ANALYSIS OF THE TAYLOR VORTEX FLOW

Tadashi HODOHARA , Junichi MATSUMOTO and Mutsuto KAWAHARA
Civil Engineering, Chuo University, Tokyo, Japan

INTRODUCTION

Recently, numerical simulation of complex flow in multi-scale structures plays an important role in engineering problems. The turbulence is a common example. The viscous flow in the annular space between two concentric rotating circular cylinders is a very interesting shear flow without a pressure gradient in the direction of mean flow. There is primarily the case that the inner cylinder is rotated and the outer is at rest. In this flow, a basic laminar axisymmetric flow is known as the Couette flow. When the rotation speed of the inner cylinder is increased beyond a certain critical value, the Couette flow becomes unstable. The instability leads to the transition to a laminar cellular vortex flow, referred to as the Taylor vortex flow.

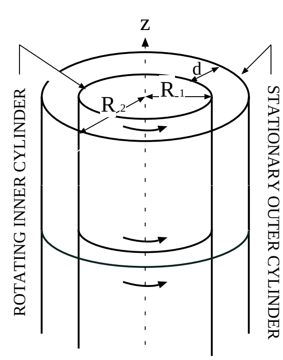


Fig. 1. Two concentric rotating circular cylinders

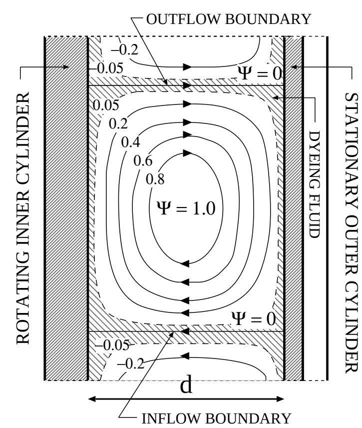


Fig. 2. Cross-section streamline of Taylor vortices

The viscous flow in the annulus between two concentric cylinders with the inner one rotating is of great importance not only in mechanical and chemical engineering but also in fluid physics because the flow may offer the key to an understanding of the transition-to-turbulence problem.

BASIC EQUATION

The basic equation of three-dimensional, incompressible viscous flow is described by the Navier-Stokes equations. The momentum equation and the continuity equation can be written as follows;

$$\dot{u}_i + u_j u_{i,j} + p_{,i} - \nu u_{i,jj} = f_i, \quad \text{in} \quad \Omega, \tag{1}$$

$$u_{i,i} = 0, \quad \text{in} \quad \Omega, \tag{2}$$

where the velocity and pressure are denoted by u_i and p.
The boundary conditions are;

$$u_i = \hat{u}_i, \quad \text{on} \quad \Gamma_1, \tag{3}$$

$$t_i = \{-p\delta_{ij} + \nu u_{i,j}\} n_j = \hat{t}_i, \quad \text{on} \quad \Gamma_2. \tag{4}$$

where $\hat{}$ means the given value and t_i is the flux on the boundary, in which n_i denote the outward normal on the boundary.

TEMPORAL DISCRETIZATION

A Crank-Nicolson method is applied to the momentum equation (1) is discretized by the full implicit scheme for the discretization in time, and the continuity equation (2) is expressed implicitly as follows;

$$\frac{u_i^{n+1} - u_i^n}{\Delta t} + u_j^* u_{i,j}^{n+\frac{1}{2}} + p_{,i}^{n+1} - \nu u_{i,jj}^{n+\frac{1}{2}} = f_i, \tag{5}$$

$$u_{i,i}^{n+1} = 0, \tag{6}$$

where

$$u_i^* = \frac{1}{2}\left(3u_i^n - u_i^{n-1}\right), \tag{7}$$

$$u_i^{n+\frac{1}{2}} = \frac{1}{2}\left(u_i^{n+1} + u_i^n\right), \tag{8}$$

u^* is the linear approximation of advection velocity is given by the Adams-Bashforth formula which has second order accuracy. Thus, as for the temporal direction, this is the linear scheme which has second order accuracy.
In this scheme, the fractional step method is applied to obtain the Pressure-Poisson equation. The pressure p^{n+1} in eq.(5) is approximated by p^n in the previous time step, and the unknown velocity u^{n+1} is replaced by the intermediate velocity $\tilde{u}^{n+1}$ which may not satisfy eq.(6). Thus, eq.(5) can be described as follows;

$$\frac{\tilde{u}_i^{n+1} - u_i^n}{\Delta t} + u_j^* \tilde{u}_{i,j}^{n+\frac{1}{2}} + p_{,i}^{n} - \nu \tilde{u}_{i,jj}^{n+\frac{1}{2}} = f_i, \tag{9}$$

where

$$\tilde{u}_i^{n+\frac{1}{2}} = \frac{1}{2}\left(\tilde{u}_i^{n+1} + u_i^n\right). \tag{10}$$

By taking the difference between eq.(5) and eq.(9), the following equation can be obtained;

$$\frac{u_i^{n+1} - \tilde{u}_i^{n+1}}{\Delta t} + \frac{1}{2}u_j^*\left(u_{i,j}^{n+1} - \tilde{u}_{i,j}^{n+1}\right) + \left(p_{,i}^{n+1} - p_{,i}^n\right) - \frac{1}{2}\nu\left(u_{i,jj}^{n+1} - \tilde{u}_{i,jj}^{n+1}\right) = 0. \tag{11}$$

By taking the divergence of eq.(11), substituting eq.(6) into eq.(11) and omitting the 2nd and 4th order terms of eq.(11), the Pressure-Poisson equation can be obtained as follows;

$$\Delta t\left(p_{,ii}^{n+1} - p_{,ii}^n\right) = \tilde{u}_{i,i}^{n+1}. \tag{12}$$

The algorithm of this scheme is described as follows;

1) Assume initial velocity $u_i^{(0)}$ and $p^{(0)}$.
2) Compute $\tilde{u}_i^{n+1}$ by eq.(9).
3) Compute p^{n+1} by eq.(12).
4) Compute u_i^{n+1} by eq.(11).
5) Go to 2).

In a certain problem which converges to the steady state, as the intermediate velocity $\tilde{u}$ converges to the real velocity u, the difference between p^{n+1} and p^n is zero. Therefore, the continuity condition doesn't depend on the time increment Δt in this scheme, because the Pressure-Poisson equation (12) converges to eq.(6) in the steady state problem.
About the spatial discretization, in case of the equal-order interpolation, the solutions tend to be unstable because the stabilized term is not used in this scheme. Thus, it's necessary to introduce the mixed interpolation.

SPATIAL DISCRETIZATION

The mixed interpolation for velocity and pressure fields is applied for the spatial discretization based on the MINI element.
The bubble function element is used for the velocity, which can be expressed as;

$$\begin{aligned} u_i &= \Phi_\alpha u_{i\alpha} \\ &= \Phi_1 u_{i1} + \Phi_2 u_{i2} + \Phi_3 u_{i3} + \Phi_4 u_{i4} + \Phi_5 \tilde{u}_{i5}, \end{aligned} \tag{13}$$

$$\tilde{u}_{i5} = u_{i5} - \frac{1}{4}\left(u_{i1} + u_{i2} + u_{i3} + u_{i4}\right), \qquad \Phi_5 = \phi_e,$$

and the linear element is used for the pressure as;

$$p = \Psi_\lambda p_\lambda = \Psi_1 p_1 + \Psi_2 p_2 + \Psi_3 p_3 + \Psi_4 p_4, \tag{14}$$

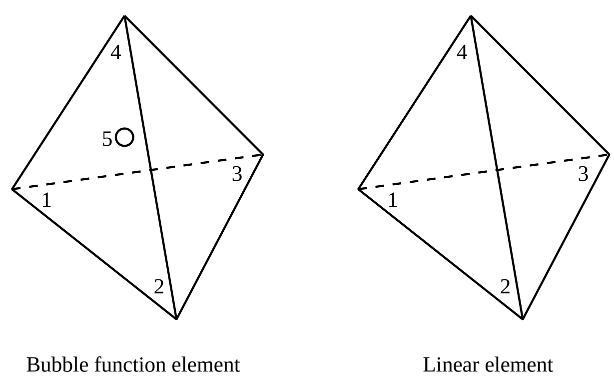


Fig. 3. MINI element

where ϕ_e is the bubble function of C^0 continuous and $\Phi_\alpha(\alpha = 1 \sim 5)$ is the bubble function element for velocities in five-node tetrahedral element, $\Psi_\lambda(\lambda = 1 \sim 4)$ is the linear interpolation for pressure in four-node tetrahedral element and $u_{i\alpha}$ and p_λ represent the nodal values at the α^{th} node of finite elements. The corresponding weighting function is similar to eqs.(13) and (14) is used.

NUMERICAL EXAMPLE

Fig.4 shows the simplified analytical domain of the flow to be used for the numerical computation. The small annular gap width is denoted by d. About this model, the radius of inner circular cylinder is 7.3195 and that of outer circular cylinder is 8.3195. The height of a circular cylinder is denoted by $h = 4$. In Fig.5, The finite element mesh is divided into 90,200 nodes and 480,000 tetrahedral elements. The mesh contains 10 elements in the radial direction, 200 elements in the circumferential direction and 40 elements in the axial direction. The boundary conditions are given as follows;

$$\begin{aligned} &U = 1, \quad w = 0, && \text{on the inner cylinder,} \\ &u = v = w = 0, && \text{on the outer cylinder,} \\ &u = v = w = 0, && \text{on the top and bottom} \\ & && \text{of the cylinders.} \end{aligned}$$

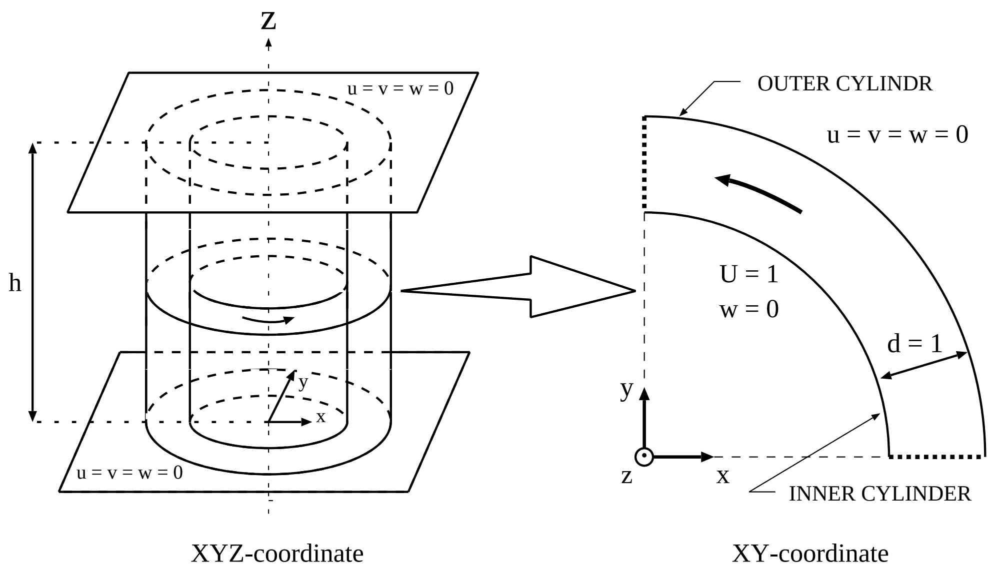


Fig. 4. Analytical domain

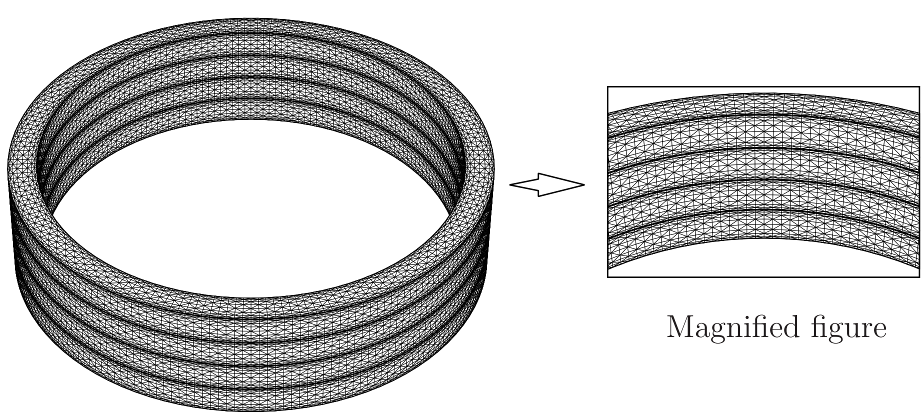


Fig. 5. Finite element mesh : 90,200 nodes 480,000 tetrahedral elements

CONCLUSION

In this study, the finite element analysis of the phenomena of the viscous flow in the annular space between two concentric rotating circular cylinders with inner one rotating in three-dimensional model is presented which is an example of the Taylor vortex flow. The Taylor vortex flow in three-dimensional model will be solved by the present method.

REFERENCES

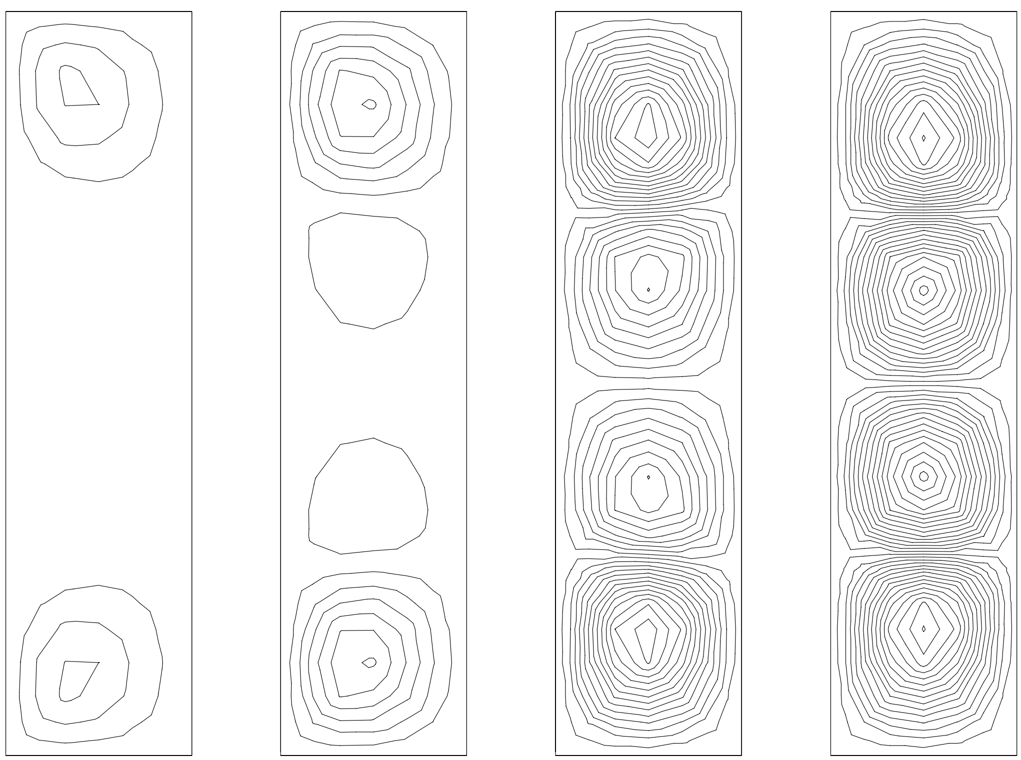
[1] Kunio Kataoka: Taylor Vortices and Instabilities In Circular Couette Flows.

[2] Tomomasa Tatumi: Hydrodynamics, pp350-363, 1982.

[3] Tutomu Kanbe, P.G.Drazin: Fluid Dynamics Stability and Turbulence, pp78-86, 1998.

[4] Junichi Matsumoto and Tsuyoshi Umezu: The Examination of Explicit Finite Element Analysis Using Bubble Function, 1995.

[5] Junichi Matsumoto, Tsuyoshi Umetsu and Mutsuto Kawahara: Incompressible Viscous Flow Analysis and Adaptive Finite Element Method Using Linear Bubble Function, 1999.



(a) T = 20 (b) T = 30 (c) T = 50 (d) T = 75

Fig. 6. Numerical Result Streamline ($Re = 200$)