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 longer travel distance. The mean ceiling temperature is distributed lower as droplet size is smaller because the small droplet does not reach to fire source by the buoyancy effect occurred during free combustion and heat transfer occurs fast near ceiling by interaction between water mist and fire plume. Among the spray nozzle used in the present simulation, K-3 by KIMM is showed the best effect of fire suppression because it has the largest heat transfer area by supplying the flow rate more than about 6 times than others.

W-1G-3. DEVELOPMENT AND TESTING OF AN AERODYNAMIC OPTIMISATION CODE USING DISCRETE ADJOINT METHOD

Manoj T. NAIR, *National Aerospace Laboratories, India,* A method for performing aerodynamic optimisation for aerofoils and wings is presented. The design variables used for optimisation are the parameters representing the geometry. The compressible Euler equations for flow are the governing equations. The discrete adjoint method is used to compute the sensitivity of the objective function with respect to the design parameters. The Complex Taylor's Series Expansion method is used to numerically compute the residual Jacobian required for the discrete adjoint method. The Euler equations are solved using the implicit matrix-free LU-SSOR method. The convective terms are discretized using the van-Leer flux vector splitting approach. The discrete adjoint equations are developed based on this discretization. The adjoint equations are solved using the scalar diagonal LU-SSOR method after introducing a pseudo time term. The constraints are applied by using the penalty function method. The code is parallelized using OPENMP directives. The developed code has been tested for aerofoil and wing optimisation at transonic speeds.

W-1G-4. EFFECT OF MERGING ANGLE ON MIXING OF HYDROGEN AND AIR BEHIND A THICKNESS BASE

Mohammad ALI, S. ISLAM, *BUET, Bangladesh,* A. K. M. SADRUL ISLAM, *IUT, Bangladesh,* To study the effect of merging angle on the mixing of hydrogen and air, the characteristics of the flow field, and flame holding capability of a supersonic combustor a numerical investigation has been performed. The merging angles of two streams are varied from $10^\circ \sim 50^\circ$. The flow fields are investigated by solving Two-Dimensional Navier-Stokes equations. A zero-equation algebraic turbulence model proposed by Baldwin and Lomax has been used to calculate the eddy viscosity coefficient. To delineate the purely fluid dynamic effects, the flow has been treated as non-reacting. It can be found that recirculations and penetration of hydrogen play an important role to enhance mixing. The area of recirculation decreases with the increase of merging angle but mixing efficiency increases. The recirculation regions and several shocks such as expansion shock, recompression shock and reattachment shock in the flow field are evident.

10:40-12:00 (Room101)

Supersonic and Hypersonic Flows (I)

Session Chair : Prof. J. Kurian, IIT Madras/India

W-2A-1. NUMERICAL STUDY ON SUPERSONIC IMPINGING JET FROM COLD SPRAY NOZZLE

H. KATANODA, M. FUKUHARA, *Kagoshima University, Japan,* The cold spray is an innovative spray coating method which was patented in 1994 in the USA. It uses a supersonic gas flow to spray solid particles to make a coating on the substrate. In the paper, the over-expanded impinging jet from a cold spray nozzle, as well as the velocity of the particle accelerated by the gas flow, was studied by numerical simulation. The effects of the stagnation pressure and temperature upstream of the throat on the flow field and the particle velocity were investigated. The nozzle has a throat diameter of 2.0mm and the exit diameter of 5.0mm. The distance from the nozzle exit to the impinging wall was set at 10mm. The nitrogen gas was used as a process gas. The ranges of the stagnation pressure and temperature upstream of the nozzle throat are set as 2.0 - 3.0 MPa and 300 - 675 K, respectively. In this simulation, the spherical copper particle with 15 m in diameter was selected as the spray particle. The particle velocity was calculated based on the one-way coupling method along the center line of the gas flow. The numerical results of the gas flow shows that there exists minor effect of the stagnation conditions on the Mach number distribution in the nozzle. On the other hand, the gas velocity in the nozzle increases by increasing the stagnation temperature. The calculated particle velocities show that the shock wave structure at the nozzle exit has a negligible effect on the velocity distributions of the 15 m copper particle. In addition to that, the stagnation temperature has a larger effect on the particle velocity than the stagnation pressure. From the present numerical simulation it is concluded that increasing the stagnation temperature is more effective than the stagnation pressure to increase the impact velocity of the particle.

W-2A-2. UNSTEDY BEHAVIOR OF SHOCK WAVES AROUND A CIRCULAR ARC BLADE WITH BUMP IN TRANSONIC MOIST AIR FLOW

S. MATUO, A. B. M. T. HASAN, I. TOMOHIRO, T. SETOGUCHI, *Saga University, Japan,* H. D. KIM, *Andong National University, Korea,* The transonic flow over the airfoil is characterized by shock waves standing on the surface. In this case, the interaction between the shock wave and boundary layer becomes complex because the shock wave imposes an adverse pressure gradient on the boundary layer. As a result, the self-excited shock wave oscillation occurs in the flow fields. The unsteady phenomena in the transonic flow around airfoils are also observed in the flow field of fan, compressor blade, butterfly valves and so on. In the transonic or supersonic flow where vapor is contained in the main flow (moist air), a non-equilibrium condensation process occurs at a supersaturated state. The condensation phenomena coupled with fluid flow is important in many engineering and technical application such as supersonic nozzle, steam turbine, cryogenic turbomachinery, shock tube, transonic wing, helicopter blades and so on. However, the effect of non-equilibrium condensation on the internal flow around the transonic airfoil has not been studied satisfactorily. In the present study, the effect of non-equilibrium condensation in moist air flow on the characteristics of self-excited shock wave oscillation on the circular arc blade with or without the bump was investigated experimentally and numerically. Results obtained are as follows: shock strength in the case of blade with bump becomes weak compared to that without bump model, and amplitude and range of oscillations become small for the case of blade with bump for all blade angles of attack. For a circular arc blade with angle of attack, the distributions of condensate properties are mainly observed on upper region around the blade. Furthermore, the non-equilibrium condensation reduces the frequency and amplitude of the oscillation compared with the case of no condensation for all angles of attack.

W-2A-3. INVESTIGATIONS OF WEAK NORMAL SHOCK WAVE/LAMINAR BOUNDARY LAYER INTERACTIONS IN DUCTS

Y. MIYAZATO, H. YAJI, K. MATSUO, *The University of Kitakyushu, Japan,* The aim of the present research is to elucidate the interaction of a weak normal shock wave formed at just downstream of a nozzle throat with a laminar boundary layer in a two-dimensional duct. The wall contours of the two-dimensional nozzle used in the present experiment are designed by the method of characteristics to be uniform flow at the nozzle exit. The nozzle has heights of 4.4 mm at the throat and of 4.9 mm at the exit with a design Mach number of 1.39. The height at the inlet of the test section (or the nozzle exit) is 4.9 mm, and both the upper and lower walls of the test section are inclined at 0.7 deg to the central axis to stabilize the location of the shock wave in the test section. A static pressure measuring system on a central axis called a through-tube has been devised and the centerline static pressure data in the interaction region have been obtained by the through-tube. Flow visualization by the colour schlieren method with a tricolour filter has been employed to observe the structure of the shock wave interacting with a laminar boundary layer. As conclusions, flow visualization shows that a weak normal shock wave for the freestream Mach number below around 1.3 interacts with a laminar boundary layer to form a shock train. Centerline static pressure distribution has two peaks to show the existence of first and second shocks in a shock train. Also, the static pressure rise across the interaction region decreases as the ratio of the laminar boundary layer thickness to the duct half height increases.

W-2A-4. CHARACTERISTIC OF MULTIPLE PRESSURE WAVE CAUSED BY DISCHARGING OF PULSATING PRESSURE WAVE FROM OPEN-END OF TUBE

T. YASUNOBU, *Kitakyushu College of Technology*, *Japan*, H. KASHIMURA, *Kitakyushu College of Technology*, *Japan*, T. SETOGUCHI, *Saga University*, *Japan,* When the pulsating pressure wave propagated in the tube reaches at open end of a tube, the reflection and emission of pressure wave occurs and the multiple pressure wave is formed. This multiple pressure wave causes the some noise problems as like exhaust tube of automobile so that the mechanism and characteristic of the multiple pressure waves must be more cleared to control the noise problem. Many papers had described for the emission of a single pressure wave from an open end of a tube, as like the shock wave or the compression wave. But, it seems that the emission of the pulsating pressure wave, which is very strong