

*University of Science, Japan.* In Electro Chemical Machining process, generation of air bubbles and the flow separation complicate the flow behavior of electrolyte and decrease the shape accuracy of the product. It is important to understand the influence of air bubbles on the separated flow. In this study, the flow characteristics of gas-liquid two-phase flow with millimeter-scale bubbles and separation in a channel are examined. Bubbles are fed to the liquid phase from stainless-steel pipes. Flow separation is generated by a square rod. All the experiments are conducted at the bulk velocity,  $U_b$ , of 370 mm/s in the single-phase flow, corresponding to the Reynolds number of 2000 based on the square rod height and the bulk velocity. The void fraction  $\alpha$  is set from 0 to 0.5 %. In order to compare the effect of the bubble diameter with the same void fraction, 3-pentanol ( $C_5H_{11}OH$ ) of nearly 30 ppm is added as a surfactant. We have applied a PIV system with fluorescent particles to the measurement of the liquid flow. In order to capture the shape of bubbles, the projected shadow image technique is applied. For the case without the surfactant, bubble diameter takes the peak value at about 0.7 and 2.9 mm. On the other hand with the surfactant, it takes the peak value at about 0.7 mm within a small dispersion for all conditions. The streamwise velocity becomes higher in the core flow region and the flow reversal as the void fraction increases, while that in the separating shear layer become lower. For the case of the flow without surfactant, streamwise velocity fluctuation takes high value at the high void fraction. Also, transverse velocity fluctuation is higher compared with in the single-phase flow. On the other hand, in the case with the surfactant, transverse fluctuation becomes lower as the void fraction increases.

11:00 ~ 12:20 (Room107-108)

**Computational Fluid Dynamics ( I )**

Session Chair : Dr. S. K. Hong, ADD/Korea

**M-1G-1. STABILITY ANALYSIS OF UNIFORM ANNULAR PASSAGES CONDUCTING INCOMPRESSIBLE LAMINAR FLOWS FOR SMALL AMPLITUDE OSCILLATORY ROTATION OF THE OUTER CYLINDER**

H. YARJABADI, *Department of Mechanical Engineering BuAli-Sina University, Iran.* In this paper a computational method is developed involving the simultaneous integration of the Navier–Stokes and structural equations for the purpose of studying the stability of concentric annular passages conducting incompressible laminar flows. It is assumed that one side of the annulus, i.e. the centre-body, is fixed and the outer cylindrical duct is flexibly supported. The outer cylinder is displaced or rotated from its equilibrium position and is then released. In this situation, the fluid part of the problem is solved by an accurate method using a three-point backward implicit scheme, followed by a pseudo-time iteration using an artificial compressibility factor. The fluid equations are discretized in space based on a finite-difference formulation and primitive variables, for which stretched staggered grids are used. The resulting equations are cast in delta form and are solved using an Alternating Direction Implicit (ADI) scheme. The fluid forces acting on the vibrating cylinder are calculated from the integration of the unsteady pressure and shear stresses resulting from the unsteady primitive variables calculated. The equations of motion of the structure, subjected to the calculated fluid forces are solved using the Runge–Kutta scheme to obtain the displacement or rotational angle of the moving cylinder. The problem is solved for small amplitude motions, by means of the so-called mean position (MP) analysis. It is shown that, between translational and rotational motions of the outer cylinder, the most stable configuration is that of rotational motion. This study can be used for FIV and FSI analysis of the annular structures.

**M-1G-2. SIMULATION OF FLOW AROUND A MOVING BODY USING UNSTRUCTURED CHIMERA GRID METHOD**

S. Z. NI, X. ZHANG, G. W. HE, *Institute of Mechanics, CAS, China.* Flow around a moving body is ubiquitous in practical problems such as insect flight, fish swimming and leave falling. The Chimera (or overset) grid method is an efficient method for treating such problems. In this method, Grids for each sub-domain can be generated separately. Governing equations are also solved independently and the solutions of each sub-domain are coupled through the exchange of information across the interior boundaries. The unstructured grid can easily generate the mesh complex geometry and reduce the number of sub-domains significantly. The present method preserves the advantages of the Chimera grid method and the unstructured grid. A novel numerical scheme proposed recently by our research group is used to solve the incompressible N-S equations and a “hole-cutting” method is developed to determine the intergrid boundary in background mesh. The moving mesh control volume method is deployed to treat the moving sub-domain. For the moving control surface, the space

conservation law (SCL) has to be satisfied. The SIMPLEC algorithm is used to couple the pressure with the velocity. A second-order upwind scheme is used for the discretization of convective term and the Crank–Nicholson scheme is used for the temporal advancement. The collocated grid arrangement is deployed and a Rhie–Chow interpolation is used to compute mass fluxes at faces to eliminate pressure oscillation. And the Schwarz method is followed to couple the solutions in different sub-domains. Flows around an oscillating circular cylinder and a hovering wing are simulated. The numerical results are in good agreement with other experimental and computational data in literature. Basic features of flow around these moving bodies are successfully captured. These numerical examples demonstrate the capability of our method in handling moving boundaries.

**M-1G-3. NUMERICAL SIMULATION OF FLOW OVER PITCHING BODIES USING AN IMPLICIT REYNOLDS AVERAGED NAVIER STOKES SOLVER**

Sharanappa V. SAJJAN, Vimala DUTTA, P. K. DUTTA, *Computational and Theoretical Fluid Dynamics Division National Aerospace Laboratories, India.* There is a great demand for efficient and accurate computation of unsteady aerodynamic loads for aeroelastic investigation of modern aircraft in the transonic regime. The availability of high speed computers and advances in numerical methods in recent years have led to the successful development of sophisticated aeroelastic simulation techniques. Recently, several aeroelastic applications have employed Euler and Reynolds-Averaged Navier-Stokes (RANS) solvers for flutter analysis using both frequency domain and time domain approaches. However, in order to be an integral part of a flutter analysis package, the Euler or RANS solver is to be evaluated through time accurate computations for flow past standard aeroelastic configurations. The present paper is an attempt towards demonstrating the capability of an implicit Reynolds-averaged Navier-Stokes solver (IMPRANS) for simulating unsteady compressible flows over pitching aerofoils and wings. The RANS solver used for obtaining the time-accurate solutions is based on an implicit finite volume nodal point spatial discretisation, wherein a control volume is formed by joining the centroids of the neighbouring cells around a nodal point in the computational domain. The algebraic eddy viscosity model due to Baldwin and Lomax is used for turbulence closure. Computations are presented for transonic flow past a pitching NACA 64A10 aerofoil and a pitching LANN wing. The aerodynamic coefficients as well as the mean and fundamental frequency pressure data for these two- and three-dimensional aero-elastic configurations at transonic Mach numbers are found to be in good agreement with the experimental data. While the mean surface pressure distribution compares very well for both the cases, the first harmonic shows only reasonable agreement over the outboard region of the LANN wing, possibly due to insufficient grid size. The present work thus demonstrates the capability of the solver to provide useful unsteady pressure data for aero-elastic analysis.

**M-1G-4. FLOWS INDUCED IN A FLUID-FILLED COMPLIANT CYLINDER OSCILLATED BY AN EXTERNAL FLOW**

M. YOKOYAMA, O. MOCHIZUKI, *Toyo University, Japan.* The objective of this research is to understand internal flow fields inside a fluid-filled compliant cylinder placed in a uniform flow. The inside flow induced by the deformation of a soft-matter object was investigated numerically. Since the pressure and shear stress determined by the flow fields act on the outer and inner surfaces of the membrane of the cylinder, the Navier-Stokes equation and equation of motion of the soft material were combined. The membrane consisted of units of a mass-spring-damper (MSD) system. The simulation was carried out in a 2-dimensional region. The initial cross section of the compliant cylinder was a circle and placed at the origin of the coordinates. The non-slip condition was satisfied at the wall. The mesh was regenerated every time steps to fit the deformed shape of the cylinder. The inner fluid was the same as that of the outer flow in this study. We showed the change in aspect ratio of the cross section and the velocity fluctuation due to the Kármán vortices. The AR oscillates with the natural frequency of the cylinder for a while after the beginning of the calculation because of the impulsive start. After the natural vibration settles, the rear wall vibrates with the shedding frequency of Karman's vortices. The amplitude of the vibration of the rear wall was approximately 0.2% of the initial diameter of the cylinder. The inner flow pattern during the natural vibration became the saddle point. On the other hand, the source-sink flow between the different portions at the rear wall inside the cylinder is induced by the excitation due to the shedding vortices. The direction of the flow becomes opposite due to the shedding frequency. This indicates the possibility to be applied the outer vibration to mixture of different liquids inside a compliant container.