The Publications of the XXVI. microCAD International Scientific Conference

ISBN: 978-963-661-773-8

University of Miskolc, Hungary
29-30 March 2012

ITTC
Innovation and Technology Transfer Centre
University of Miskolc

A felsőoktatás minőségének javítása kiválósági központok fejlesztésére alapozva a Miskolci Egyetem stratégiai kutatási területein (TÁMOP-4.2.1.B-10/2/KONV-2010-0001)

ÚJ SZÉCHENYI TERV

Nemzeti Fejlesztési Ügynökség
www.ujszechenyiterv.gov.hu
06 40 638 638

MAGYARORSZÁG MEGÚJUL

A projektek az Európai Unió
támogatásával valósulnak meg.

THREE-DIMENSIONAL COMPUTATIONS FOR LOW-REYNOLDS NUMBER FLOWS PAST A CYLINDER

Betti Bolló[1], Gábor Janiga[2], László Baranyi[3]
[1]Assistant lecturer, [2]Associate Professor, [3]Professor
[1,3]*Department of Fluid and Heat Engineering, University of Miskolc, Hungary*
[2]*University of Magdeburg "Otto von Guericke" (ISUT/LSS), Germany*

INTRODUCTION

The incompressible flow past a cylinder is an important engineering problem. Examples are chimney stacks, cables of suspended bridges, offshore structures and risers, and transmission lines, which are exposed to wind or ocean currents. Despite the simple geometry of a circular cylinder, the flow past a stationary circular cylinder is the baseline for more complex cases.

The steady twin recirculation vortices behind a stationary circular cylinder are susceptible to perturbations in the transverse direction and begin to shed downstream alternatively at a critical Reynolds number of approximately $\mathrm{Re}_c=47$ [1]. Reynolds number Re is defined as $\mathrm{Re}= u_\infty D/\nu$, where u_∞ is the free-stream velocity, D is the cylinder diameter and ν is the kinematic viscosity of the fluid. At Re > 47, the alternatively shed vortices develop into vortex rows, known as the von Kármán vortex street. The unsteady wake flow at this stage is also called laminar two-dimensional wake regime or laminar periodic wake regime. The flow remains two-dimensional (2D) up to $\mathrm{Re}\approx190$. Between Re=190 and 260, wakes of two-dimensional cylinders are observed to be susceptible to a primary instability mechanism which leads to the amplification of three-dimensional disturbances and eventually to the development of strong streamwise oriented vorticity structures [2].

In this paper the three-dimensional (3D) flow around a stationary circular cylinder at the Reynolds number of 200 is investigated numerically using two different CFD software packages. The main aim of the present study is to compare results from two very different CFD codes for exactly the same case, in order to further support previous findings and prepare further investigations with more complex configurations. The first author used the commercial software Ansys Fluent (Ansys Inc., Canonsburg, PA, USA), based on the finite volume method (FVM). The second author has applied the open-source OpenFoam (SGI Inc., Fremont, CA, USA), also based on a finite volume discretisation. After performing the numerical computations using both software packages, various flow properties such as drag and lift coefficients and Strouhal number are investigated.

NUMERICAL SOLUTION

For the numerical simulations two different computational domains are investigated. In one case the same computational domain as well as the same computational mesh is employed for both software packages (Ansys Fluent and OpenFoam) in order to facilitate an easier comparison. The schematic diagram of the rectangular computational domain is shown in Fig. 1 (left). The rectangular domain consists of a block structured computational mesh defining a circular region around the circular cylinder and was used only for the OpenFoam simulation. The circular mesh region is discretised by 200x60 (peripheral x radial) finite volume cells. The length of the cylinder is 10D meshed using 160 finite volume cells in the spanwise direction. The top, bottom and inlet boundaries are located at a distance of 20D, while the outlet boundary is defined at an 80D distance. The computational grid of the rectangular domain involves 3,896,320 finite volume cells.

Figure 2 shows the round domain, where the domain is characterised by D_∞/D=60 with mesh points of 180x118 (peripheral x radial). The length of the cylinder is 4D with 40 mesh points. In the physical domain logarithmically spaced radial cells are used, providing a fine grid scale near the cylinder wall and a coarse grid in the far field, and along the length of cylinder a uniform grid is used. This mesh contains 849,600 finite volume cells.

For both domains, a uniform velocity distribution (u_∞) was prescribed at the inlet. The origin of the coordinate system is in the centre of cylinder. Incompressible Newtonian fluid flow with constant flow properties is assumed.

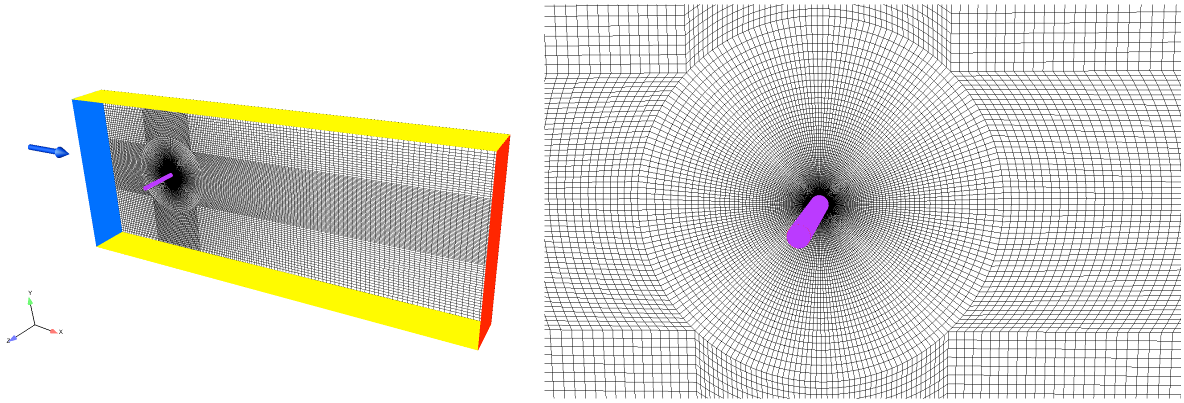

Fig. 1
Computational domain and mesh of the rectangular domain

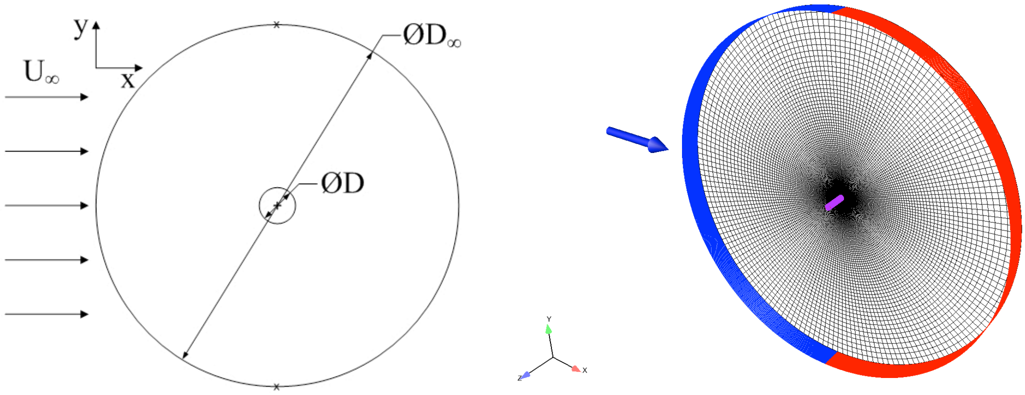

Fig. 2
Computational domain and mesh of the round domain

In Ansys Fluent commercial software simulations the 3D unsteady, laminar, segregated solver is used to solve the incompressible flow on a collocated grid arrangement. The *Second Order Upwind Scheme* was used to discretise the convective terms in the momentum equations. The semi-implicit method for the pressure linked equations (SIMPLE) scheme is applied for solving the pressure-velocity coupling.

The governing equations of a transient, isothermal and incompressible flow problem were also investigated applying the open-source code OpenFoam. The finite volume discretisation employs the collocated variable arrangement, where all physical values share the same control volumes.

The time-dependent simulation was carried out using a pressure-implicit algorithm (PISO) executing 2 steps of pressure correction within each loop. The Crank-Nicholson scheme was applied for the time integration. A constant time step size of 5×10^{-4} s was chosen in order to preserve the maximum Courant number in an acceptable range stabilizing computation.

Due to the high number of finite volume cells the computational domain was decomposed in advance in order to reduce the user waiting time and to allow parallel computation. Here, a simple decomposition algorithm has been chosen.

The numerical simulations based on OpenFoam were performed in parallel on the computing cluster Kármán at the University of Magdeburg. This system consists of 544 computing cores (AMD Quad Core 2.1 GHz) and an InfiniBand network connection. By using 192 computing cores, the unsteady simulation of 20 seconds of physical time on the rectangular domain required up to 105 hours wall clock time. These values are far from optimal; therefore, further optimization is required.

COMPUTATIONS

The accuracy of the numerical results is compared by means of integral quantities such as lift C_L and drag C_D coefficients. The coefficients of lift and drag are defined as follows:

$$C_L = \frac{2F_L}{\rho U_\infty^2 D}, \qquad C_D = \frac{2F_D}{\rho U_\infty^2 D}, \tag{1}$$

where ρ is the fluid density, F_L and F_D are the lift and drag forces per unit length of the cylinder, respectively. The Strouhal number (St), the non-dimensional vortex shedding frequency is defined as St=fD/u_∞, where f is the vortex shedding frequency. The time-mean and root-mean-square (rms) values of a T-periodic function $C(t)$ are defined as

$$C_{mean} = \frac{1}{nT}\int_{t_1}^{t_1+nT} C(t)dt, \; C_{rms} = \sqrt{\frac{1}{nT}\int_{t_1}^{t_1+nT} \left[C(t)-C_{mean}\right]^2 dt}. \tag{2}$$

Here T is the period of vortex shedding, n is the number of periods and C refers to the drag and lift coefficients.

COMPUTATIONAL RESULTS

Various computations using different mesh resolutions and different Reynolds-numbers were performed. Here, only selected results are presented for a single Reynolds number of 200. For both codes the lift and drag coefficients are analysed against the non-dimensional time (non-dimensionalised by D/u_∞) for the different computational domains (see Figs. 3 and 4). Figure 3 illustrates the amplitude of lift is nearly the same for the round domain and for the two programs. This value is slightly higher for the rectangular domain. A good agreement can be observed in Fig. 4 between the drag coefficient curves obtained by OpenFoam and Fluent on the round domain. The drag curve obtained by OpenFoam on the rectangular domain is overpredicted compared to the results on the round domain. More systematic analysis is required in order to quantify this issue.

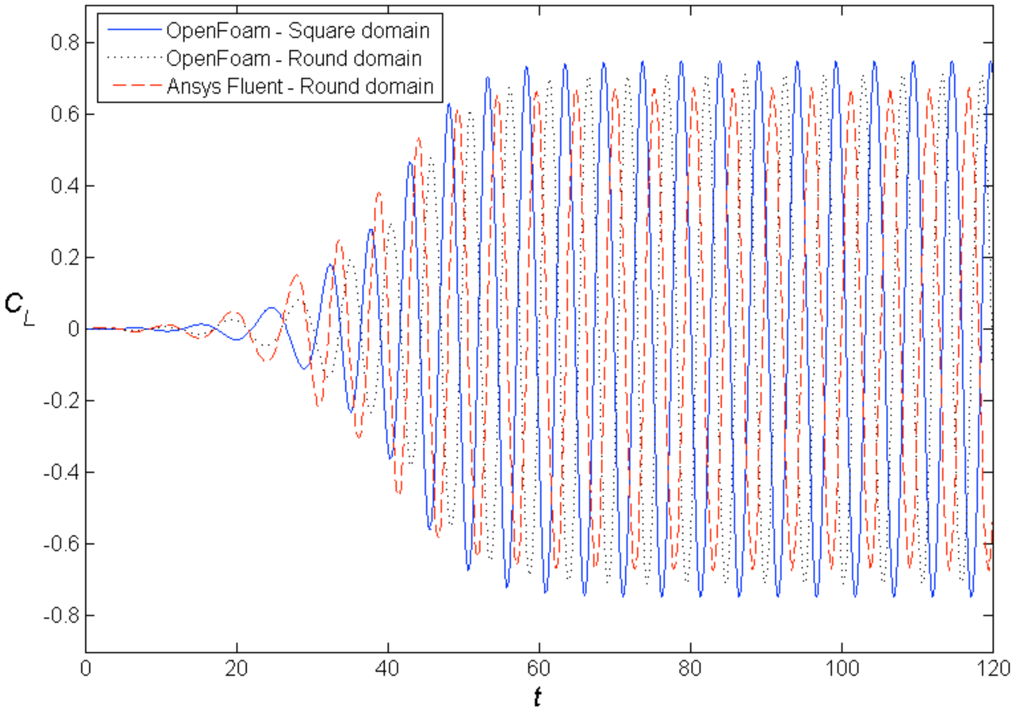

Fig. 3
Lift coefficient versus non-dimensional time, Re=200

In Table 1 the results of a few studies at Re=200 are included. For the round domain the results of OpenFoam and Ansys Fluent are nearly identical. The Strouhal number for the rectangular domain is higher than for the round domain, but the Strouhal number of Mittal [3] and Williamson and Brown [4] fit quite well to the present data on the rectangular domain. Williamson and Brown [4] experimented with the flow past a circular cylinder, while Mittal [3] numerically investigated steady and unsteady flow past a cylinder using a stabilised finite element formulation.

The rms of lift coefficient of Norberg [1] agrees well with the results of the Fluent simulation. The drag coefficients of the round domain agree excellently with those of Rajani et al. [5] using an implicit pressure-based finite volume method.

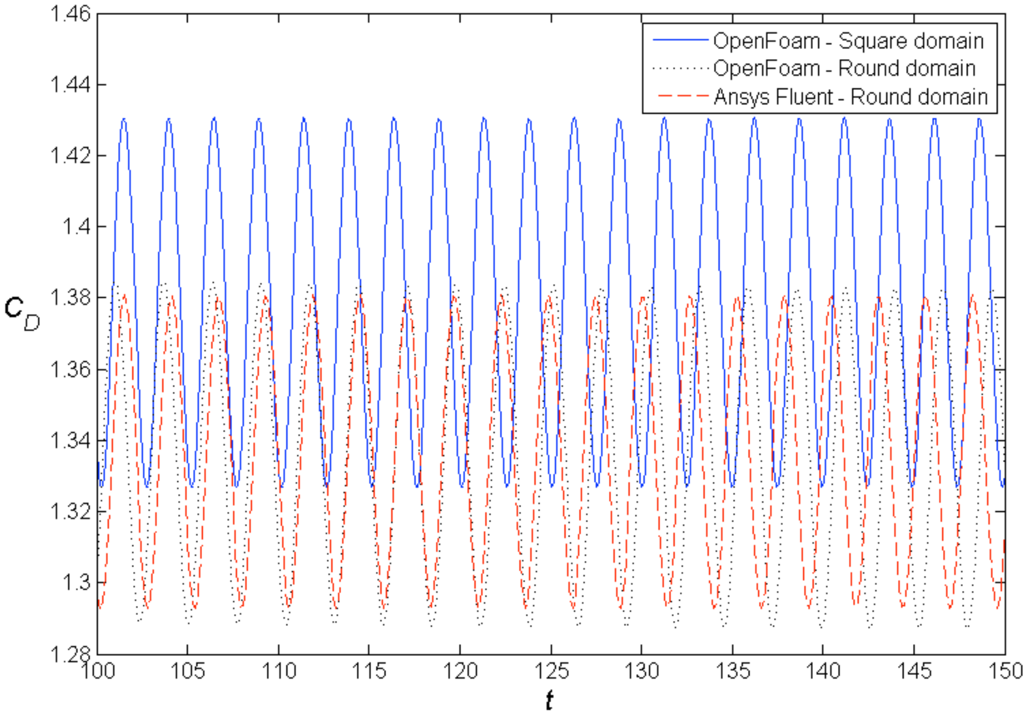

Fig. 4
Drag coefficient versus non-dimensional time, Re=200

Table 1
Comparison of dimensionless coefficients with other studies at Re=200

	St	C_{Lrms}	C_D	C_{Drms}	C_{Lmax}	
OpenFoam, ***rectangular domain***	0.1955	0.5281	1.3796	0.0366	0.7466	CFD
OpenFoam, ***round domain***	0.1921	0.4996	1.3349	0.0338	0.7071	CFD
Ansys Fluent, ***round domain***	0.1925	0.4754	1.3373	0.0312	0.6631	CFD
Norberg [1]	0.1964	0.4757				EM
Williamson and Brown [4]	0.1945					EM
Rajani et al. [5]	0.1936	0.4276	1.338			CFD
Mittal [3]	0.1947	0.489	1.327	0.0309		CFD

(CFD – Computational Fluid Dynamics, EM – experimental measurement)

In our investigations the flow was fully 2D up to 16 seconds of physical time (non-dimensional time, *t*=467), after which 3D flow structures started to appear. The first 3D disturbances are illustrated in Fig. 5, where the isosurfaces of the pressure field as well as the streamlines of the flow are shown at the physical time of 20 seconds obtained by OpenFoam for the round domain.

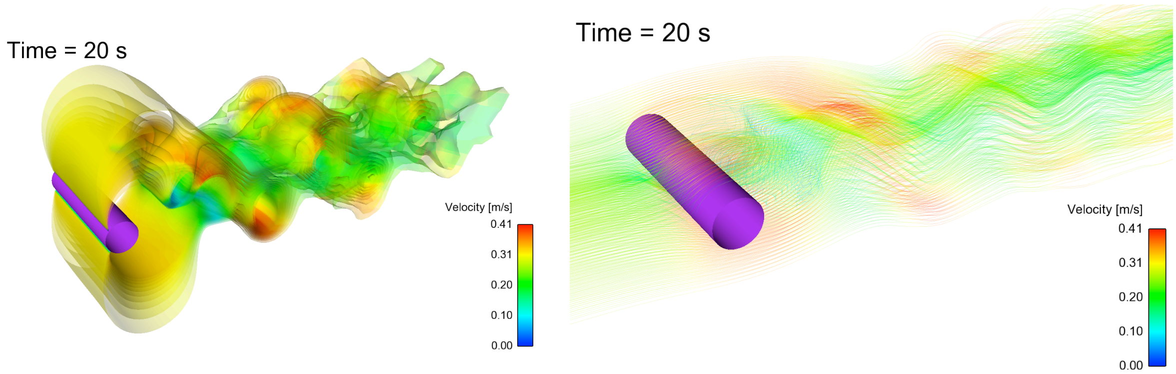

Fig. 5

Pressure isosurfaces coloured by velocity magnitude (left) and streamlines (right)

CONCLUSIONS

Three-dimensional laminar flow around a stationary circular cylinder at Reynolds number 200 has been investigated numerically using two CFD software packages, OpenFoam and Ansys Fluent. The two programs show good agreement for the rms value of lift for the round domain. This value is slightly higher for the rectangular domain. Good agreement can also be observed for the drag coefficient curves for the round domain. However, the drag curve obtained by OpenFoam on the rectangular domain is overpredicted compared to the results for the round domain. Further analysis is required in order to quantify this issue.

The present results form the basis of a long-term research project. Further steps include modelling cylinder oscillation and investigation of Reynolds number effects using the two software packages.

ACKNOWLEDGEMENTS

The support provided by the Hungarian Scientific Research Fund under contract No. OTKA K 76085 is gratefully acknowledged. The work was carried out as part of the TÁMOP-4.2.1.B-10/2/KONV-2010-0001 project in the framework of the New Hungarian Development Plan. The realization of this project is supported by the European Union, co-financed by the European Social Fund.

REFERENCES

[1] NORBERG, C.: **Flow around a circular cylinder: aspects of fluctuating lift**. Journal of Fluids and Structures **15** (2001), 459-469.

[2] WILLIAMSON, C.H.K., ROSHKO, A.: **Vortex formation in the wake of an oscillating cylinder**. Journal of Fluids and Structures **2** (1988), 355-381.

[3] MITTAL, S.: **Excitation of shear layer instability in flow past a cylinder at low Reynolds number.** Int. Journal for Numerical Methods in Fluids **49** (2005), 1147–1167.

[4] WILLIAMSON, C.H.K., BROWN, G.L.: **A series in 1/√Re to represent the Strouhal-Reynolds number relationship of the cylinder wake**. Journal of Fluids and Structures **12**(8) (1998), 1073-1085.

[5] RAJANI, B.N., KANDASAMY, A., MAJUMDAR, S.: **Numerical simulation of laminar flow past a circular cylinder.** Applied Mathematical Modelling **33** (2009), 1228–1247.