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