# Finite-Element Analysis of Unsteady Flow Past a Circular Cylinder Based on a Variational Multiscale Method

M. Hashiguchi[1]
1. Keisoku Engineering System Co., Ltd., 1-9-5 Uchikanda, Chiyoda-ku, Tokyo, Japan

## Introduction

Fluid can take any flow state of laminar, transition or turbulent, and the state taken strongly affects the magnitude of fluid force occurred on an object or the mixing state of the flow field. Therefore, CFD which can treat these flow states automatically, is strongly desired. DNS and implicit DNS can treat flow transition as well as laminar and turbulent, but these need huge resources for the flow computation. LES has been developed to reduce computational resources, but the classical Smagorinsky model is too dissipative to treat laminar and transitional flow. LES based on the spatial filtering techniques also has a problem of non-commutative operator. Without such spatial filtering techniques, the variational multiscale method has been proposed and developed.

This paper conducted incompressible flow computations based on the variational multiscale method and reports some numerical results of various incompressible flow fields, including flow past a circular cylinder immersed in a channel between parallel plates. Finite-element analysis of the incompressible Navier-Stokes equations based on a residual-based variational multiscale method was performed. In the present actual computation, the residual-based variational multiscale (RBVM) model[1)] which has been implemented in the latest version of COMSOL Multiphysics® Ver.5.4 was utilized here.

## Method of Approach

### Governing Equations

The incompressible Navier-Stokes equations with the continuity equation are the governing equations, which is displayed below, to be solved to obtain the primitive flow variables of pressure p and velocity vectors **u,** with the no-slip boundary condition and the initial condition.

$$\frac{\partial \boldsymbol{u}}{\partial t} + \nabla \cdot (\boldsymbol{u} \otimes \boldsymbol{u}) + \nabla p = \nu \Delta \boldsymbol{u}$$

$$\nabla \cdot \boldsymbol{u} = 0$$

where ν is the kinematic viscosity.

### RBVM

The variational multiscale method has been proposed by Hughes (1995) [2)]. His formula is expressed in terms of the classical Green's function and the projector which defines the decomposition of the solution into coarse and fine scales. Bazilevs et al.[3)] developed an LES-type variational theory of turbulence, where any ad hoc devices, such as eddy viscosities, are not employed. In order to compute the fine-scale field, the element-wise stabilization operator τ is computed from the formula for the fine-scale Green's operator, and is represented as the product of τ and the local coarse-scale residual. This formulation has a remarkable character: if the coarse-scale (i.e., grid scale) is fully resolved, the fine-scale model becomes zero. This means this model is consistent.
All of computations here is executed based on the three-dimensional and time-dependent study.

## Simulation Results and Discussion

### Hele-Shaw Flow

When the Reynolds number, Re, of flow field is quite low as Re~1, the viscous flow sandwiched by two parallel end plates which are set at very short distance, can be considered as potential flow field, as explained by Hele-Shaw experimentally. Figure 1 shows the present computation. This quite resembles the experimental flow visualization which is shown by van Dyke [4)] in his famous "an album of fluid motion".

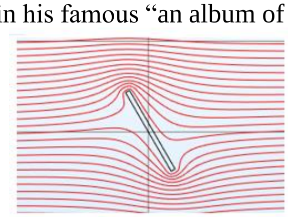


**Figure 1. The present computation for Hele-Shaw flow.**

**Figure 2. van Dyke's flow visualization for Hele-Shaw flow[4)].**

**Low Reynold number flow of Circular cylinder**

In the history of CFD the epoch-making work was done by Prof. Kawaguchi[5]. He completed the computation of circular cylinder flow of low Reynolds number Re=40 and obtained the result shown in Fig.3, by using mechanical hand calculator (Tiger calculator developed by Japanese), spending one and a half years with 20 hours a week. His computation is executed by the two-dimensional steady computation.

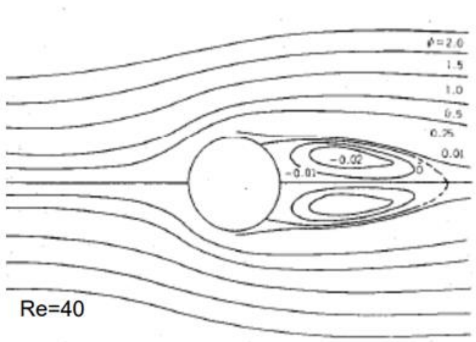



**Figure 3**. **Prof. Kawaguchi's computation results[5].**

Figure 4 shows the present result which is obtained by the three-dimensional unsteady flow computation after the dimensionless time of 100. It took only 8 hours. The resultant flow field is two-dimensional and steady state, and agrees with the two-dimensional flow assumption of Prof. Kawaguchi. The present computation experienced the wake behind the cylinder gradually developed and took a lot of time in order to develop its size toward the downstream.

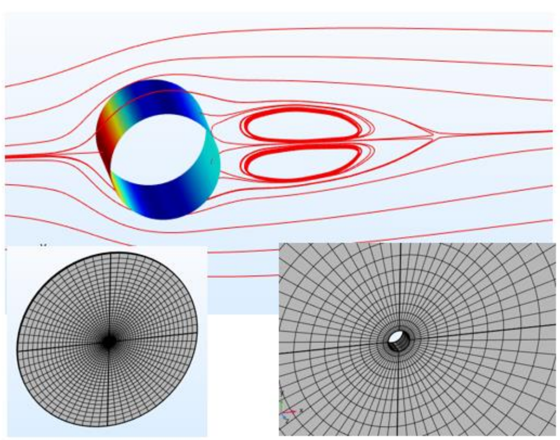


**Figure 4**. **The present computation for cylinder flow of Re=40 at dimensionless time of 100.**

**Lid-Driven Cavity Flow**

As a famous benchmark flow problem, there is a lid-driven cavity flow and finite-difference solutions has been reported by Ghia et al [6]. Based on this benchmark problem, cavity flow fields having various Reynolds numbers were examined in the present study. Figure 5 shows the present result of Re=1000, with the Reynolds number based on the wall speed of the ceiling and a side length of the cavity. After dimensionless time of 50, using mesh size of 32×32×2 (equi-distributed), the present streamlines, the velocity components (u,v) approaches to Ghia's reference solution ,as time goes.

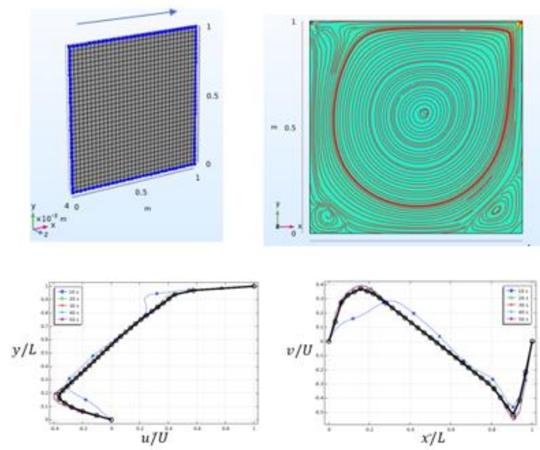
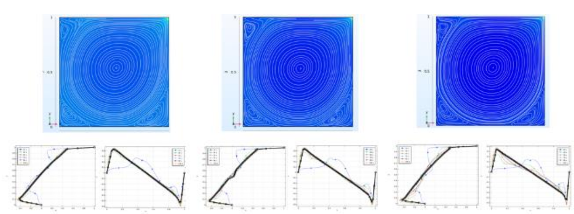



**Figure 5**. **The present computation for lid-driven cavity flow of Re=1000 at dimensionless time up to 50; Mesh of 32×32×2 (equi-distributed).**

Figure 6 shows the present result of Re=3200, 5000 and 7500. Mesh size is 64×64×2 (equi-distributed). Each case approaches to Ghia's reference solution as time goes.



(a) Re=3200 (b) Re=5000 (c) Re=7500

**Figure 6**. **The present computation for lid-driven cavity flows of Re=3200 (a), 5000 (b) and 7500 (c) at dimensionless time up to 100; Mesh of 64×64×2 (equi-distributed).**

For Re=10000, firstly equi-distributed mesh system was featured, but the convergence to Ghia's solution was not sufficient. Therefore, stretched mesh system was featured and the result is shown in Fig.7. The present solution has a tendency to approaches to the solution of Ghia et al. as time goes up to 100, although it still has a small discrepancy in detail.

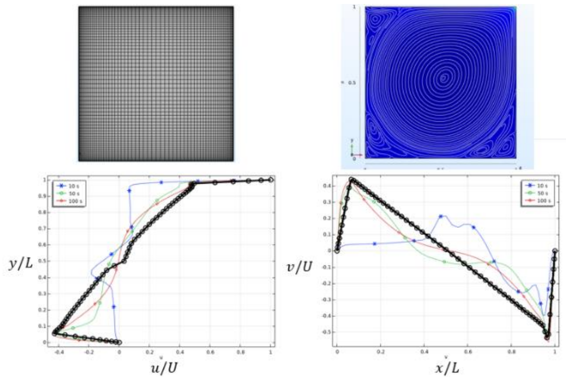



**Figure 7. The present computation for lid-driven cavity flows of Re=10000 at dimensionless time up to 100; Mesh of 64×64×2 (stretched-type).**

## Flow past a Circular Cylinder in a Channel

Flow past a circular cylinder in a channel is frequently observed in practical flow application. It is well known that flow fields developed around a body is influenced from the surrounding channel and it is called blockage effect. Comparing with free flow developed in free space, channel accelerates the flow near the objects such as circular cylinder.

The present computation of the Reynolds number 3900, which is defined by the free upstream velocity and the diameter of the circular cylinder.
The resulting pressure is displayed in Fig.8.

In Fig. 9, the instantaneous streamlines (the envelopes of the instantaneous velocity vector fields) are also shown. Two-dimensionality of flow field is kept upstream the circular cylinder, while three-dimensional flow is developed behind the circular cylinder as well as pressure fields shown in Fig.8.

Maskell proposed the correction formula in closed-wind tunnel, as shown in Fig. 10. The present coefficient of drag shows 1.3. The corresponding measurement is around 0.98. As in Fig.10, it was found that the corrected value coincides with the measurement value.

## Flow past a Semi-Sphere on the Ground

The last computation is performed for three-dimensional flow past a semi-sphere on the ground plate. This flow field is strongly influenced from the ground plate. Figure 11 shows the present result at the dimensionless time 100. The Reynolds number, defined by the radius of the semi-sphere and the free stream velocity, is 40000. The experiment has been conducted by Fedrizzi et al.[8] in wind tunnel of the blockage ratio which is lower than the one in the present computation. By comparing with their results, horseshoe vortex resembles the experiment (Fedrizzi et al., Figure 2 (a) and (b)[8]), but the separation line in the present paper moved downstream than the experiment. It seems plausible because of the stronger flow acceleration in the present computation than the experimental one of small blockage effect.

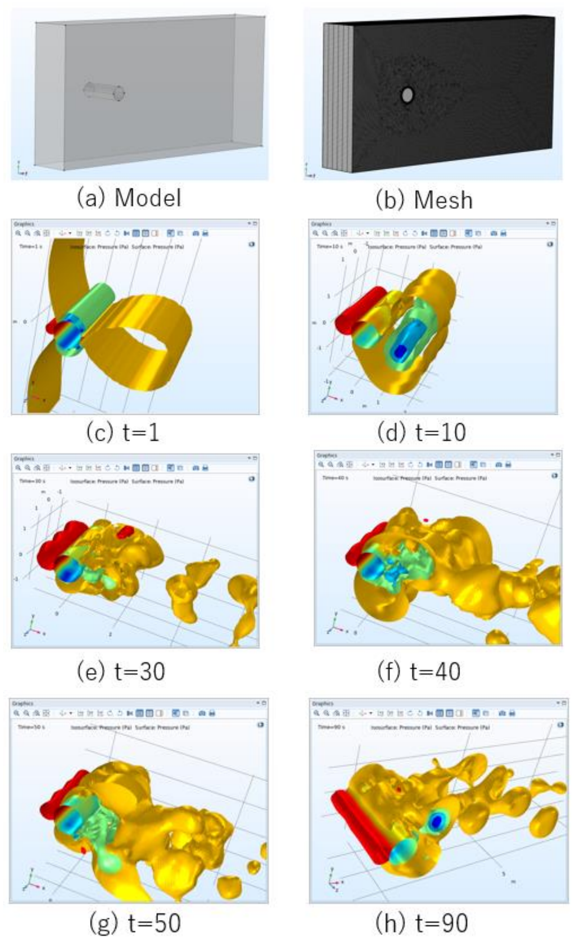



**Figure 8. Computational model, mesh system and the time history of time-dependent pressure field around a circular cylinder in a channel; Re=3900.**

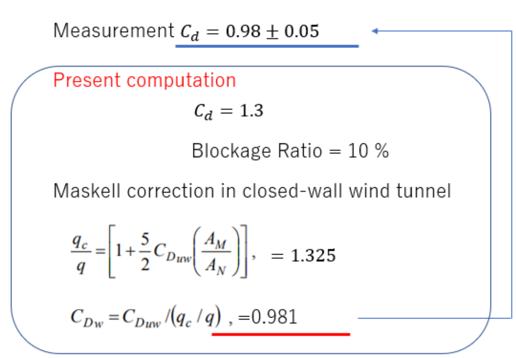



**Figure 10. Maskell's formula and the resulting corrected value of the coefficient of drag of circular cylinder.**

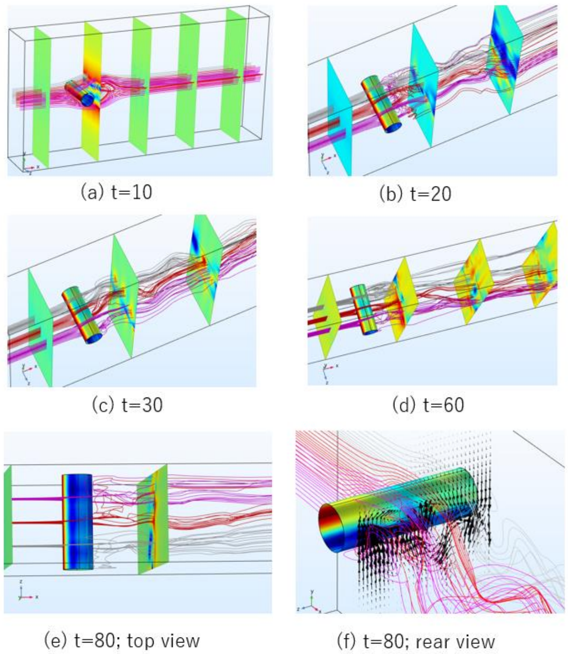



**Figure 9. The time history of instantaneous streamlines around a circular cylinder in a channel; Re=3900.**

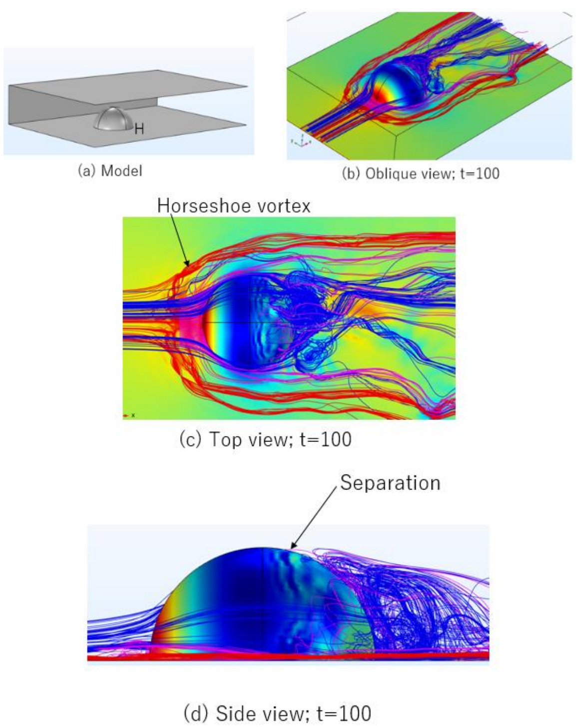



**Figure 11. Computational model and results; the instantaneous streamlines and surface pressure of semi-sphere and ground plane at dimensionless time 100.**

## Conclusions

Variational multiscale method based on RBVM, which has been implemented in COMSOL Multiphysics® Ver.5.4, was applied to various flow fields with the wide range of the Reynolds numbers. It seems to work well, but it took a long computing time, depending on the mesh size to be used.

Based on the present study, it is expected that the RBVM of COMSOL Multiphysics®Ver.5.4 will discover important features of fluid dynamics, in the wide range of the flow Reynolds number, including laminar, transition and turbulent flow.

## References


1. Mats Nigam, The Basics of CFD Simulation with COMSOL® in 18 Minutes, https://www.comsol.jp/video/intro-to-cfd-simulation-with-comsol-multiphysics.
2. T.J.R. Hughes, Comput. Methods Appl. Mech. Engrg.127 (1995), pp387-401.
3. Y. Bazilevs et al., Comput. Methods Appl. Mech. Engrg. 197 (2007), pp.173-201.
4. van Dyke, An Album of Fluid Motion.
5. Kawaguchi, M., J. Phys. Soc. Jpn.8 (1953).
6. Ghia, U. et al., Journal of Computational Physics 48(1982), pp.387-411.
7. Maskell, E.C., ARC R&M 3400, Nov. (1963).
8. Fedrizzi, M. et al., 18th Australasian Fluid Mechanics Conference, Australia 3-7 Dec. (2012).