Computational Fluid Dynamics (VI)
Session Chair : Prof. K.-S. Yang, Inha Univ/Korea

T-3G-1. ESTIMATE THE INFLUENCE OF THE DIFFERENT DIFFUSER CONFIGURATION BY CFD

L. P. CHUA, B. Y. SU, T. M. LIM, S. C. M. YU, *Nanyang Technological University, Singapore,* Partially due to the deficiency of the available heart donors, the ventricular assist devices (VADs) have been accepted as an alternative therapy for patient at risk of dying from severe heart failure. Axial flow VAD has gained tremendous interest due to its small size and favorable anatomical fit. Most axial flow VADs basically consist of a straightener, an impeller (and an inducer) and a diffuser, and these are enclosed by the pump casing. The diffuser could significantly influence the performance of the VAD, and its design is even more challenging than the impeller due to the adverse pressure along flow direction and the swirling flow from the impeller. It was found in the CFD simulation that the flow pattern at different radial location was varied from hub to tip. The undesirable flow was appeared close to the hub, because flow at lower radial location did not have enough kinetic energy to over come the adverse pressure across the diffuser. In addition, the velocity and pressure distributions close to either side of the diffuser blade were significantly different from each other, especially at lower radial location close to the hub. In order to improve the flow pattern close to diffuser hub, the first two diffuser configurations were evaluated with zero and 0.3 mm clearance when the diffuser hub was stationary. The last model had a 0.3 mm clearance when the impeller and diffuser hub were combined so that the diffuser hub rotates with the impeller. In this study, these different diffuser configurations were evaluated and the other configurations of the axial VAD were constant. The simulation result showed that the clearance close to the diffuser hub improved the flow field with a decreased hydraulic performance and the possible location of the thrombus formation could be vanished. Additionally, the hydraulic performance recovered when the diffuser hub rotated with the impeller at the same speed. Although, the rotating diffuser hub increases the hemolysis level, it mainly results from the reduced flow rate. This issue could be solved by decreasing the clearance gap until the flow rate completely recovers as Model 1. Therefore, the rotating diffuser hub with a suitable clearance is viable choice to optimize the hydraulic performance, hemolysis and thrombus formation.

T-3G-2. A STUDY ON THERMAL VAPOR COMPRESSION IN MULTI EFFECT DESALINATION PROCESS

T. S. UTOMO, *Gyeongsang National University, Korea*, M. K. JI, *Gyeongsang National University, Korea*, D. Y. CHOI, *Gyeongsang National University, Korea,* H. S. CHUNG and H. M. JEONG, *Gyeongsang National University, Korea,* TVC is a dominant element governing total process of MED (Multi Effect Desalination). The accurate prediction of the TVC performance promotes the reliability of the process and the enhancement of the TVC entraining efficiency improves the performance of MED significantly by reducing the amount of motive steam. TVC is a kind of typical steam ejector which entrains low pressure steam using the shear force of supersonic flow made by a small supersonic nozzle mounted in the center of a large converging-diverging duct. In this study, CFD (computational fluid dynamics) analysis based on the finite volume method was employed to investigate the influence of angle of converging duct on the ejector performance. The CFD results were verified with available experimental data. Flow field analysis was also carried out in this study. Variation on the angle of converging duct was made as 0^o, 0.25^o, 0.5^o, 1.25^o, 2.0^o and 2.75^o. The converging duct with an angle of 0.5^o gives the highest value of entrainment ratio that is 0.941. Furthermore, from this study it can be concluded that the optimum angle of converging duct when the entrainment ratio reached the highest value is obtained at 0.5^o.

T-3G-3. A HIGH-SPEED COMPUTATION FOR THE ADDED MASS IN VISCOUS FLUID AND ITS VALIDITY

K. HIRATA, *Mechanical Engineering, Doshisha University, Kyoto, Japan ,* H. SHIMOHARA, *Doshisha International High School, Kyoto, Japan*, S. KIKUKAWA, *Mechanical Engineering, Doshisha University, Kyoto, Japan,* M. YASHIMA, *Mechanical Engineering, Doshisha University, Kyoto, Japan,* In many fluid-structure interaction problems which are common in the designs for marine structures, heat exchangers of nuclear plants and so on. The added mass, namely, the vertual mass, is one of the most important concepts. In order to specify the added mass more efficiently and more conveniently, various analytical and/or numerical methods have been developed. For non-simple boundary problems, it has been needed to solve the full Navier-Stokes equations using a domain method such as the finite element method and the finite differential method. The domain method is not efficient, and requires much computer resources and time. In the present study, we consider a simple method to specify the added-mass coefficients of arbitrary two-dimensional solid bodies with non-simple cross sections, efficiently and conveniently. Specifically, we consider a two-dimensional incompressible and viscous fluid under the assumption of infinitesimal oscillation amplitude of the body, and properly modify the full Navier-Stokes equations into linear equations, that is, the Brinkman equations. The solving method is based on a discrete singularity method (refer to as DSM), in which we employ a fundamental solution of the Brinkman equations as a singularity. In order to show the method's effectivity and validity, we compute some examples in infinite flow fields, for some of which we confirm good agreement with analytical results. We solve the full Navier-Stokes equations by a finite difference method (referred to as FDM), and compare such solutions with the DSMs specifying the valid range for the DSM. As a result, we have confirmed the non-linear amplitude effect and specified the valid range of the DSM. It is seen that the FDM results asymptote to the DSMs with decreasing *KC*. In order to specify the DSM's valid range quantitatively, we consider a relative error *e*. *e* increases with increasing *KC* or with increasing *S*. In addition, we show the flow field such as streamlines and vorticity distribution.

T-3G-4. FLOW PAST TWO NEARBY SPHERES

D.-H. YOON, *Inha University, Korea*, K.-S. YANG, *Inha University, Korea,* In this investigation, flow patterns past two identical nearby spheres at Re=300 were numerically studied. We considered all possible arrangements of the two spheres in terms of the distance between the spheres and, the angle inclined with respect to the main flow direction. It turns out that significant changes in shedding characteristics are noticed depending on how the two spheres are positioned. Collecting all the numerical results obtained, we propose diagrams for flow pattern on the distance vs. angle plane. The perfect geometrical symmetry implied in the flow configuration allows one to use those diagrams to identify flow patterns past two identical spheres arbitrarily positioned in physical space with respect to the main flow direction.