Implicit LES for Two-Dimensional Circular Cylinder Flow by Using COMSOL Mutiphysics®

M. Hashiguchi
Keisoku Engineering System Co., Ltd.
1-9-5 Uchikanda, Chiyoda-ku, Tokyo, 101-0047 Japan

Abstract: In this paper, implicit Large Eddy Simulation(Implicit LES) based on COMSOL Multiphysics has been utilized to simulate drag crisis phenomenon on circular cylinder flow.

Implicit LES means that it uses the full Navier-Stokes equations instead of Reynolds Averaged Navier-Stokes equations (RANS) and does not use any explicit turbulence model. For execution of implicit LES with reduced number of mesh system, we need some numerical stabilization technique which may introduce an amount of implicit turbulence model. We used here both techniques of streamline stabilization and cross wind stabilization which are prescribed in COMSOL Multiphysics. Here, unsteady two-dimensional incompressible flow past a circular cylinder with/without surface roughness was investigated numerically based on implicit LES, and it was found that the drag crisis in high Reynolds number flow could be reproduced. On the case of the circular cylinder with surface roughness, it was also found that the critical Reynolds number becomes one-order lower than that of smooth surface and it coincides with experiment. When this implicit LES is applied for fluid flow simulation because there is ability to automatically capture flow transition, the present results suggest an advantage of multiphysics simulation using COMSOL Multiphysics.

Keywords: Implicit LES, Circular cylinder flow, Drag crisis, Finite-element analysis, COMSOL Multiphysics.

1. Introduction

In the flow past a circular cylinder when the Reynolds number increases, it exhibits some interesting phenomena such as steady symmetric flow pattern of twin vortices, generation of unsteady flow pattern, Hopf bifurcation, Kármán vortex street, the transition from laminar flow to turbulent flow, and the decreasing of the drag coefficient called by drag crisis. Flow field around the circular cylinder can be controlled by the surface conditions such as surface roughness, which is applied e.g., to the reduction of the critical Reynolds number with a tripping wire in wind tunnel experiment and to the elongation of the flying distance of a golf ball with surface dimple. Computer simulation which includes fluid flow physics at high Reynolds number, therefore, must reproduce flow transition from laminar to turbulent and resolve the influence of surface roughness.

In 1984, Kawamura and Kuwahara[1] presented computations at high Reynolds number of the range from 10^3 to 10^5 for flow around circular cylinder with surface roughness. By introducing a new third-order upwind finite-difference scheme, they solved the incompressible Navier-Stokes equations without incorporating a turbulence model. Their computations were the first to reproduce the drag crisis phenomenon. Tamura and Kuwahara[2] performed two-dimensional flow computation and three-dimensional flow computation for the smooth surface circular cylinder with no turbulence model and they found that three-dimensional computation showed a very good agreement with the experimental data in both the subcritical and supercritical flow regime. In 1985, Ishii et al.[3] had simulated a two-dimensional flow past a circular cylinder based on a high accurate scheme for compressible flow, showing quantitatively good agreement in the prediction of the drag coefficient around the critical flow regime at a Mach number equals to 0.3. Hashiguchi and Kuwahara[4] also executed two-dimensional flow computation for a smooth surface circular cylinder at the Reynolds number of the wide range from 0.1 to 10^6 based on the multi-directional upwind finite difference scheme which is proposed by them, and they showed its accurate prediction of the drag crisis and the separation points even though they used no explicit turbulence model. Kakuda et al.[5] presented the Petrov-Galerkin finite element scheme using exponential weighting functions for solving effectively incompressible Navier-Stokes equations and showed that their three-dimensional flow computation around a circular cylinder within the range of Reynolds number from 10^3 to 10^6 agrees with other existing data. The above introduced computations can be entered into the category of implicit LES. Although the possibility of implicit LES based on finite-element analysis by using a commercial code of finite-element analysis, COMSOL Multiphysics, has already been discussed in the two-dimensional lid-driven cavity flow up to the Reynolds number of 10^6 by Hashiguchi[6], it has not been applied to the flow field of circular cylinder.

Based on these computations, where explicit turbulence model was not introduced, we can observe that two-dimensional computation of the drag crisis of circular cylinder flow is significant for the validation of numerical scheme without no turbulence model, although the resulting drag coefficient around the critical flow regime takes higher value due to the assumption of two-dimensionality.

In the present paper, therefore, two-dimensional flow past a circular cylinder with smooth surface and surface roughness is investigated numerically based on implicit LES, by using COMSOL Multiphysics[7][8]. A wide range of the Reynolds number of 0.1 to 10^6 was examined here.

2. Method of Approach

Newtonian fluids governed by the incompressible Navier-Stokes equations of motion are considered.

We solve the full Navier-Stokes equations and the continuity equation:

$$\rho(\partial_t u^i + u_j \partial_j u^i) = -\partial_i p + \mu \partial_k \partial_k u^i$$

$$\partial_i u^i = 0$$

where t is the time, and u^i(i=1,2) the flow velocity vector components in the Cartesian coordinate system x^i, p the pressure, ρ the density, μ the viscousity coefficient. The Reynolds number is defined based on the diameter of circular cylinder d as Re=(ρ u_0 d)/μ, where u_0 is the upstream velocity.

Computational region solved is the square region shown in Fig.1, whose side length is 60d, and a circular cylinder is immersed at 15d from the left boundary along the center line. This computational region has a sufficient size because the blockage ratio d/H=1/60, H being the height, is so small that the blockage effect can be ignored (The blockage correction may be around 1% for the circular cylinder flow of the aerodynamic coefficient of drag of 1.2, which can be estimated by the blockage correction formula (2) of Roshko [9]). We investigated two types of circular cylinder: smooth surface and partially rough surface as shown in Fig. 2, where saw-tooth like grooves are set as a surface roughness.

As the boundary conditions for velocity field, uniform flow condition on the left-side inflow boundary and the no-slip condition on the surface of the circular cylinder were set. Slip flow condition was set on both top boundary and bottom boundary. Since the flow field considered here is incompressible, pressure p (gauge pressure) was set to zero on the right-side outflow boundary.

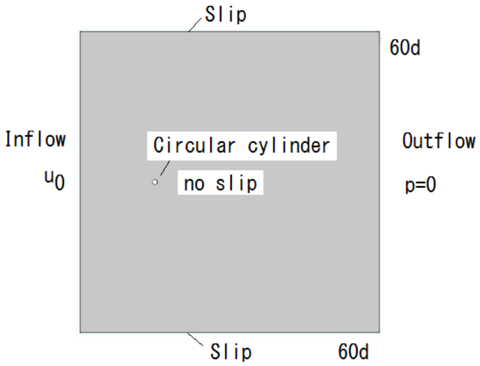


Figure 1. Computational domain.

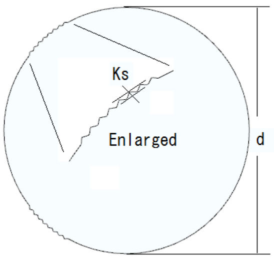


Figure 2. Circular cylinder and definition of surface roughness.

3. Stabilization Technique of COMSOL Multiphysics

The governing equations were discretized based on finite-element analysis and were solved numerically by using COMSOL Multiphysics Ver.4.3b. In principle, a very dense mesh resolution beyond which the discretization is stable is there, but in practical it is not feasible to use such huge number of mesh system so that the addition of artificial diffusion is required to reduce a number of mesh system. For this, we utilized streamline diffusion and crosswind diffusion prescribed in COMSOL Multiphysics. The streamline diffusion stabilization used here is Galerkin Least Squares(GLS) but without any viscous terms in the test operator in the stabilization term. [10] The crosswind diffusion addresses spurious oscillations which can occur in the numerical solution for the case where sharp gradients are there. [10]

In the numerical computation of circular cylinder flow in high Reynolds number flow regime, we have to resolve thin boundary layer flow to be developed along the wall surface and its separation on the wall surface and the resulting vortex shedding into the main flow field. For solving this, boundary mesh technique which is prescribed in COMSOL Mutliphysics was utilized and 8 boundary mesh layers with the minimum thickness of the first layer of $0.1d/Re^{0.5}$ were set on the wall surface of the circular cylinder. The resulting mesh system is shown in Fig. 3.

For two-dimensional lid-driven cavity flow, we have succeeded the implicit LES up to the Reynolds number of 1,000,000.[6]

4. Results and Discussion

Time-dependent computation was executed for the dimensionless time $(u_0/d)t$ from 0 to 100. Only for small Reynolds number from 0.1 to 40, steady flow computation is executed. This is valid from the experimental observation of the flow pattern reported by Thom [11].

The aerodynamic coefficient of drag Cd, which is defined as $C_d=D/(1/2Q\ d)$, where D is the drag force and Q the dynamic pressure of ρu_0^2, is computed for the dimensionless time up to 100. Typical time histories of Cd are displayed in Fig.4 (smooth surface) and Fig.5 (rough surface), and time averaging during the period of the dimensionless time from 50 to 100, is taken in order to obtain the time-averaged value of Cd. A wide range of Reynolds number, 0.1<Re<1e6, was studied here. (1e6 means 10^6, it is the same afterward.)

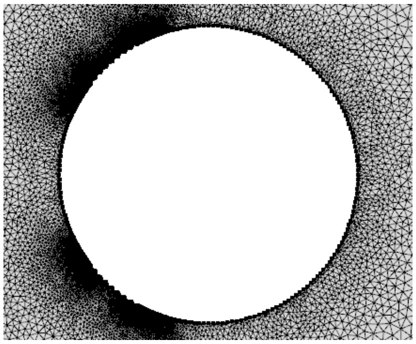
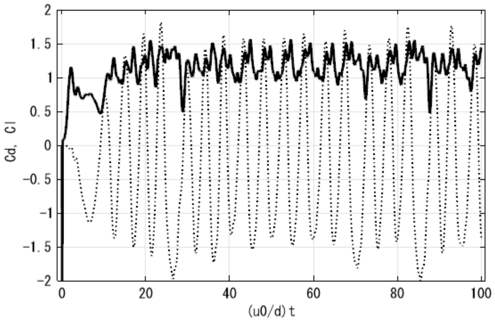
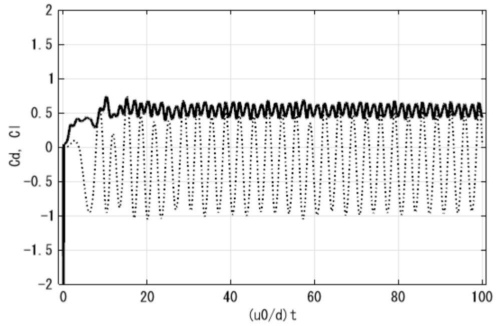


Figure 3. Finite element mesh system.



(a) Re=$1e^5$

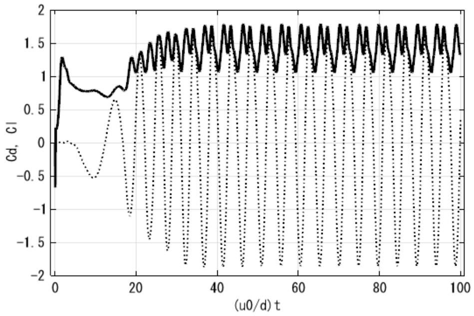


(b) Re=$1e^6$

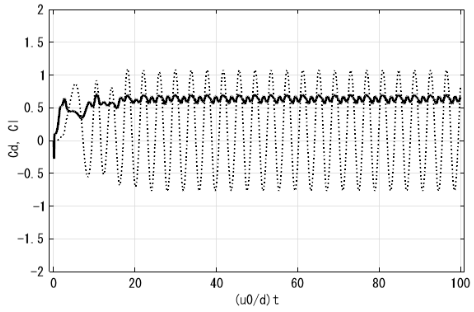
Figure 4. Time histories of Cd and Cl of smooth surface circular cylinder.

Solid circles(●) in Fig. 6 shows the dependence of time-averaged C_d predicted for the circular cylinder with smooth surface. The solid line shows the experimental results of Wieselsberger[12] for 10<Re<8e5. All the computations and the experiment of Wieselsberger[12] agree qualitatively, although the computation shows higher values for 1000<Re<10000. The present results also showed good agreement with the data[13] for Re<10. The decrease of Cd can be observed around Re=3e5. The Strouhal number St was obtained by using Fast Fourier Transformation analysis of the time history of the lift coefficient C_l=L/(1/2Q d), where L is the lift force. The result showed the increase of St occurred around Re=2e5. The predicted St was 0.22 at Re=1e5, 0.24 for 2e5=<Re<=3e5, 0.26 at Re=4e5 and 0.33 for 6e5 =<Re<=1e6. These values of St agrees well with the experimental data (0.2 at Re=1e5, around 0.22-0.23 for 2e5<Re<6e5 , 0.3 for 6e5 <Re< 8e5 and around 0.4 for Re>1e6 as shown in Roshko[9]).

Based on these results, it can be concluded that drag crisis of smooth surface circular cylinder was started at Re=2e5 within the accuracy of the present Reynolds number-sweep. Instantaneous flow streamlines at Re=1e5 and 1e6 are depicted in Fig.7(a) and Fig.7(b), respectively. It can be seen that the separation



(a) Re=1e4



(b) Re=1e5

Figure 5. Time histories of Cd and Cl for rough surface.

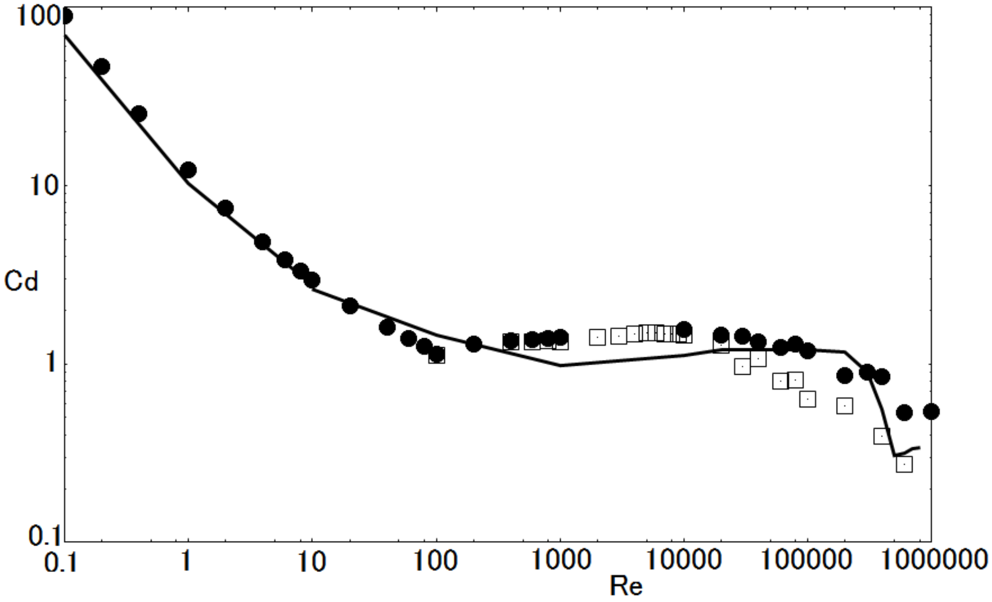


Figure 6. Drag coefficient versus Re.

point moved downstream at Re=1e6. These results agree with existing data.

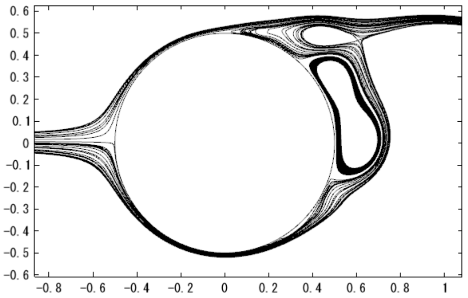
Similarly, in the case of circular cylinder with surface roughness(plot with □ in Fig.6), we can observe that drag crisis started at Re=6e4 because that the Strouhal number was changed from 0.22 to 0.24 and Cd was also reduced. The instantaneous flow streamlines at Re=1e4 and Re=1e5 are depicted in Fig.8(a) and Fig.8(b), respectively. Like the flow pattern of the smooth surface circular cylinder, the separation point moved downstream at Re=1e5.

Adachi [14] investigated the effect of surface roughness of critical Reynolds number of cylinder and found there was a limiting value of the Reynolds number at the critical point, that is, critical surface roughness, being the critical surface roughness of 0.0005. In the present study, the relative surface roughness Ks/d=0.008, Ks is the height of roughness, was used. Based on Fig.3 in his report, we can read the critical Reynolds number is around 5e4 for the present relative surface roughness. This agrees with the present prediction of 6e4.

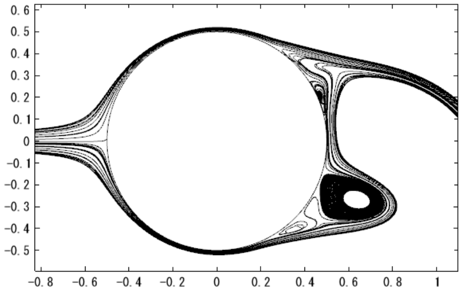
Hashiguchi and Kuwahara [4] was the first to capture the stationary deflected flow pattern and asymmetry location of the separation points at Re=5e5 in the case of smooth surface circular cylinder. The present computation also shows non-zero value of the time-averaged lift force as shown in Fig.4(b) as well as in Fig.5(b). This interesting phenomenon has also been reported experimentally by Kamiya et al. [15].

5. Concluding Remarks

The flow field around a circular cylinder was numerically investigated based on the implicit LES by using finite element method with numerical stabilization techniques of GLS and crosswind stabilization. COMSOL Multiphysics has been utilized in this study because it includes the full Navier-Stokes equations and GLS and crosswind stabilization technique. we solved the full Navier-Stokes equations without any explicit turbulence model. As a result, it was found that the present method can produce the essential features of the flow past a circular cylinder for a wide range of Re=0.1-1,000,000 including drag crisis. Although it was not shown here, we could solve the flow up to Re=10,000,000 and observed Cd=0.2348 and St=0.36.

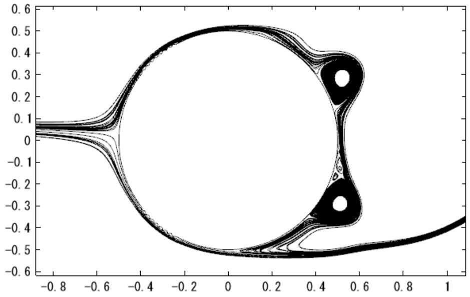


(a) Re=1e5

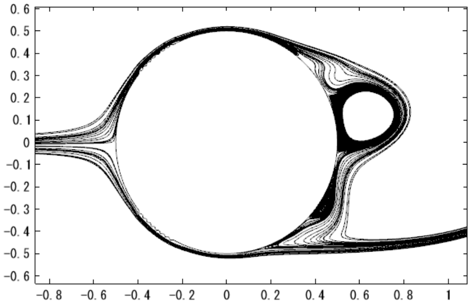


(b) Re=1e6

Figure 7. Instantaneous streamlines at $(u_0/d)t=100$; smooth surface.



(a) Re=1e4



(b) Re=1e5

Figure 8. Instantaneous streamlines at $(u_0/d)t=100$; rough surface.

When this implicit LES is applied for fluid flow simulation because there is ability to automatically capture flow transition, the present results suggest an advantage of multiphysics simulation using COMSOL Multiphysics.

On the other hand, to consruct a more quantitative prediction of Cd, three-dimensional implicit LES must be performed. To this end, more efficiency in memory consumption and high speed computation are strongly desired.

References

1. Kawamura,T. and Kuwahara,K., Computation of high Reynolds number flow around a circular cylinder with surface roughness, AIAA paper No.840340(1984).

2.Tamura,T. and Kuwahara,K., Direct Finite Difference Computation of Turbulent Flow around a Circular Cylinder, Num. Meths. Fluid Dyns., (Eds., Yasuhara,M. et al.), pp.645-650(1989).

3.Ishii,K. et al., Computational Flow around a Circular Cylinder in a Supercritical Regime, AIAA paper No.851660(1985).

4.Hashiguchi, M. and Kuwahara,K., Two-Dimensional Study of Flow Past a Circular Cylinder, RIMS Kokyuroku, vol.974, pp.164-169(1996).
http://www.kurims.kyoto-u.ac.jp/~kyodo/kokyuroku/contents/pdf/0974-16.pdf/

5.Kakuda, K. et al., Computational Flow Simulations around Circular Cylinders Using Finite Element Method, ICCES, vol.5, no.4, pp.199-204(2008).

6.Hashiguchi, M., Possibility of Implicit LES for Two-Dimensional Incompressible Lid-Driven Cavity Flow Based on COMSOL Multiphysics, COMSOL Conference Tokyo 2012(2012).
http://www.comsol.jp/paper/download/159355/hashiguchi_paper.pdf/

7. http://www.comsol.jp/products/

8.Pryor,R.W., Multiphysics Modeling Using COMSOL 4, Mercury Learning and Information(2012).

9.Roshko,A., Experiments on the flow past a circular cylinder at very high Reynolds number, J. Fluid Mech, vol.10(1960).
http://authors.library.caltech.edu/10105/1/ROSjfm61.pdf/

10. COMSOL_ReferenceManual

11.Thom,A., The Flow Past Circular Cylinders at Low Speeds, Proc. Roy. Soc. Lond. A, pp.651-669(1933).

12.Wieselsberger,C., Neuere Feststellungen uber die Gesetze des Flussigkeits –und Luftwiederstands, Phys. Z.22, pp.321-328(1921). In the present paper, data were reproduced from Roshko9).

13.Data reproduced from Hashiguchi and Kuwahara4).

14.Adachi, T., The Effect of Surface Roughness of a Body in the High Reynolds-Number Flow, Int. J. of Rotating Machinery, vol.2,no.1,pp.23-32(1995).

15.Kamiya,et.al., AIAA paper No.791457(1979).