14:30 ~ 15:50 (Room 104)

Aerodynamics (I)

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

T-2D-1. USE OF CONFORMAL MAPPING TECHNIQUE TO THEORETICALLY VISUALIZE THE FLOW AROUND AN AIRFOIL GENERATED FROM A LIFTING CYLINDER IN A UNIFORM AERODYNAMIC FLOW FOR FLUSH PORT AIR DATA SYSTEM

U. N. MUGHAL, *NED Univ. of Eng. and Tech., Pakistan,* J. MASUD, *Air Univ., Pakistan,* Many methods have already been devised and are in use to calculate pressure co-efficient at various Mach Numbers and AOAs; a basic part of Aerodynamics. Conformal Mapping is also one of the various techniques by virtue of which we can develop such flows from simple aerodynamic bodies like cylinder. Potential flows start with an ideal flow over cylinders since the mathematics is more tractable. However, to use potential flow theory on usable airfoils the author have used conformal mapping to show a relation between realistic airfoil shapes and the knowledge gained from flow about cylinders. This method can further be used in the designing of an airfoil section. In this paper, author has used Joukowski Tranform to generate the flow around airfoils of various geometries. During the transformation the stagnation points on the cylinder map to stagnation points on the airfoil that were not realistic. Kutta condition was then used to force the stagnation point at the trailing edge. This was done by adjusting the value of vorticity strength Γ. Using different offset values different airfoil geometries were obtained. Once the profile of airfoil at different AOAs is generated in w plane then it is used to calculate the velocity profile over the entire contour which is further used to calculate the C_P variation over the complete geometry. These C_p values were then corrected using Karman-Tsien compressibility correction equations. Calculated C_p values were then compared with standard NACA Airfoil and were found very close to them. Corresponding pressure differential, ΔP values were then evaluated at different AOAs for different airfoil geometries.

T-2D-2. A CELL VERTEX BASED FINITE VOLUME TIME DOMAIN TECHNIQUE FOR MAXWELL'S EQUATIONS

Narendra DEORE and Avijit CHATTERJEE, *Department of Aerospace Engineering, IIT Bombay, India,* A three-dimensional cell-vertex finite volume time domain (FVTD) technique is developed to predict an electromagnetic scattering from aerospace configurations. The dynamics of the electromagnetic scattering is governed by the Maxwell equations. The time-dependent Euler equations of gas dynamics are normally solved using characteristic based numerical techniques in the FVTD framework. A similar approach is adopted to solve the time-domain Maxwell's equations posed as a set of hyperbolic conservation laws written in a total field form. This similarity accounts for the popularity of the FVTD technique for aerospace applications where characteristics based method is increasingly being applied to solve for electromagnetic scattering from aerospace configurations. Initial value problems based on "fluctuation-signal" framework was first proposed by Roe for time dependent Euler equations of gas dynamics. This approach leads to a cell vertex based finite volume techniques for hyperbolic conservation laws in multidimensions. The fluctuation-signal concept for one-dimensional time dependent Euler equations leads to the distribution of the nodal information based on the characteristics property of the hyperbolic conservation laws. Literature does not report a generalized natural multidimensional concept as an routine extension of one-dimension. The traditional way of extension to multidimensions is done by dimensional splitting equivalent to solving Riemann problems normal to cell faces. A basic feature of the scheme is, the values of the dependent variables are specified at cell vertices instead of cell centre and they form the basis for approximation of the flux balance for a computational cell. Special "distribution formulas" based on the fluctuation-signal approach, are used to distribute the second order change from the cell centre to the cell vertices. A three-dimensional cell vertex based finite volume time domain technique to solve the Maxwell's equations on structured grids has not been previously reported in literature. The benchmark test cases are validated and simulation results are compared successfully with the analytical results.

T-2D-3. ON THE EFFECTS OF ADVANCED AND DELAYED PITCHING ON A 2-D HOVEING WING

K. B. LUA, T. T. LIM and K. S. YEO, *National University of Singapore, Singapore,* This paper reports the results of an experimental investigation on the effects of advanced and delayed pitching on a two-dimensional (2-D) hovering wing. The hovering motion consists of simple harmonic translation/flapping and pitching motions. In the case of advanced pitching, the pitching motion is +30° phase angle (ϕ) in advance of the translation/flapping motion, and in the case of delayed pitching, the pitching motion is -30° phase angle behind the translation/flapping motion. Digital particle image velocimetry (DPIV) and strain gage force measurement techniques have been employed to measure the flow field and the associated aerodynamic forces. The DPIV setup consists of a 2-D flapping mechanism which is capable to execute coupled translation/flapping and pitching motions, a synchronized Nd-YAG pulsed laser as the illumination source and one megapixel digital CCD camera to capture the image data. Force measurement was conducted using a separate force measurement flapping mechanism installed on a 1m x 1m x 0.38m square tank. A 2-D force sensor was installed at the base of the wing above fluid surface. For a simple harmonic flapping with a symmetric pitching, both the lift and drag distributions show a "dual-peak" pattern over a single stroke of flapping. The first peak is mainly due to the induced velocity of the attached leading edge vortex formed in previous stroke and the second peak is due to the new leading edge vortex formed in the current stroke and also the wing pitching up motion. As for the advanced pitching, although it significantly increases the second peak in lift due to the early pitching up motion, it also causes negative lift at the beginning of a stroke due to the low angle of attack. Overall, the advanced pitching increases the average lift, accompanying by a significant increase in drag. On the other hand, the delayed pitching increases the first peak in drag and decreases the second peak significantly. This behavior can be attributed to the increasing angle of attack during the early part of a stroke and the decreasing angle of attack during the second half of the stroke. Overall the delayed pitching reduces the average lift and increases the average drag significantly.

T-2D-4. ALLEVIATION OF UNSTART OF A SUPERSONIC INTAKE ADOPTING COWL DEFLECTION AND BLEED

S. DAS, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India*, J. K. PRASAD, *Department of Space Engineering & Rocketry, B.I.T Mesra - Ranchi, India,* Ramjet powered aerospace vehicle needs compressed air, for a stable combustion, which could be obtained through air-intakes for improved performance of engines. At supersonic speeds, the shock wave system generated inside the intake influences the overall efficiency. One of the critical issues is the unstart phenomena of intake which could be induced by variations in the operating conditions or non-sustainable back pressure from the combustor. In general, this phenomena is unsteady in nature and may be detrimental for the vehicle. Mixed compression intakes are highly susceptible to the unstart phenomena compared to other forms of configuration. The intake unstart can be identified by the presence of subsonic flow near the throat which will have a complex internal flow. Suppression of these unstarts are done usually by boundary layer bleed, fluid injection, spillage through wall perforations etc. The present study deals with suppression of unstart by means of cowl deflection as well as by bleed and its effect on overall performance. A numerical simulation to study the starting phenomena of a supersonic air-intake has been made for a mixed compression rectangular intake designed using inviscid solutions. Computations have been made with RANS solver using k-ω turbulence model. Intake unstart was observed during viscous simulations which could be due to a large scale separation at the intake throat, expelling shock system out of the duct. Experiments were also made at a Mach number of 2.2. Schlieren photographs were taken and the flow field details inside the intake duct were investigated. It is observed through the computations made and as well through the experiments, that a deflected cowl could start the intake. Separately, simulations through adoption of boundary layer bleed, could also avoid the unstart. Studies were made for different cowl deflection angles and bleeding rates. Results indicate that deflected cowl has better performance in comparison to boundary layer bleed. This suggests that cowl deflection to control the separation could be thought of as an alternative to boundary layer bleed, to alleviate the starting problems of mixed compression intakes.

14:30 ~ 15:50 (Room 105)

Compressible Flows (I)

Session Chair : Prof. Y. Miyazato, Kitakyushu Univ/Japan

T-2E-1. INFLUENCE OF TRAILING WALL ANGLE ON THE UNSTEADINESS OF CAVITY FLOWS

VIKRAMADITYA, N. S. *IIT MADRAS, India*, Job KURIAN, *IIT MADRAS, India,* An experimental study of supersonic flow over two-dimensional cavities is presented in this paper. All the experiments were carried out at uniform free stream Mach number of 1.63. The Reynolds number of the boundary layer at the leading edge of the cavity is found out to be 1.47548e+07. For constant length to depth ratio (L/D=3) trailing wall angle of cavity was varied. Experiments were performed on six different trailing wall angle cavities to understand its effect on the unsteadiness of cavity flows. A measurement of unsteady surface pressure at various locations inside the cavity was carried out. Standard statistical analysis methods have been used to obtain various quantities of interest including the spectral distributions. On the analysis of pressure signals, reduction in amplitudes of the modes was observed as the angle decreases to lower value. High magnitude oscillations were observed for 90 and 75 degrees cavities. Steep fall in amplitudes of oscillations was noticed as the angle is reduced below 75 degrees. Virtually no oscillation was observed for cavities with 30 and 15 cavities. Temporal mode switching was observed dependent on trailing wall angle of the cavity. Multiple modes exist as the trailing wall angle is reduced to a lower value. Existence of strong acoustic wave inside the cavity responsible for high amplitude oscillations in 90 and 75 degrees trailing wall angle cavities was detected. It is noticed that as the trailing wall angle was reduced below 75 degrees the feed back mechanism has weakened considerably. It is found that overall cavity behavior changes significantly as the trailing wall angle is reduced below 75 degrees.

T-2E-2. COMPUTATIONAL STUDY ON THE OPERATING PROCESSES OF A TWO-STAGE LIGHT-GAS GUN

G. RAJESH, *Department of Mechanical Engineering, College of Engineering, Trivandrum, Kerala, India*, H. D. KIM, *Andong National University, Korea*, Y. K. LEE, *Poongsan Company, Chungnam, Korea,* Two-stage light gas guns are commonly used to simulate projectile velocities in the ballistic regimes, and have been extensively been used in hyper-velocity impact engineering, supersonic and hypersonic projectile aerodynamics and aeroballistics. In general, the conventional two-stage light-gas gun consists of three tubes, two diaphragms, a piston and a projectile. The high-pressure tube serves as the reservoir of high-pressure gas. The pump tube, which contains a light-gas, to increase the speed of sound, is connected to the high-pressure tube through a diaphragm separating both at the junction. A massive, freely movable piston is placed near the diaphragm in the pump tube. Projectile is kept in the launch tube which is connected to the pump tube through another diaphragm. Rupture of the diaphragm between the high-pressure tube and pump tube causes the piston to move at a high-speed and isentropically compress the light-gas to a much higher pressure than that in the high-pressure tube. With this rapid rise of the pressure inside the pump tube, a state is reached at which the second diaphragm ruptures and shock tube flow is initiated with the production of a strong unsteady shock wave in the launch tube. Resulting high-pressure state just behind the projectile produced by the reflection of the shock wave from the projectile base, drives the projectile with a very high-velocity. The performance of such a two-stage light-gas gun can be determined by the projectile speed at a given pressure in the high-pressure tube and a given mass of the piston. In this case, the projectile speed is dependent on many other parameters such as, the kind of light-gas (driver gas), the length and diameter of each tube, the isentropic compression process due to the piston motion and the shock compression on the base of the projectile. A large number of the works have been carried out analyze the processes inside the two-stage light-gas gun. However, none of the works has focused on the complete simulation of the device, which is very much important in terms of the fluid dynamic and structural aspects of the device. In the present study, a CFD method has been applied to predict the compressible flow field inside the two-stage light-gas gun, and to find out the dependence of several operating parameters on the projectile velocity and the peak pressures in the device, aiming at the performance enhancement of the two-stage light-gas gun. The unsteady, compressible Euler equations were numerically solved using a fully implicit finite volume method. The chimera scheme was employed to simulate the moving piston in the pump tube and the motion of the projectile in the launch tube. The computational results are compared with experimental data and found to be in very good agreement. Based on the computational results, it is seen that the complete interior ballistics of such guns can be simulated using CFD method with reasonable accuracy.

T-2E-3. EXPERIMENTAL STUDY OF ACTIVE CONTROL IN TRANSONIC DIFFUSER

M. YAGA, *University of the Ryukyus*, *Japan*, Y. UECHI, *University of the Ryukyus*, *Japan*, S. MATSUDA, *Okinawa National College of Technology*, *Japan*, I. SENAHA , *University of the Ryukyus*, *Japan,* Preliminary experiments of an active control of shock waves and the pressure fluctuations in a transonic diffuser were carried out using a piezo actuator attached at a throat of the diffuser. The experiments were performed with a 0.7MPa blow down wind tunnel. The test section consists of a 500mm circular arc half nozzle and the piezo actuator set at the throat. The flow was measured with the high response semiconductor pressure sensors and observed with the high speed camera by mean of schlieren technique. As the input signals to the piezo actuator, the sinusoidal voltage of 100Hz and 200Hz were adopted. As expected, the shock wave in the diffuser has clear correlation with the piezo actuator, where the dominant frequency of the unsteady positions of the shock wave is exactly the same as the input frequency. It is also confirmed that the flow pattern and the shape of the shock wave remain unchanged under the different input frequencies. The time averaged shock positions increases with the wind tunnel pressure ratio, which means that the oscillating shock wave moves downstream monotonically with the increase in the wind tunnel pressure ratio. The rms (root mean square) of the wall static pressure ratio also provide us with the information on where and how the shock wave approaches to the five monitoring positions. It illustrated the clear peaks at different pressure ratios at each monitored position. These clear peaks are caused by the large pressure difference between downstream and upstream of the approaching shock wave. However, the results of detailed FFT analyses of the wall static pressure fluctuations under various pressure ratio show that for the input frequency of 100Hz the dominant frequency is the same as the input frequency until the shock wave is located downstream of the monitored position. It also must be noticed that when the shock wave approached from upstream of the monitoring position, the spectrum of the relatively low frequencies than the input frequency becomes large. On the other hand, in case of f=200Hz, although the dominant frequency is still the same as the input frequency, the low frequencies have much less spectrum power than that for f=100Hz. However, when the state of the flow at the monitoring position becomes supersonic due to the increase in the pressure ratio, the spectrum of all the frequencies is decreased, which are also deduced from the sudden reduction of the rms of the pressure fluctuation. On the whole all the experimental results show that quite small displacement of the piezo actuator at the throat causes the large shock wave displacement and the large pressure fluctuations, which suggest the promising and potential application of the actuator to the noise reduction and to the high powered speaker if all the characteristic of the behavior of the shock wave and the flow are fully understood.

T-2E-4. INFLUENCE OF THE EXPANSION RATE OF NOZZLE ON TWO-DIMENSIONAL SUBSONIC JET

S. Y. SHIN, *Kyungpook National University*, *Korea*, S. H. KIM, *Kyungpook National University*, *Korea*, Y. D. KWON, *Kyungpook National University*, *Korea,* S. B. KWON, *Kyungpook National University*, *Korea,* From the view points of frequent applications in diverse industries such as a mixing augmentation scheme, an air knife system and so on, two-dimensional turbulent free jets issuing from a symmetrical constant expansion rate nozzle are studied by a numerical analysis and experiment. In numerical analysis, we used the commercial code of Fluent 6.0, and two-dimensional Navier-Stokes equation with standard k-ε model is used as governing equation. To calculate the dynamic viscosity of working fluid, the Surtherland equation is used, and the working fluid is air. In the case of the same nozzle stagnation condition and system external configuration, the influences of the nozzle expansion rate on the jet structures, the velocity distributions, the potential core width and length and the growth of half widths are investigated. In the measuring of velocity, we used a pressure measuring system made with a stainless string of 0.8mm in outer diameter. As results, in the potential core region, we can't find exactly the similarities in velocity with the variations of expansion rate of nozzle, while the similarity of velocity in the fully developed region exists. And, for the same nozzle stagnation conditions, we can’t find nearly the difference in potential core length with P. Furthermore, we can’t find the difference of velocity gradients in y direction at the potential core regions of the same x. Finally, it is found that the decay angle of potential core θ regardless of nozzle expansion rate is around of 5.0^{o}.

14:30 ~ 15:50 (Room 106)

Drops and Bubbles (II)

Session Chair : Prof. Mohammad Ali, BUET/Bangladesh

T-2F-1. MODELLING THE FORMATION OF A THERMAL SPRAY COATING USING A STOCHASTIC APPROACH

Mohammad P. FARD, Ali R. TEYMOURTASH and Ebrahim KAMALI, *Department of Mechanical Engineering, Ferdowsi University of Mashhad, Mashhad, Iran,* Thermal spray coating is a particulate deposition process in which powders of a material are injected into a high temperature flame region where they are melted and propelled towards the surface of a