Proceedings of the Korean Nuclear Society Spring Meeting
Cheju, Korea, May 1996

Development of A Three-Dimensional Thermo-Hydraulic Computer Code for Incompressible Flows in Complex Geometries

Seok Ki Choi, Yong Bum Lee, Jong Sun Hwang, Ho Yun Nam and Mann Cho
Korea Atomic Energy Research Institute

ABSTRACT

A three-dimensional thermo-hydraulic computer code is developed for simulation of incompressible flows in complex geometries. The computer code employs a body-fitted, nonorthogonal grid system in order to efficiently handle the complex geometries encountered in many engineering applications. The finite volume method is used to discretize the governing equations and the convection term is treated by higher-order bounded schemes. The cell-centered, nonstaggered grid arrangement is adopted and the resulting checkerboard pressure oscillation is avoided by use of momentum interpolation practice. The computer code employs the SIMPLE algorithm for pressure and velocity coupling and the k-ε turbulence for turbulent calculation. The computer code has been tested through application to a variety of test problems and some results are presented in this paper

1. INTRODUCTION

Most of the flows of engineering interest occur in domains of three dimensional irregular geometries. The development of an efficient calculation method that can resolve fluid flow and heat transfer in such a complex geometry is very important for the practical engineering applications. In recent years, several calculation methods [1] which employ the nonorthogonal, body-fitted coordinates have been developed for a better resolution of flow field in complex geometries. Among these methods, Rhie and Chow [2] proposed a scheme based on the momentum interpolation method.
In this scheme, the momentum equations are solved at the cell centered locations using the Cartesian velocity components as dependent variables and

the cell face velocities are obtained through the interpolation of momentum equations for the neighbouring cell centered Cartesian velocity components. A special kind of interpolation practice is devised to prevent the splitting of pressure field.

In the present study, a three-dimensional finite volume calculation procedure for incompressible flows in complex geometries is presented. The scheme is based on the Rhie and Chow's scheme in which the curvilinear contravariant velocity components are selected as cell face velocities. The present scheme is applied to the test problems to assess its capabilities. The numerical results are compared with available experimental data.

2. MATHEMATICAL FORMULATION

2.1 Governing Equations

The conservation form of transport equation for a general dependent variable ϕ in a generalized coordinate system x^j can be written as follows;

$$\frac{\partial}{\partial x^j} \left(\rho U_j \phi - \frac{\Gamma_\phi}{J} D_m^j \frac{\partial \phi}{\partial x^m} \right) = J S_\phi \qquad (1)$$

where the contavariant velocity components U_j and the geometric coefficients D_m^j are defined as $U_j = b_k^j u_k$, $D_m^j = b_k^j b_k^m$ and the geometric coefficients b_i^j represent the cofactor of $\partial y^i/\partial x^j$ in the Jacobian matrix of the coordinate transformation $y^i = y^i(x^j)$ and J is the determinant of the Jacobian matrix. In these equations, ρ is the density of fluid, Γ_ϕ is the diffusion coefficient of varible ϕ, u_i are the Cartesian velocity components in y^i directions and S_ϕ denotes the source term of variable ϕ.

2.2 Discretization of Transport Equations

The computational domain is divided into hexahedral control volumes and all Cartesian velocity components and scalar variables are stored at the geometric center of each control volume cell. The discretization of transport equations is performed in the physical solution domain following finite volume approach with the convection terms treated by the higher-order bounded scheme. The resulting algebraic equations for a variable ϕ can be written in the following general form.

$$A_P\phi_P = A_E\phi_E + A_W\phi_W + A_N\phi_N + A_S\phi_S + A_T\phi_T + A_B\phi_B + b_\phi \qquad (2)$$

2.3 Momentum Interpolation Method

In the Rhie and Chow's scheme, the momentum equations are solved at the cell centered locations. The dicretized form of momentum equations for the Cartesian velocity components can be written as follows;

$$(u_i)_P = (H_{u_i})_P + (D^1_{u_i})_P(p_w - p_e) + (D^2_{u_i})_P(p_s - p_n) + (D^3_{u_i})_P(p_b - p_t)$$
$$+ (1-\alpha)(u_i^{n-1})_P \qquad (3)$$

where

$$H_{u_i} = \alpha(\sum_{nb} A^{nb}_{u_i} u_i^{nb} + b_{u_i})/A_P^{u_i}, \qquad D^j_{u_i} = \alpha b^j_i / A_P^{u_i} \qquad (4)$$

In these equations, α is the under-relaxation factor and the superscript (n-1) denotes the previous iteration level.

In order to compute the mass fluxes through the cell face, it is necessary to calculate the velocities at the cell face locations. The momentum equations for the Cartesian velocity components at the cell faces, for example at the east face, can be written as follows;

$$(u_i)_e = (H_{u_i})_e + (D^1_{u_i})_e(p_P - p_E) + (D^2_{u_i})_e(p_{se} - p_{ne})$$
$$+ (D^3_{u_i})_e(p_{be} - p_{te}) + (1-\alpha)(u_i^{n-1})_e \qquad (5)$$

In the present modified Rhie and Chow's scheme, these cell face Cartesian velocity components are obtained through the interpolation of momentum equations for the neighbouring cell centered, Cartesian velocity components. Following assumptions are introduced to evaluate these cell face velocities, for example u_i at the east cell face.

$$1/(A_P^{u_i})_e = f_e^+/(A_P^{u_i})_E + (1 - f_e^+)/(A_P^{u_i})_P \qquad (6)$$

$$(H_{u_i})_e = f_e^+ (H_{u_i})_E + (1 - f_e^+)(H_{u_i})_P \qquad (7)$$

where f_e^+ is a geometric interpolation factor.

In the present study, the coupling between the continuity and the momentum equation is effected using the SIMPLE algorithm [3]. The strongly implicit procedure is used to solve the algebraic equations. The computer code is made so that the inlet, outlet, wall, symmetry boundary conditions and the internal blockages are treated automatically. For turbulent calculations the k-ε turbulence model is used.

3. APPLICATIONS TO TEST PROBLEMS

The present code based on the modified Rhie and Chow's scheme is applied to the several test problems to assess its capabilities. In this paper only the solutions of laminar flow through a square duct with 90 degree bend are presented. This particular problem was studied experimentally by Humphrey et al. [4]. The Reynolds number based on the hydraulic diameter and the bulk velocity is 790. Only a symmetric half of the solution domain is solved to reduce the computer storage requirement. The nonuniform 71*41*21 grids are generated within the solution domain in which the inlet boundary is located at the 7 hydraulic diameter upstream of the curved section and the oulet boundary is located at the 10 hydraulic diameter downstream of the curved section. The fully developed laminar velocity profiles are prescribed at the inlet and the vanishing streamwise gradient conditions are imposed at the exit.

Fig.1 shows the streamwise velocity profiles at the various streamwise stations. At each station, the streamwise velocity profiles are plotted along two lines, z/b=0.5 and z/b=0.25. The x_h in the figures denotes the upstream distance from the initial curved section (θ=0). The computed results are fairly well agree with the experimental measurements by Humphrey et al. [4] although some differences are observed at the θ=60 station. The predicted secondary flow patterns in the curved section are shown in Fig.2. As the flow enters the curved section, the centrifugal force induces a strong secondary flow to move the fluid to the outside wall as shown in Fig.2-(a) for the θ=30 station. As the flow approaches further downstream, the strong vortex core moves to the inside wall (θ=60) by pressure gradient and it brings the second maximum high velocity flow near the inside wall. A very complicated secondary flow pattern is established at the θ=90 station. The primary vortex is evolved into two vortices and very small vortices are begun to form near the symmetry line. We can notice that these predicted secondary flow patterns are consistent with the trends of the predicted streamwise velocity distributions shown in Fig.1.

4. CONCLUSIONS

A three-dimensional calculation procedure for incompressible flows in complex geometries is presented. The scheme is based on the finite volume method and the Rhie and Chow's scheme [2]. The capabilities of the present code are demonstrated through its applications to the benchmark test problem. The results of the test problem show that the present code can be successfully applied to the analysis of fluid flow and heat transfer in complex geometries.

5. REFERENCES

1. K. C. Karki, 'A Calculation Procedure for Viscous Flows at All Speeds in Complex Geometries', Ph.D. thesis, University of Minnesota, Minneapolis, 1986.

2. C. M. Rhie and W. L. Chow, 'Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation', AIAA J., Vol. 21, No. 11, 1525-1532, 1983.

3. S. V. Patankar, Numerical Heat Transfer and Fluid Flow, McGraw-Hill, New York, 1980.

4. J. A. Humphrey, A. M. K. Taylor and J. H. Whitelaw, 'Laminar Flow in a Square Duct of Strong Curvature', J. Fluid Mech., Vol. 83, Part 3, 509-527, 1977.

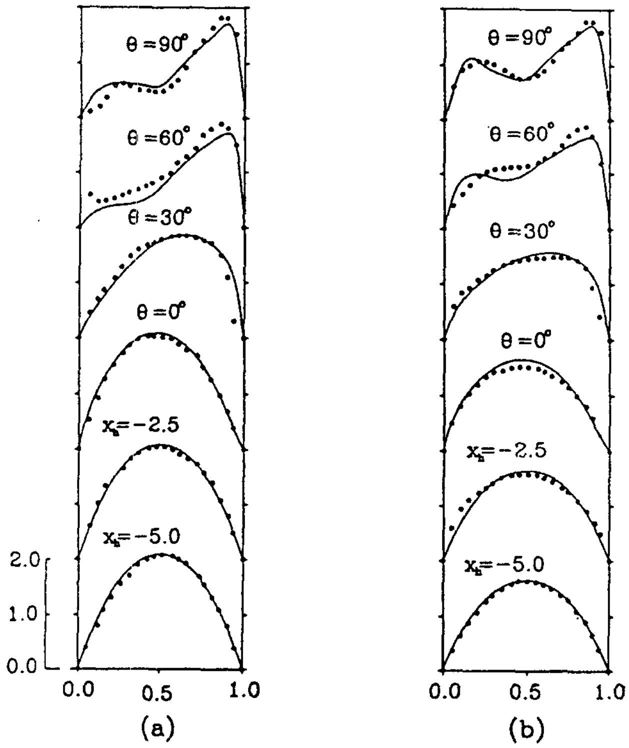

Fig.1 Streamwise velocity profiles: (a) $z/b = 0.5$; (b) $z/b = 0.25$.

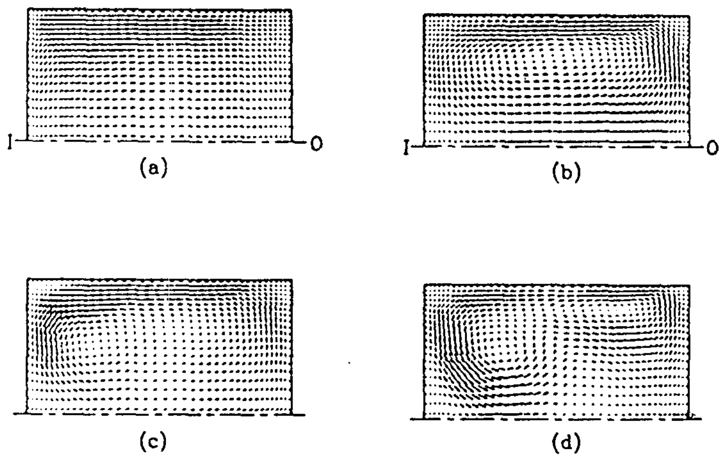

Fig.2 Secondary flow distributions: (a) $\theta = 0$; (b) $\theta = 30$; (c) $\theta = 60$; (d) $\theta = 90$.