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 reports on the transient characteristics of the ejector systems at start-up and none of the works to date discloses the detailed flow process until the secondary chamber flow reaches an equilibrium state and what happens after that. The objective of the present study is to investigate the transient flow processes of an ejector-diffuser system. Also, the present study is planned to identify the operating range of ejector-diffuser systems where the steady flow assumption can be applied without uncertainty. The results obtained show that the flow through the ejector attains a dynamic equilibrium state after a particular time depending on the secondary chamber volume. At the pressure equilibrium state, a re-circulation zone appears in the vicinity of the primary nozzle exit. Due to this re-circulation zone, continuous mass entrainment into the primary jet prevails even when there is no flow from the secondary chamber. This is the one and only condition for the two conflicting phenomena to occur simultaneously, i.e., the finite mass flow from the secondary chamber and infinite mass entrainment into the primary jet. The present analysis thus permits to identify what is happening inside the ejector and how the flow field behaves during the starting process of such ejector systems. Further works to investigate the influence of the secondary chamber volume on the time to achieve the dynamic equilibrium of pressure are in progress. It is expected that the secondary chamber will have a deterministic effect on the recirculation phenomenon by which the infinite entrainment is possible in the ejector system.

T-3E-4. A STUDY ON GAS PRESSURE FLUCTUATION CHARACTERISTICS INSIDE INLET PIPE, OUTLET PIPE AND THROUGH SNUBBER BY EXPERIMENT AND CFD SIMULATION

M. Sq. RAHMAN, G. H. LEE, H. J. LEE, H. M. JEONG and H. S. CHUNG, *Departmentof Mechanical and Precision Engineering, Gyeongsang National University, Korea,* Fossil-energy is declining because of its diversified uses by the rapid increased populations for modern civilization. Also there are some environmental adverse impact are due to uses of these fossil fuels. So it is needed to search for environment-friendly non-fossil fuels. Hydrogen energy is one of the promising future fuels for its properties including its renewability and environment-friendly nature. As it has less volumetric energy content and has less density compare to other gases so it is a prime need to compress its volume. Compression system is one of the most important processes in its production, transportation, storing and end-use. Reciprocating compressor is technically best appropriate for compression. Because of this, pressure becomes highly fluctuated. This phenomenon, of course, is very bad not only for hydrogen processing itself, but also for lifetime of equipment used. In order to reduce fluctuation of pressure produced by reciprocating compressor, snubber- a pressure pulsation damper unit is used. Then, gas flow crashes in the buffer of the snubber and is distributed to whole part the tube. By this way fluctuation of the gas is reduced. An experiment to observe reduction of pressure in the compressing system utilizing snubber has been conducted. From result obtained, the fluctuation is increasing proportionally when frequency of motor is increased. The pressure amplitude reduction values for 10, 20, 30, 40, 50 and 60 Hz motor frequency are varied from 55.95% to 58.46% with pressure loss of 0.07% to 2.48%. CFD analysis gives us detail information about the pressure including the critical pressure zone inside the tube of the snubber and the whole system.

16:00 ~ 17:20 (Room 106)

Drops and Bubbles (III)

Session Chair : Dr. Y. W. Ooi, Monash Univ/Malaysia

T-3F-1. PERFORMANCE OF ACTUAL SUGAR SYRUP WITH DIFFERENT METALS IN CAVITATING CONDITION

N. DIZADJI, *Islamic Azad University, Iran*, M. ASHRAFZADEH, *Islamic Azad University, Iran*, S. KANANPANAH, *Tehran University, Iran,* Cavitation erosion and measurement were carried out in closed circuit tunnel on lead, soft and hard aluminum to study the effect actual sugar syrup on cavitation erosion. The cavitation damage was studied by weight loss, scanning electron microscopy(SEM) and optical microscopy techniques. As had been found, for syrup the erosion rate in flow cavitation increased with increasing hardness metals and also erosion rate decreased with increasing viscosity syrup.

T-3F-2. ACCURACY ANALYSIS ON TAYLOR ANALOGY BREAKUP MODEL FAMILY

V. ESFAHANIAN, *Department of Mechanical Engineering, Faculty of Engineering, University of Tehran, Iran*, H. MOQTADERI, P. MOVAHED and F. Vakili FARAHANI, *Vehicle, Fuel and Environment Research Institute, University of Tehran, Iran,* Spray modeling plays an important role in engineering analysis, design in industry and also in state of art research on two-phase flow phenomena. Different sub-models including droplet breakup, atomization, collision, evaporation, energy and momentum interaction with main flow are required in order to model spray dynamics completely. One of the most important issues in spray modeling is breakup phenomena, which has modeled in different manners. In this paper, the focus will be on the Taylor analogy breakup family. O'Rourke and Amsden introduced the Taylor Analogy Breakup (TAB) model. It is based on Taylor's analogy i.e. the analogy between oscillating-distorting drops and a spring-mass system. The drop distortion is governed by a linear ODE for a forced, damped harmonic oscillator. The forcing term is given by the aerodynamic droplet gas interaction, the damping is due to the liquid viscosity and the restoring force is due to the surface tension. Two other breakup models have been introduced by Tanner based on TAB model called ETAB and CAB model which have overcome the shortcomings of the elder model. Several papers have discussed about each of above models in different case studies, but there is no complete comparison between the models of the TAB family in which different aspects of numerical accuracy, weakness and validity of each model is reported. In this study, these models are compared in different aspects due to experimental case studies using KIVA3V code. In order to compare these models the ETAB and CAB algorithms are added to the original KIVA3V code. The models have been validated and compared for non-evaporating sprays using the experimental data of Hiroyasu et al. In addition to gain more insight into different behavior of the models, first breakup time dependency on Weber number and distribution of Weber number of droplets at first breakup are studied.

T-3F-3. THE CALCULATION OF ENERGY FROM BREAK WATER

Y. F. AL-OBAID, *Faculty of Technological Studies, Kuwait,* In this paper the analysis is given for the energy due to the breakwater. This energy depends on the algebraic sum of energy transmitted by reflected from dissipated by incident upon and supplied by breakwater. A simple analysis is given to calculate all these effects. A simple experiment is described in order to evaluate such energy. The analytical results are compared with those from this experiment. Many tests have been carried out on both pneumatic and hydraulic breakwaters. In some cases full scale tests show energy parameters required for breakwater energy calculation using various surface waves. The variables taken care of in these experiments are due to air flow rate, air bubble size, water depth, number of manifolds, flow rate of water, break water inclination and depth and jet size. Using AL-Obaid's simple experiment, some parameters have been determined and they are used in the proposed analysis.

T-3F-4. CHARACTERIZATION OF BUBBLE-DRIVEN FLOW FIELD BY USING TIME-RESOLVED PIV / POD TECHNIQUE

S. J. YI, *Pusan National University, Korea*, H. D. KIM, *Pusan National University, Korea*, J. W. KIM, *Pusan National University, Korea,* K. C. KIM, *Pusan National University, Korea.,* The recirculation flow motion and mixing characteristics driven by air bubble stream in a rectangular water tank is studied. The time-resolved PIV technique is adopted for the quantitative visualization and analysis. 488nm Ar-ion CW laser is used for illumination and orange fluorescent (λex = 540nm, λem = 560nm) particle images are acquired by a 1280×1024 high-speed camera. To obtain clean particle images, 545nm long pass optical filter and an image intensifier are employed and the flow rates of compressed air is 3l/min at 0.5MPa. The recirculation and mixing flow field is further investigated by time-resolved POD analysis technique. It is observed that the large scale recirculation resulting from the interaction between rising bubble stream and side wall is the most dominant flow structure and there are small scale vortex structures moving along with large scale recirculation flow. The time-mean flow field has about 75% of total kinetic energy, which is the dominant dynamic structure in the set of instantaneous velocity fields. It is also verified that the sum of 20 modes of velocity field has about 67.4% of total turbulent energy. In the case of the lowest eigenvalue, the corresponding spatial mode represents the most dominant flow structure. Other, higher spatial modes represent smaller-kinetic-energy and smaller-scale flow structures; that is, of all the flow structures, they represent a high-frequency, small-scale flow field. The phase-space projection of the 1st temporal mode and the 2nd temporal mode is shown that it has approximately circular shape with small oscillation along the circular shape. The phase-space projection shows the periodic nature in small and large time scales.

16:00 ~ 17:20 (Room 107-108)