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 m³/h, the average void fraction was 0.22. When the level and pump flowrate were 5 m and 260 m³/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 m³ with a dead zone of W×L×H=1.22×1.1×2.4 m³. 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