longer travel distance. The mean ceiling temperature is distributed lower as droplet size is smaller because the small droplet does not reach to fire source by the buoyancy effect occurred during free combustion and heat transfer occurs fast near ceiling by interaction between water mist and fire plume. Among the spray nozzle used in the present simulation, K-3 by KIMM is showed the best effect of fire suppression because it has the largest heat transfer area by supplying the flow rate more than about 6 times than others.

W-1G-3. DEVELOPMENT AND TESTING OF AN AERODYNAMIC OPTIMISATION CODE USING DISCRETE ADJOINT METHOD

Manoj T. NAIR, *National Aerospace Laboratories, India,* A method for performing aerodynamic optimisation for aerofoils and wings is presented. The design variables used for optimisation are the parameters representing the geometry. The compressible Euler equations for flow are the governing equations. The discrete adjoint method is used to compute the sensitivity of the objective function with respect to the design parameters. The Complex Taylor's Series Expansion method is used to numerically compute the residual Jacobian required for the discrete adjoint method. The Euler equations are solved using the implicit matrix-free LU-SSOR method. The convective terms are discretized using the van-Leer flux vector splitting approach. The discrete adjoint equations are developed based on this discretization. The adjoint equations are solved using the scalar diagonal LU-SSOR method after introducing a pseudo time term. The constraints are applied by using the penalty function method. The code is parallelized using OPENMP directives. The developed code has been tested for aerofoil and wing optimisation at transonic speeds.

W-1G-4. EFFECT OF MERGING ANGLE ON MIXING OF HYDROGEN AND AIR BEHIND A THICKNESS BASE

Mohammad ALI, S. ISLAM, *BUET, Bangladesh,* A. K. M. SADRUL ISLAM, *IUT, Bangladesh,* To study the effect of merging angle on the mixing of hydrogen and air, the characteristics of the flow field, and flame holding capability of a supersonic combustor a numerical investigation has been performed. The merging angles of two streams are varied from 10˚ ~ 50˚. The flow fields are investigated by solving Two-Dimensional Navier-Stokes equations. A zero-equation algebraic turbulence model proposed by Baldwin and Lomax has been used to calculate the eddy viscosity coefficient. To delineate the purely fluid dynamic effects, the flow has been treated as non-reacting. It can be found that recirculations and penetration of hydrogen play an important role to enhance mixing. The area of recirculation decreases with the increase of merging angle but mixing efficiency increases. The recirculation regions and several shocks such as expansion shock, recompression shock and reattachment shock in the flow field are evident.

10:40-12:00 (Room101)

Supersonic and Hypersonic Flows (I)

Session Chair : Prof. J. Kurian, IIT Madras/India

W-2A-1. NUMERICAL STUDY ON SUPERSONIC IMPINGING JET FROM COLD SPRAY NOZZLE

H. KATANODA, M. FUKUHARA, *Kagoshima University, Japan,* The cold spray is an innovative spray coating method which was patented in 1994 in the USA. It uses a supersonic gas flow to spray solid particles to make a coating on the substrate. In the paper, the over-expanded impinging jet from a cold spray nozzle, as well as the velocity of the particle accelerated by the gas flow, was studied by numerical simulation. The effects of the stagnation pressure and temperature upstream of the throat on the flow field and the particle velocity were investigated. The nozzle has a throat diameter of 2.0mm and the exit diameter of 5.0mm. The distance from the nozzle exit to the impinging wall was set at 10mm. The nitrogen gas was used as a process gas. The ranges of the stagnation pressure and temperature upstream of the nozzle throat are set as 2.0 - 3.0 MPa and 300 - 675 K, respectively. In this simulation, the spherical copper particle with 15 m in diameter was selected as the spray particle. The particle velocity was calculated based on the one-way coupling method along the center line of the gas flow. The numerical results of the gas flow shows that there exists minor effect of the stagnation conditions on the Mach number distribution in the nozzle. On the other hand, the gas velocity in the nozzle increases by increasing the stagnation temperature. The calculated particle velocities show that the shock wave structure at the nozzle exit has a negligible effect on the velocity distributions of the 15 m copper particle. In addition to that, the stagnation temperature has a larger effect on the particle velocity than the stagnation pressure. From the present numerical simulation it is concluded that increasing the stagnation temperature is more effective than the stagnation pressure to increase the impact velocity of the particle.

W-2A-2. UNSTEDY BEHAVIOR OF SHOCK WAVES AROUND A CIRCULAR ARC BLADE WITH BUMP IN TRANSONIC MOIST AIR FLOW

S. MATUO, A. B. M. T. HASAN, I. TOMOHIRO, T. SETOGUCHI, *Saga University, Japan,* H. D. KIM, *Andong National University, Korea,* The transonic flow over the airfoil is characterized by shock waves standing on the surface. In this case, the interaction between the shock wave and boundary layer becomes complex because the shock wave imposes an adverse pressure gradient on the boundary layer. As a result, the self-excited shock wave oscillation occurs in the flow fields. The unsteady phenomena in the transonic flow around airfoils are also observed in the flow field of fan, compressor blade, butterfly valves and so on. In the transonic or supersonic flow where vapor is contained in the main flow (moist air), a non-equilibrium condensation process occurs at a supersaturated state. The condensation phenomena coupled with fluid flow is important in many engineering and technical application such as supersonic nozzle, steam turbine, cryogenic turbomachinery, shock tube, transonic wing, helicopter blades and so on. However, the effect of non-equilibrium condensation on the internal flow around the transonic airfoil has not been studied satisfactorily. In the present study, the effect of non-equilibrium condensation in moist air flow on the characteristics of self-excited shock wave oscillation on the circular arc blade with or without the bump was investigated experimentally and numerically. Results obtained are as follows: shock strength in the case of blade with bump becomes weak compared to that without bump model, and amplitude and range of oscillations become small for the case of blade with bump for all blade angles of attack. For a circular arc blade with angle of attack, the distributions of condensate properties are mainly observed on upper region around the blade. Furthermore, the non-equilibrium condensation reduces the frequency and amplitude of the oscillation compared with the case of no condensation for all angles of attack.

W-2A-3. INVESTIGATIONS OF WEAK NORMAL SHOCK WAVE/LAMINAR BOUNDARY LAYER INTERACTIONS IN DUCTS

Y. MIYAZATO, H. YAJI, K. MATSUO, *The University of Kitakyushu, Japan,* The aim of the present research is to elucidate the interaction of a weak normal shock wave formed at just downstream of a nozzle throat with a laminar boundary layer in a two-dimensional duct. The wall contours of the two-dimensional nozzle used in the present experiment are designed by the method of characteristics to be uniform flow at the nozzle exit. The nozzle has heights of 4.4 mm at the throat and of 4.9 mm at the exit with a design Mach number of 1.39. The height at the inlet of the test section (or the nozzle exit) is 4.9 mm, and both the upper and lower walls of the test section are inclined at 0.7 deg to the central axis to stabilize the location of the shock wave in the test section. A static pressure measuring system on a central axis called a through-tube has been devised and the centerline static pressure data in the interaction region have been obtained by the through-tube. Flow visualization by the colour schlieren method with a tricolour filter has been employed to observe the structure of the shock wave interacting with a laminar boundary layer. As conclusions, flow visualization shows that a weak normal shock wave for the freestream Mach number below around 1.3 interacts with a laminar boundary layer to form a shock train. Centerline static pressure distribution has two peaks to show the existence of first and second shocks in a shock train. Also, the static pressure rise across the interaction region decreases as the ratio of the laminar boundary layer thickness to the duct half height increases.

W-2A-4. CHARACTERISTIC OF MULTIPLE PRESSURE WAVE CAUSED BY DISCHARGING OF PULSATING PRESSURE WAVE FROM OPEN-END OF TUBE

T. YASUNOBU, *Kitakyushu College of Technology*, *Japan*, H. KASHIMURA, *Kitakyushu College of Technology*, *Japan*, T. SETOGUCHI, *Saga University*, J*apan,* When the pulsating pressure wave propagated in the tube reaches at open end of a tube, the reflection and emission of pressure wave occurs and the multiple pressure wave is formed. This multiple pressure wave causes the some noise problems as like exhaust tube of automobile so that the mechanism and characteristic of the multiple pressure waves must be more cleared to control the noise problem. Many papers had described for the emission of a single pressure wave from an open end of a tube, as like the shock wave or the compression wave. But, it seems that the emission of the pulsating pressure wave, which is very strong

compared with the sound wave, is not enough researched in past papers. This paper, therefore, aims to clear the characteristic of multiple pressure wave, especially the attenuation and directivity of that wave, and the relation between the pulsating pressure wave and multiple pressure wave by the experiment and numerical analysis using TVD method. In the experiment, the spherical valve is used for generation of the pulsating pressure wave in the propagation tube and the tube diameter are D=19 mm , 35mm The metal spherical with diameter 25mm is rotated by the motor and the pulsating pressure wave is formed in the propagation tube. In the numerical calculation, the x-y cylindrical coordinates system was considered. The basic equation is the compressible unsteady axisymmetric Euler's equation and was solved by the TVD method.
The results are as follows. (1) The attenuation and directivity of the multiple pressure wave are clearly observed and this characteristic corresponds to that of the simple impulsive wave caused by the weak shock wave or the compression wave. (2) The over pressure of the multiple pressure wave, which is non-dimensionalized by the over pressure of the pulsating pressure wave propagated in the tube Δp^*, Δpr, $\max/\Delta p^*$ decreases with an increasing of the distance from an open end of a tube r/D and the degree from a tube axis θ, caused by the attenuation and directivity. The directivity for axis direction is a large compared with the radial direction. (3) The over pressure of the multiple pressure wave Δpr,$\max/\Delta p^*$ decreases with an increasing of the length of the pulsating pressure wave L/D and the strength of that wave strongly depends on the length of the pulsating pressure wave. (4) The degree of attenuation of the multiple pressure wave approximately corresponds to the result obtained by the analysis for the simple impulsive wave and it is possible to estimate the attenuation of the multiple pressure wave using this analysis for simple impulsive wave.

10:40-12:00 (Room102)

Biofluid Dynamics (II)

Session Chair : Prof. M. S. Saidi, Isfahan Univ of Tech/Iran

W-2B-1. LOAD SUPPORTED BY FLOATING ELASTIC SHEETS

K. J. PARK, *Seoul National University, Korea*, D.-G. LEE, *Seoul National University, Korea*, H.-Y. KIM, *Seoul National University, Korea,* Water striders can float and move on water by effective use of surface tension. In particular, their one superhydrophobic leg is known to be able to support much heavier load than their body weight without sinking. The legs of water striders and their biomimetic robots are long and thin so that they are flexible. We focus on how the legs in static equilibrium are deformed by hydrostatic pressure and surface tension. As a model for the legs, we theoretically and experimentally study the deformation of floating flexible sheets. To predict the deformation we use a theory of elasticity under assumption that the transverse deflection is very small compared to the sheet length. The deflection is determined with the boundary conditions including the surface tension effects of water-air interfaces, whose profiles are obtained by using the Young-Laplace equation. The equilibrium shapes of the sheets are experimentally measured. We find that the descent distance of the edge clamped to the load is different from the descent distance of the tip free edge due to the deflection. The experimental data are compared with the predictions and it shows that they are in good agreement. Finally, by calculation of the shear force at the clamped edge, we show that flexible sheets can sustain heavier load than rigid sheets. In addition, we find out that one important parameter, which determines the deformation and load capacity, is the elastocapillary length.

W-2B-2. DETACHMENT OF SEED FROM A DANDELION CLOCK

M. HASEGAWA, *Department of Mechanical Engineering, Toyo University, Japan,* O. MOCHIZUKI, *Department of System Robotics, Toyo University, Japan,* We discussed aerodynamic reasons why the seeds detached from the portion of the dandelion clock first in a uniform flow through visualization experiments of a porous sphere. The first detachment position of the seeds from the peicarp was in a range from 33 degrees to 106 degrees measured from a front stagnation point. The flight performance of the wind-borne dandelion seeds was investigated through a free fall experiment in a box without external turbulence. We used the high-speed CCD camera to measure the speed of a free falling seed The seed consists of a pappus, stalk and achene. Effects of numbers of thin feathered structures of the pappus on the final speed were observed to know the drag coefficient of the structure. To know which seeds start to fly, visualization experiment of the seeds detaching from a pericarp of a dandelion clock was carried out in a uniform flow with various flow-speeds. We used the CCD camera to record the first detachment portion. The flow penetrates the dandelion clock. The dandelion clock was supposed to be a porous sphere. The drag force, D, was measured by a thin plate pasted strain gages which was installed the porous sphere. Streak lines were observed by ink. The definition of the angle measured from the front stagnation point of the dandelion clock in a uniform flow and forces acting on a seed. The first portion where the seed detached was found to be in a range from 33 degrees to 106 degrees. The relation between the final speed of a seed and the number of thin feathered-structure of the pappus was found that the final speed is found to be inversely proportional to a root. This means that the total drag of the pappus is able to be expressed by linear summation of drag-values of individual thin feathered-structures. Thus, the forces acting on the seed are easy to estimate if the low Reynolds number flow around the thin feathered-structure is known. Interval between vortices behind the porous sphere is smaller than that behind the solid sphere. The drag coefficient of the porous sphere was smaller than that of the solid one. The close-up view of the flow ejecting from holes at rear side of the porous sphere, this ejection was intermittent but was synchronized with shedding vortices. The back flow was reached at = 105 degrees. This should affect the outer flow along the surface. We estimated force acting on a thin-feathered structure of the pappus and observed the first portion of the detachment of seeds from the dandelion clock. The force to lift the seed was expressed by the individual force times the number of the thin feathered-structure. The first portion of the detachment was the front area centered at 70 degrees from the front stagnation point of the dandelion clock.

W-2B-3. THREE-DIMENSIONAL SIMULATION OF A FLAPPING FLAG IN A UNIFORM FLOW BY THE IMMERSED BOUNDARY METHOD

W.-X. HUANG, H. J. SUNG, *KAIST, Korea,* We proposed an immersed boundary (IB) method for three-dimensional simulation of flapping flags in a uniform flow. In the present method, a direct numerical scheme is developed to calculate the flag motion, with the elastic force treated implicitly, while the fluid motion defined on an Eulerian grid is calculated using an efficient Navier-Stokes solver. An additional momentum forcing is formulated from the flag motion equation in a way similar to the direct-forcing IB formulation and acts as the interaction force between the flag and its ambient fluid. When the gravity force is excluded, the flag flaps almost uniformly along the spanwise direction, with slight asymmetry about the centerline, which is attributed to the hyperbolic property of the flag motion equation. An O-shape vortical structure is shedding from the trailing edge, connected by a Ω-shape structure shedding from both side edges. After including the gravity force, the sagging-down of the flag and the rolling-up of the upper-corner deform the vortical structures. The Strouhal number defined in terms of the flapping amplitude increases slightly with increasing the Reynolds number and is between 0.16 and 0.22, consistent with the general value of a flying or swimming animal. The onset of regular flapping is found to be subcritical and the bistability region is narrow in our simulations. A linear stability analysis for a flag of infinite spanwise width shows that the most unstable mode corresponds to the flag uniform in the spanwise direction. The aspect ratio effect was analyzed through numerical simulations and the theoretical results were compared with the numerical results and those from previous studies. Nodeless flapping and flapping with an imperfect node were observed in the present simulations.

W-2B-4. LIFT PRODUCTION OF A TWO-LINK FLAPPER IN A VISCOUS FLUID

J. BAI, *The State Key Laboratory of Nonlinear Mechanics, IMECH, CAS, China*, X. ZHANG, *The State Key Laboratory of Nonlinear Mechanics, IMECH, CAS, China*, A recent experimental study by Childress et al. (Physics of Fluids (2006) 18, 117103) shows that small flexible bodies made of stiffened tissue paper can hover in an oscillating air flow. In this paper, a numerical simulation is conducted to investigate the lift production mechanism of such ‘flapper’. The flapper is modeled as two articulated rigid plates joined by one hinge. Instead of an oscillating flow, a harmonic motion of the articulated body in the vertical direction is assumed. In order to treat the moving boundaries associated with this problem, a flow solver based on the Immersed Boundary (IB) method is developed. In this approach, the partial differential equations are solved on a Cartesian grid which is not body-fitted. Since no re-meshing is needed, IB method exhibits its advantages in simulating flows with moving boundaries. For the details of the IB method used in this study, please refer to the paper by Takeno Kajishima.[3] The state of motion of the flapper is determined by solving the Euler-Lagrange equations. Numerical results indicate that the flapping of the articulated body can produce lift with certain combinations of amplitude and frequency. In this paper, instead of an oscillating flow, the flapper is oscillating in a fluid which is considered to be incompressible and viscous. The hinge that links the two elements is modeled by a torsion spring with positive stiffness. While the point which connects the two elements is oscillating along the vertical direction, the vibration function is