fluctuations in the flow field is identified. These observations are important in understanding the response of a flame anchored in a backward-facing-step flow to acoustic-like external perturbations, as in situations of combustion instability.

T-3C-2. APPLICATION OF GATES ON A SETTING DAM AT THE ENTRANCE OF A CANAL

M. AKHYANI, *Department of Marine Science, Science and Research Branch, Islamic Azad University, Tehran, Iran*, S. M. MOSADDAD, *Islamic Azad University, Shoushtar Branch, Iran,* This is an empirical study to justification an ancient river engineering structure with its effect on its neighborhood. When river pass through a city, its effect on land will be more noticeable especially for agricultural and living processes around river basin. Building a setting dam on river gate, necessary water body will be conducted to river route and risk of flood or coastal destruction and erosion in banks of river would sweep away. Number and position of gates appointed on dam is very important hydraulically and hydro dynamically. In Shoushtar, a city lies beside one the major rivers of Iran, called "Karoon" River, "Band Mizzan" is a setting dam and divides Karoon water into two branches, called "Gar_Gar" and "Shotteit" with portion of water bodies in the ratio of 2 to 4 , the gates on it applies well and hydraulically existence of them has been very useful. Existence of setting dam will lead to water storage and flood prevention. The number of gates and their position in "Band Mizzan" were well appointed at hundreds years ago. Some useful and applied results of this study is: Setting dam should be establish perpendicular to river currents direction. Building of setting dam and establishing of the second canal in place of maximum curvature of river is an applied task to decrease the rate of sedimentation and bank erosion. Fastening of bank line of river, building of river coastal walls and deepening of the river basin would be useful to avoid happening of flood around of river. Flood currents and sedimentation in the river with high current velocity could be controlled by setting dam creation. Superposition of surface waves and secondary current wavelengths with dam structure mouths can lead to decrease erosion and make a calm River.

T-3C-3. SEPARATED-LAYER INSTABILITY AND GLOBAL UNSTEADINESS OF LAMINAR SEPARATION BUBBLES

A. V. DOVGAL, *ITAM SB RAS, Russia*, V. V. KOZLOV, *ITAM SB RAS, Russia,* Wind-tunnel data on hydrodynamic instabilities associated with local regions of laminar boundary layer separation are reported. Even at low Reynolds numbers separation bubbles are prone to growth of velocity perturbations resulting in a nonstationary flow pattern. The latter is dominated by several instability features including the transition to turbulence in the separated shear layer and the large-scale unsteadiness of separation bubbles in the form of coherent vortices shedding from the region of reattachment. There are a number of indications that these phenomena may be quite different, that is, related to the convective instability of local mean-velocity profiles and to global dynamics of the entire separation bubble. Global modes of oscillations at laminar flow separation were found through stability analysis in a series of recent studies. The experiments we performed on this subject were as follows. Laminar separation bubbles behind 2D backward-facing steps on a plate surface were examined at low subsonic oncoming-flow velocities through hot-wire measurements. Several experimental regimes differing by the step height comparable with the boundary layer thickness were investigated. Under "quiet" free-stream conditions, the unstable flow in separation bubbles was obtained with transition to turbulence occurring well behind the region of reattachment. As a result, two scales of the natural separated-flow perturbations were distinguished. Those were high-frequency instability waves of the separated shear layer and low-frequency oscillations in the universal frequency range of vortex shedding. The latter originated irrespective of the shear-layer instability and were found as much different from the convective disturbances generated at the step by the oncoming-flow perturbations. To control the global separated-flow unsteadiness, active techniques were tried including continuous suction of the near-wall fluid and small-amplitude periodic forcing of the separation bubble. Both of them appeared as modifiers of the large-scale vortex motion.

T-3C-4. THE EXISTENCE OF A CRITICAL BLOCKAGE FOR CIRCULAR CYLINDERS UNDERGOING VIV AT LOW RE

T. K. PRASANTH, S. MITTAL, *Department of Aerospace Engineering, Indian Institute of Technology Kanpur, India,* Vortex-induced vibration (VIV) of a circular cylinder in the laminar regime has been studied numerically using a stabilized finite element method in two dimensions. The computations are carried out at various mass ratios ($1 \le m^* \le 100$) and blockages ($0.25\% \le B \le 12.5\%$). The effect of mass ratio and blockage on hysteresis phenomenon near the onset of synchronization has been investigated. It is found that for a given m*, hysteresis depends on blockage. The hysteresis loop width decreases with decrease in blockage at all mass ratios. For low mass ratios ($m^* < 11$) the hysteresis loop width decreases with decrease in blockage and completely disappears at a *critical blockage*. The variation of this critical blockage with m* is found to be non-monotonic in nature. However for $m^* > 11$, the response is hysteretic irrespective of the blockage. At higher blockage, large hysteresis loop width is observed. The hysteresis loop width decreases with decrease in blockage and attains a minimum value at very low blockage. There is a critical mass ratio ($m^* = 10.11$) which divides the entire m* range into two. Below $m^* = 10.11$ non-hysteretic response is observed at sufficiently low blockage. Above $m^* = 10.11$ the response will always be hysteretic irrespective of the blockage. Hence the critical blockage is not defined for $m^* > 10.11$. The variation of hysteresis loop width with blockage is found to be similar at various mass ratios for $m^* > 10.11$. The hysteresis loop width for mass ratios, $m^* > 10.11$ can be represented as a function of mass ratio and blockage. This enables us to predict the hysteresis loop width for an experiment once mass ratio and blockage are known. The various hysteresis loop width contours are plotted in the m* v/s blockage plane. The critical blockage curve ($\Delta Re = 0$) divides the m* v/s blockage plane into two. Inside the curve, the response is non-hysteretic and outside the curve, it is hysteretic in nature. This clearly explains the discrepancy in the observation of hysteresis behavior by various researchers. This is the first time the discrepancy in the observation of hysteresis in VIV by various researchers has been explained based on m* and blockage. The hysteresis loop width reported at higher Re is found to match exactly with the value obtained from the present computations. This is despite the fact that most experiments have been conducted at higher Re, beyond the laminar flow regime. It appears that the effect of Re on the blockage v/s m* curve is not significant.

16:00 ~ 17:20 (Room 104)

Aerodynamics (II)

Session Chair : Dr. D. S. Lee, KARI/Korea

T-3D-1. EFFECTS OF JET BLOWING ON THE SIDE FORCE ON FOREBODIES WITH DIFFERENT CROSS SECTION

Youbing ZENG, Zhiyong LU, *Fluid Mechanic Institute, Beijing University of Aeronautics and Astronautics, China,* A flying wing is one of choices for the purpose of reducing Rader Cross Section in the next generation fighter aircraft. A jet blowing at the nose is one of the most popular methods to control the sideforce. The asymmetry of vortices over the forebody and their aerodynamic characters are changed by blowing. The influence of blowing on the side force and yawing moment depends on the cross section shape of the forebody. Three different section shapes of forebodies were chosen to be used in the force measurement experiment. Two blowing methods were adopted in the wind tunnel test which were the blowing normal to the surface through a hole and blowing tangential with the surface through a jet (circumferential pointing angle is able to change the blowing direction of the jet). In the case the maximum side force coefficient of normal blowing on the cone-cylinder is reduced by 43% from Cz=3.5 to 1.8 at incidence ranging from 20°to 65°. It is found that the change of the blowing momentum coefficient has a little influence on the side force coefficient on the cone-cylinder while using normal blowing. In the case of the tangential blowing, test result shows when the circumferential pointing angle of the nozzle is set at 330° with 0.03 momentum coefficient the maximum side force coefficient on the cone-cylinder is reduced from -3.6 to -2.5. And the reverse side force coefficient nearly disappears which is good for yaw control. The experiment of force measurement with the ellipstic section forebody shows when the circumferential pointing angle of the nozzle is set at 300° and 330°the side force coefficient is reduced from Cz=2.5 to 1.0. The experiment result with the chined forebody illustrates that the jet blowing would increase the side force coefficient on the chined forebody slightly. The pointing circumferential angle of 90° and 270° are the optimum blowing pointing position.

T-3D-2. INVESTIGATION OF FLOW FIELD AROUND BLUNT PROTRUSIONS AT SUPERSONIC SPEED

J. K. PRASAD, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India,* S. DAS, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India,* Many of the aerospace vehicles like rockets, missiles, aircraft, etc, have blunt protrusions projecting on the external surface due to various reasons. Blunt fins are also being adopted or thought to generate the possible control forces for future aerospace vehicles. At supersonic speed, the shock wave generated by the blunt protrusion, interacts with the approach boundary layer and leads to a complex flow

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_∞). 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