secondary currents and cross sectional shape effects should be responsible. The transverse distribution of depth-averaged velocity and boundary shear stress across the channel section for various flow conditions are presented. Local boundary shear stress around wetted perimeter, τ_b, and depth-averaged velocity, U_d, data are used to evaluate the local friction factor, f. The results illustrate that the friction factor is shown to be a function of flow depth as well as being dependent upon the Froude number, Fr. Therefore, the local friction factor does not remain constant across the channel. The results of the present study indicate that the global friction factor decreases as flow discharge increases and channel bed slope getting steeper in the experiments that included subcritical and supercritical flows. However, it can be seen that the effect of walls is so high at low Froude numbers. The results indicate that the perturbations in the distribution of local friction factor are quite considerable in steeper channels as Froude number increases. It may also be occurred for the distributions of boundary shear stress as well as velocity results at the same flow conditions.

W-1F-3. NUMERICAL ANALYSIS ABOUT THE PRESSURE DROP INSIDE THE METAL FOAM AS HEAT EXCHANGER

P. H. KIM, M. H. JIN, Y. J. KIM, *Gyeongsang National University, Korea,* H. M. JEONG, H. S. CHUNG, *Gyeongsang National University, Institute of Marine Industry, Korea,* Recently, the use of high porosity metal foams have spread to include applications, such as aircraft wing structures for the aerospace industry, catalytic surfaces for chemical reactions, core structures for high strength panels, and containment matrices and burn rate enhancers for solid propellants. Due to for several decades, heat transfer enhancement has been the focal point of interest for high-performance thermal systems. On this basis, Porous medium was considered in the present study for the heat transfer enhancement. This was attributed to its high surface area to volume ratio as well as intensive flow mixing by tortuous flow passages. Especially, aluminum foam recently introduced has low pressure drop and high effective thermal conductivity for its high porosity and inter-connected solid ligaments. In this study, copper foam was used to fill inside the pipe instead of aluminum foam. It is because copper material has higher heat conductivity than aluminum material. Furthermore, the copper foam has excellent characteristic in the structural strength. Therefore, the copper foam can be a promising candidate for the heat transfer enhancement toward the development of the high performance heat exchanger. But when the air or water flow through in the porous medium, it is occurred the pressure drop between inlet and outlet. So in the present study investigated numerical analysis result about the pressure drop in the porous medium before apply to heat exchanger. In the numerical analysis, the thickness of the solid inside the porous medium region was varied 0.2 mm to 0.4 mm. And then the numerical analysis result were compared the pressure drop in the same unit cell (0.5 mm x 0.5 mm x 0.5 mm). To make the analysis model, it was assumed the 14-sided tetrakaidecahedron cell which has long been considered the optimal packing cell first proposed by the Lord Kelvin in 1887. And then the simulation is carried out using by STAR-CCM+ which is commercial software. The simulation result can be showed quantified pressure drop by solid effect in the porous medium.

W-1F-4. VIBRATION SOURCE OF PRIMARY LNG PUMP

H. Y. KIM, *KOGAS, Korea*, C. M. KIM, *KOGAS, Korea*, Y. S. HONG, *KOGAS, Korea,* H. S. LIM, *KOGAS, Korea,* H. S. SEO, *KOGAS, Korea,* KOGAS(Korea Gas Corporation) has a severe vibration from primary LNG(Liquefied Natural Gas) pumps installed in-ground type LNG tanks. First, the physical relation of operating pumps and pipeline network was observed by using Newton method as corrective flowrate, which is a solving model of pipeline network. The vibration source could be found from the network analysis. The pump vibration occurred from the decrease of LCV(Liquid Control Valve) opening and a pump in the lowest level of tank. The LCV controls the flow rate of LNG, which is controlled with the level of LNG in a BOG(Boil-Off Gas) recondenser. An LNG 3-D flow analysis of pump was carried out to investigate the occurrence of cavitation below a specified level. In case of the level of 10 m and the pump flowrate of 260 m^3/h, the average void fraction was 0.22. When the level and pump flowrate were 5 m and 260 m^3/h, the fraction was 0.45. The LNG level showed the increase of cavitation in the pump. Especially, the case of the former also showed the decrease of void fraction when the pressure resistance of the network is decreased. Consequently, one vibration source of primary LNG pump proved to be the decrease of LCV, which brought about the pressure fluctuation from the latter part of it. Also, FCV(Flow Control Valve), which controls the flow rate of LNG at main header, could make an similar impact on LNG pump such as LCV. This work gave light upon the vibration source of the pumps under the complex LNG pipeline network and another vibration source proved to be the external impact force of LNG filling line. KOGAS will establish a counter plan to dampen the vibration occurred from the pumps in LNG receiving terminal.

09:00-10:20 (Room107-108)
Computational Fluid Dynamics (VII)
Session Chair : Dr. C. Kim, KARI/Korea

W-1G-1. LES OF TRANSITONAL FLOW PAST AIRFOIL SD7003 USING IMMERSED BOUNDARY METHOD

X. L. YANG, G. W. HE, X. ZHANG, *Institute of Mechanics, CAS, China,* Large eddy simulation (LES) combined with immersed boundary method is used to study the flow past an airfoil SD7003. The Reynolds number based on inflow velocity and chord length is 6×10^4. This flow is characterized by a laminar separation bubble and transition to turbulence. The immersed boundary method used in present case is a delta function based approach in which delta function is used in interpolation and spreading processes. In large eddy simulation, the Smagorinsky SGS model is used. And the Van Driest damping function is applied in the near wall region. The friction velocity is calculated from Lagrange force. The spatial discretization scheme is based on a second order finite volume formulation. The third order Runge-Kutta scheme is used for terms treated explicitly and second order Crank-Nicholson is used for terms treated implicitly in temporal discretization. In present simulation, the time averaged drag and lift coefficients are in good agreement with other people's results with comparable mesh resolution. The laminar separation bubble is also captured. The separation and transition point is under-predicted compared with other author's numerical and experiment results, while the reattachment point is comparable with other people's results. One reason for this phenomenon is that the mesh resolution in the leading part of the airfoil is not adequate. The separation happens early, and so the transition. The other reason is the 4-point delta function used in present simulation, which can smear the results in the near wall region. Another reason is that the density of Lagrange point in the leading part is not enough. he present study shows the capability of the hybrid LES/IB method to simulate the transitional flows. A work is in progress on mesh refinement, compact Delta function and dense Lagrange points, in order to investigate the reasons for the under-predictions of separation and transition.

W-1G-2. PERFORMANCE PREDICTION OF WATER MIST FIRE SUPPRESSION MECHANISM FOR SHIPBOARD ENCLOSURE USING CFD METHODS

K. Y. BAE, H. T. CHUNG, I. S. JEONG, *Gyeongsang National University, Korea*, Y. S. HAN, B. I. CHOI, *KIMM, Korea,* The present study aims ultimately at the performance prediction of the water mist nozzle for fire suppression systems of ship buildings. To apply the water mist system inside the living area of cruise ships the fire scenario must follow IMO rules. In the present study, we carried out the numerical analysis to test four types of nozzle in the passenger cabin. The purpose is to find the optimal design parameters by spray performances of water mist and to increase the fire suppression effect. In order to simulate on interaction between fire flame and water mist according to properties of water mist, this study uses the FDS (Fire Dynamics Simulator, version 4.07) program that is developed for fire and extinguishment by NIST (National Institute of Standards and Technology). The numerical geometry is adopted W×L×H=4×3×2.4 m^3 with a dead zone of W×L×H=1.22×1.1×2.4 m^3. The water mist nozzle is placed in ceiling center of 2.3m height from the floor. Six mattresses and four cushions are placed in the simulation space. To compare with fire suppression effect by nozzle performance, the thermocouples are installed near ceiling. In the case of MISS 4 nozzles, the inflammables were disappeared after 500 seconds of mist injection, which means that the inflammables were burnt totally by fire plume. In case of large particle that reach to fire source by the high kinetic energy and the long travel distance, the inflammables on right upper part are not disappeared even though the fire diffusion is suppressed by water mist. In the case of MISS 4 nozzle for mean ceiling temperature, it is decreased during 10 seconds after injection of water mist, but is showed distribution that decrease rising again. HRRPUA showed large effect for fire suppression as droplet size is large, whereas mean ceiling temperature is distributed lower as droplet size is smaller. The reason of these phenomena could be explained that the small droplet does not reach to fire source by buoyancy effect occurred during free combustion and heat transfer occurs fast near ceiling by interaction between water mist and fire plume. KIMM nozzle gave lower temperature distributions than MISS 4 did because of issuing larger flow rate. The effect of fire suppression is much larger at the large droplet than at the small droplet because the large droplet has the higher kinetic energy and the