phenomenon of shock wave boundary layer interaction. Various parameters affecting the flow field are free stream Mach number, approach boundary layer, size, shape and bluntness. The overall flow field around a protrusion shows the existence of a complex flow field. This problem had been of interest to researchers as a basic understanding of associated flow field and as well due to its practical application. Experimental studies have been made to obtain the overall flow field around blunt protrusions mounted on a flat surface. The experiments consisted of oil flow visualization, schlieren flow visualization and measurement of static pressures. Effect of various parameters like height and width / diameter and frontal shape, etc. have been obtained. All the experiments are made at free stream Mach number of 2 and Reynolds number of 30 X 10^6 per meter. Longitudinal separation distance measured from oil flow photographs around circular cylinder indicate that there exists a possibility of parameter involving height and diameter and boundary layer, which could be used to non-dimensionalise the longitudinal separation distance. Overall flow field could be captured using Fluent and comparison indicates reasonably good agreement.

T-3D-3. INTEGRATED ANALYSIS OF AN HIGH ANGLE OF ATTACK MANEUVER MISSILE USING FLUID-STRUCTURE INTERACTION

Kyung-Ho NOH, Jae-Woo LEE and Yung-Hwan BYUN, *Department of Aerospace Information Engineering, Konkuk University, Korea,* A missile system, even with its restrictions of size and weight, requires high speed/high maneuverability. To achieve successful missile system development, multidisciplinary analysis and optimization are most needed.
Computational Fluid Dynamics (CFD) and the Finite Element Method (FEM) are used to perform aerodynamics analysis and structure analysis. For the fluid-structure interaction analysis, each technology should be considered as well. The process of aerodynamics-structure coupled analysis can be applied to various integrated analyses from many research fields.
Analysis methods for the individual CFD and FEM analyses are matured and many commercial softwares are currently available. For the aerodynamics-structure coupled analysis, many researches are going on recently and several commercial softwares are ready to use, but the application of the method is limited to the specific or relatively simple geometry. When the configuration geometry is complex or operating conditions are difficult to impose, the meshing and remeshing process between aerodynamic analysis and FEM analysis is not an easy task and commercial softwares have limitations to be applied to the specific problems. Therefore, in this study, the aerodynamics-structure coupled analysis for the conceptual baseline configuration of missile will be investigated through the use of CFD-FEM interaction. The result of the integrated analysis will be compared with rigid geometry of the rocket and the effect of the deformation will be addressed.

T-3D-4. FLOW AND FREQUENCY CHARACTERISTICS OVER DYNAMIC DELTA WINGS

Minglu ZHANG and Zhiyong LU, *Fluid Mechanics Institute, Beijing University of Aeronautics and Astronautics, China,* The test of the flow visualization was completed in the water channel and test of dynamic unsteady pressure measurement was finished in the wind tunnel. The result of flow visualization test shows that in the case of pitching up-stop movement the vortex breakdown position is dependent on the range of incidence at which the wing is subject to pitching up-stop and the reduced frequency k($k=\dot{\alpha}C/2U_\infty$). When incidence range of the pitching up-stop at which the flow regime over the wing is from the attachment flow to the vortex flow corresponding to the static state is set the breakdown vortex would appear over the stopping wing and then the burst point moves downstream and disappear. When incidence range of the pitching-up at which the flow regime over the wing is from the vortex flow to breakdown vortex flow corresponding to the static state the lag phenomena of vortex breakdown position over the dynamic wing could be observed. It is found that the bigger the reduced frequency k is, the larger the lag is. When incidence range of the pitching-up at which the flow regime over the wing is from the breakdown vortex flow to breakdown vortex flow corresponding to the static state the vortex breakdown position moves downstream first and then upstream. Analysis of the pressure signal measured in the wind tunnel shows when the wing is subject to pitching-up the nondimensional spiral wave propagation frequency at nominal incidence in post-breakdown is higher than that at corresponding static state and the bigger the k is, the higher the nondimensional spiral wave propagation frequency is. It means vortex breakdown at the dynamic state is more hysteretic than one at the static state. The same conclusion is found with different sweep delta wings in the wind tunnel.

16:00 ~ 17:20 (Room 105)

Compressible Flows (II)

Session Chair : Prof. S. Matsuo, Saga Univ/Japan

T-3E-1. EFFECTIVE REDUCTION OF CONDENSATION SHOCK STRENGTH IN TWO-PHASE SUPERSONIC FLOW BY SPRAYING WATER DROPLETS AT INLET OF LAVAL NOZZLE

M. R. MAHPEYKAR, E. AMIRIRAD and E. LAKZIAN, *Department of Mechanical Engineering, Ferdowsi University of Mashhad, Iran,* During the course of expansion of steam in turbines, the vapour first supercools and then nucleates to become a two phase mixture. The flow initially is single phase but after Wilson point water droplets are developed and there is a non equilibrium two phase flow. This growing droplets release their latent heat to the flow and this heat addition to the supersonic flow cause a pressure rise called condensation shock. Because of irreversible heat transfer in this region the entropy will increase tremendously. The following study investigates the spraying water droplets at inlet of Laval nozzle and their effects on nucleation rate and condensation shock. According to the results, the nucleation rate is considerably decreased and therefore the condensation shock nearly disappeared. In other words the injecting droplets at the inlet of steam turbine would decrease the thermodynamic losses or improve the turbine efficiency.

T-3E-2. NUMERICAL INVESTIGATION ON CHOKING OF CONVERGING NOZZLE FLOWS

M. YONAMINE, *Kyushu University, Japan*, Y. MIYAZATO, *The University of Kitakyushu, Japan*, K. MATSUO, *The University of Kitakyushu, Japan,* In the one-dimensional isentropic analysis, in a converging nozzle, the flow velocity at the exit can be increased until it becomes sonic but cannot be made supersonic at this cross section. When the flow at the exit is choked, pressure communication between the downstream and upstream flows is broken by sonic flow at the exit. The mass flow rate of the flow then depends only on the upstream stagnation condition. This choking phenomenon is finding application in a flow meter by sonic nozzle or industrial plumbing. Many investigations for choking were performed especially in relation to measuring small mass flow rate of gas. These papers show that the critical pressure ratio to measuring small mass flow rate of gases is different from the ratio derived from one-dimensional isentropic analysis. It is considered that the developed boundary layer on the nozzle wall affect on the flow condition at the nozzle exit. But the effect of the boundary layer on the critical condition of the flow still not obvious and the details have largely remained unknown. The purpose of this study is numerically to examine the phenomenon of choked flow has given consideration to boundary layer thickness that enters an axisymmetric converging nozzle followed by straight pipe. Computational results are compared to experimental data collected by the present authors. To clarify the flow mechanism when choking occurs at the nozzle exit, the general behavior of the choked flow is depicted in static pressure and Mach number contours. As a result, the criterion of the nozzle flow is presented taking the growth of the sonic line extending across the almost entire exit passage into consideration. It is also found that the throat of the nozzle flow exists at upstream of the nozzle exit and the flow is supersonic at the nozzle exit.

T-3E-3. COMPUTATIONAL ANALYSIS OF TRANSIENT FLOWS IN AN EJECTOR-DIFFUSER SYSTEM

G. RAJESH, *Dept. of Mechanical Engineering, College of Engineering Trivandrum, Kerala, India,* H. D. KIM, *Andong National University, Korea,* S. MATSUO, *Dept. of Mechanical Engineering, Saga University, Japan,* T. SETOGUCHI, *Dept. of Mechanical Engineering, Saga University, Japan,* M. DEEPU, *Dept. of Aerospace Engg., Indian Instt. of Space Science & Tech., Trivandrum, India,* The ejector is a simple device which can transport a low-pressure secondary flow by using a high-pressure primary flow. In general, it consists of a primary driving nozzle, a mixing section, and a diffuser. The ejector system entrains the secondary flow through a shear action generated by the primary jet. Until now, a large number of researches have been made to design and evaluate the ejector systems, where it is assumed that the ejector system has an infinite secondary chamber which can supply mass infinitely. However, in almost all of the practical applications, the ejector system has a finite secondary chamber implying steady flow can be possible only after the flow inside ejector has reached an equilibrium state after the starting process. Also it is not clear how the primary jet entrains the secondary flow during the steady mode of the ejector operation, as the secondary chamber will not able to supply the secondary flow indefinitely. To the authors' best knowledge, there are no