ITTC

Innovation and Technology Transfer Centre
University of Miskolc

XXIV. microCAD International Scientific Conference 18-20 March 2010

F szekció: Áramlás- és hőtechnika

Section F: Fluid and Heat Engineering

Nemzeti Kutatási és Technológiai Hivatal

A projekt a Nemzeti Kutatási és Technológiai Hivatal támogatásával valósult meg.

Kiadja *a Miskolci Egyetem Innovációs és Technológia Transzfer Centruma*
Kiadásért felelős: *Dr. Dobróka Mihály rektorhelyettes*
Szerkesztő: *Dr. Bikfalvi Péter osztályvezető*
Nyomda: *ME Sokszorosító Üzeme*
Üzemvezető: *Kovács Tibor*
Nyomdaszám: *ME.Tu-138/2010.*
ISBN 978-963-661-925-1 Ö
ISBN 978-963-661-910-7

GRID INDEPENDENCE STUDY FOR FLOW AROUND A STATIONARY CIRCULAR CYLINDER

Betti Bolló
Assistant lecturer
Department of Fluid and Heat Engineering, University of Miskolc, Hungary

ABSTRACT

The flow around a stationary circular cylinder at low Reynolds numbers is investigated numerically using FLUENT commercial software. A systematic investigation of the effect of domain size, spatial and temporal resolution on drag, lift and base-pressure coefficients and Strouhal number is carried out in the laminar, two-dimensional regime. The results agree very well with the data known from numerical and experimental studies in the literature.

INTRODUCTION

The incompressible flow past a stationary cylinder is a classical bluff body problem in fluid mechanics. Its physical and real-life applications have attracted the attention of engineers and scientists for over a century, leading to many theoretical and experimental investigations, summarized in Zdravkovich [7]. Despite its simple geometry, the flow past a circular cylinder is considered to be a baseline case of more complex flows.

Extensive studies have been performed on resolution as well as blockage effects. The blockage B due to the cylinder is defined as the width H of the experimental apparatus or computational domain to the ratio of the cylinder diameter D. Examples of analyses of the influence of computational domain extensions on the drag at low Reynolds numbers can be found in Posdziech and Grundmann [5], Stansby and Slaouti [6], Kang [3] and of course Zdravkovich [7]. Kang concentrated on one Reynolds number ($\mathrm{Re} = 100$), and varied the time step and domain extension. While the free-stream velocity varies linearly across the cylinder, computations were carried out for constant velocity [3]. Stansby and Slaouti also investigated Strouhal number for different blockage ratios at $\mathrm{Re} = 100$ [6].

In the study of Posdziech and Grundmann [5] different computational domains were used to obtain asymptotic solutions in the steady and unsteady flow regime at different Reynolds numbers. This study shows that a very careful validation of the numerical grid, both in terms of resolution and domain size, is necessary to obtain grid-independent solutions.

In the present work, first a fixed-size domain is applied and the number of grid points is varied. In numerical simulations of unsteady flows, the choice of the time step also plays an important role in obtaining an accurate solution to the momentum equations. Therefore, the effect of the time step is also investigated in addition to the grid independence. Next, the influence of computational domain extension is investigated for a given time step. In both cases the drag, lift, pressure and velocity distributions are analyzed. The Reynolds numbers (based on the diameter of the cylinder, free-stream velocity and viscosity of the fluid) in this study are 100 and 160.

COMPUTATIONAL DOMAIN

Consider the 2-D flow of an incompressible fluid with a uniform velocity U_∞ across a circular cylinder of diameter D. The unconfined flow condition is simulated here by considering the circular cylinder of diameter placed in a circular outer boundary of diameter D_∞. The schematic representation of the computational domain is shown in left side of Fig. 1. The blockage is expressed as $B = H / D$, where in our case the width of the domain corresponds to the outer circle ($H = D_\infty$). At the inlet uniform velocity distribution (U_∞) and constant temperature (T_∞) are prescribed. The origin of the coordinate system is in the center of cylinder. The positive x-axis is in the downstream direction.

Having fixed the domain size ($H = 60D$), the grid independence section is carried out for three non-uniform grids: *Coarse* ($C = 360 \times 236$), *Medium* ($M = 480 \times 256$) and *Fine* ($F = 720 \times 276$) grid. The quadratic mesh in which the radial spacing increases exponentially with distance from the cylinder is used at outer boundary. The *Coarse* type mesh is depicted near the cylinder on the right-hand side of Fig. 1.

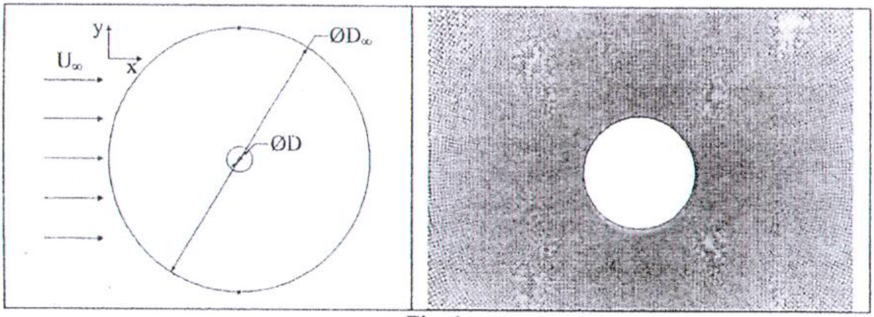

Fig. 1
Computational domain and Coarse mesh near the cylinder

NUMERICAL SOLUTION

This numerical study has been carried out using FLUENT v6.3.26 commercial software [2]. The two-dimensional, unsteady, laminar, segregated solver was used to solve the incompressible flow on the collocated grid arrangement. The fluid properties are constant. The *Second Order Upwind scheme* has been used to discretize the convective terms in the momentum equations. The semi-implicit method for the pressure linked equations (SIMPLE) scheme was applied for solving the pressure-velocity decoupling.

The finite-volume method has been validated by comparing dimensionless coefficients. The accuracy of numerical results is compared by means of integral quantities like drag, lift and the base-pressure coefficients and the Strouhal number. The drag and lift coefficients are calculated as

$$C_L = \frac{2F_L}{\rho u_\infty^2 A}, C_D = \frac{2F_D}{\rho u_\infty^2 A}, \tag{1}$$

where ρ is the fluid density, u_∞ is the free-stream velocity, F_L and F_D is the lift and drag force. A frequently considered quantity is the nondimensional pressure at the rear stagnation point, called the 'base-pressure' coefficient (C_{pb}). Generally, the base-pressure coefficient is

$$C_{pb} = \frac{2(p - p_\infty)}{\rho u_\infty^2}, \tag{2}$$

where p_∞ is the reference pressure. The Strouhal number, the nondimensional shedding frequency, is defined as $St = fD/u_\infty$, where f is the oscillation frequency of C_L. The time averaged and root-mean-square (rms) values of the pressure-based, lift and drag coefficient fluctuations are defined as

$$C_{mean} = \frac{1}{nT}\int_t^{t+nT} C(t)dt, \ C_{rms} = \sqrt{\frac{1}{nT}\int_t^{t+nT} [C(t) - C_{mean}]^2 dt}, \tag{3}$$

where T is a period of a vortex shedding, n is the number of periods. The general coefficient C stands for drag, lift, and base-pressure coefficients.

GRID INDEPENDENCE

In addition to the implementation of boundary conditions and the quality of the numerical solutions, the grid resolution in the domain and size of the time step are important for the reliability of the numerical results. Therefore any uncertainties due to the grid resolution should be avoided. Although there are no generally accepted rules to overcome the problem of grid convergence, the situations should be treated individually. The solution of the problem should be tested on different grid resolutions and different time step and by means of this process the optimum grid and time step should be determined for the calculations.

Results are presented for the computations of fix blockage ratio $H/D = 60$ at $\mathrm{Re} = 160$. A computational time step of $\Delta t = 0.0001 \div 0.05$ is used for time advancement in the simulations. Fig. 2 shows the Strouhal numbers (St), time-mean values of the base-pressure ($-C_{pbmean}$) and drag coefficients ($C_{\mathrm{D}mean}$) and the rms values of the lift coefficient fluctuations ($C_{\mathrm{L}rms}$) for different time steps.

The dimensionless coefficients greatly increase with reducing time step and at about $\Delta t = 0.001$ converge to a nearly constant value. In moving from the grid *Coarse* to *Fine*, the results change insignificantly, but are accompanied by an enormous increase in the computational time to achieve the required level of accuracy, especially for decreasing time step. Therefore, the grid *Coarse* is believed to be sufficiently refined to resolve the flow phenomena within the range of conditions of interest here. From an analysis of the dimensionless coefficients we come to the conclusion that a time step of $\Delta t = 0.001$ is sufficient for the calculations.

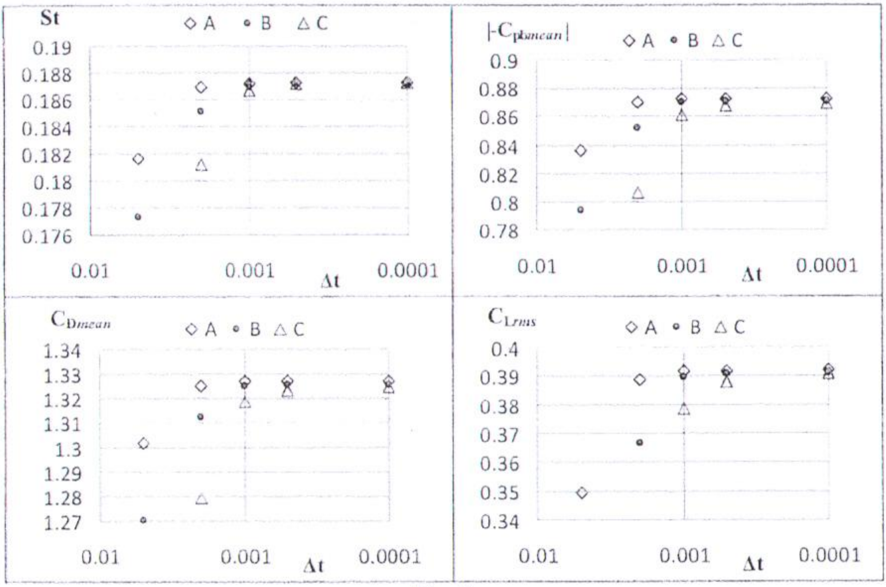

Fig. 2

The comparison of St, $-C_{pbmean}$, C_{Dmean}, and C_{Lrms} versus time step

EFFECT OF DOMAIN

In the following step, the influence of computation domain extension is investigated. The present computations have been carried out using a width of $H = 60D$ and $H = 220D$. Next $\mathrm{Re} = 100$ is considered, because a larger set of data can be found in the literature at this Reynolds number.

The dependency of the Strouhal number and mean drag coefficient on the domain extension is shown in Fig. 3. Results show that St and C_{Dmean}, remain nearly constant for large blockage ratios, which is comparable to the findings of other researchers. For both values the results agree excellently with those of Posdziech and Grundmann [5], who investigated the effect of the domain size at Reynolds numbers from 5 to 250 numerically by means of a spectral element method. Rectangular domain was applied, definition distance of upper/lower boundaries (L_b), so the blockage ratio was $B = H / D = 2L_b / D = 40 \div 8000$. A computational time step of $\Delta t = 0.005$ was used in all computations. Kang [3] used the immersed boundary method for calculations, where the governing equations are resolved with the finite volume approach. While smaller blockage ratios from 10 to 80 are applied, but for $B = 80$ same values are got as present work. Stansby and Slaouti [6] gave a polynomial formula for approximations of vortex shedding at $\mathrm{Re}=100$, when the blockage ratio varies: $St=0.166+0.0516\exp(-0.1248\,B)$. For calculations the random-vortex method is used. The character of the present result is similar to this approximation, but deviates slightly. The most likely reason for the deviation is that a smaller domain extension was investigated by Stansby and Slaouti.

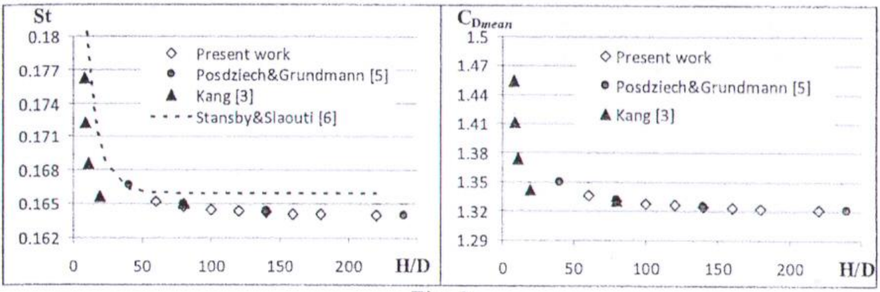

Fig. 3

Variation of St, and $C_{\mathrm{D}mean}$ with blockage ratio at $\mathrm{Re}=100$

Table 1

Comparison of dimensionless coefficients with other studies at $\mathrm{Re}=100$

Researcher	*H/D*	St	$C_{\mathrm{L}rms}$	$C_{\mathrm{D}mean}$	$-C_{pbmean}$	*Method*
Present work	220	0.164	0.225	1.321	0.698	CFD
Posdziech&Grundmann [5]	8000	0.163	0.316	1.312	0.690	CFD
Stansby & Slaouti [6]	24	0.166	0.243	1.329		CFD
Kang [3]	80	0.165	0.228	1.33		CFD
Baranyi & Lewis [1]	40	0.166	0.229	1.34	0.7147	CFD
Mittal [4]	50	0.164	0.226	1.322		CFD
Williamson [8]		0.164				EM

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

In Table 1 the results of a few studies at $\mathrm{Re}=100$ are included. The Strouhal number of Williamson, who experimented with the flow over a circular cylinder [8], fits perfectly to the present work. Baranyi and Lewis [1] used a 2D in-house code based on a finite difference solution of the governing equations. Their data coincide on the whole with present work, although the coefficients are slightly higher, but they applied a smaller blockage ratio. Mittal used a stabilized finite element formulation for steady and unsteady flow past a cylinder [4], and results match well with the results of the present study.

SUMMARY

The finite volume method has been utilized to compute flow past a cylinder, in two dimensions. The unsteady flow has been computed and compared with the available experimental and computational results in the literature and good agreement has been observed. Holding the domain size constant, it was found that a time step of $\Delta t = 0.001$ is sufficient. The effect of blockage was investigated at $\mathrm{Re} = 100$. It was found that the Strouhal number and drag coefficient first fall sharply with a small blockage then continue at constant values. It is observed that in case of blockage ratio $H/D > 100$ the results have shown good agreement with available data in the literature. Most numerical studies use the Strouhal number for the verification of the computational grid. In the present work drag, lift, base-pressure coefficient, and Strouhal number can be computed consistently by varying the domain extension. These dimensionless coefficients agree very well with available data in the literature. The present work shows that a very careful validation of the numerical grid, in terms of domain size and spatial and temporal resolution, is necessary to obtain grid-independent solutions. If this is done, well agreement between numerical and experimental data can be obtained in the laminar flow regime.

Further investigations of dimensionless coefficients at different Reynolds number will be carried out. We would like to prove that the grid-independence of the numerical solution is assured for Reynolds number between 60 and 200 if the domain size exceeds $H/D > 100$.

ACKNOWLEDGEMENT

The author is grateful to OTKA (K 76085), NKTH-OTKA (68207) and to the Hungarian-German Intergovernmental S&T co-operation programs P-MÖB/836 for the financial support of this research.

REFERENCES

[1] BARANYI, L., LEWIS, R.I.: **Comparison of a grid-based CFD method and vortex dynamics peredictions of low Reynolds number cylinder flows**. The Aeronautical Journal 110(1103) (2006) 63-71.

[2] FLUENT 6.2 **Getting Started Guide**, Fluent Inc., Lebanon, 2005.

[3] KANG, S.: **Uniform-shear flow over a circular cylinder at low Reynolds numbers.** J. of Fluids and Structures 22 (2006) 541–555.

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

[5] POSDZIECH, O., GRUNDMANN, R.: **A systematic approach to the numerical calculation of fundamental quantities of the two-dimensional flow over a circular cylinder**. J. of Fluids and Structures 23 (2007) 479-499.

[6] STANSBY, P.K., SLAOUTI, A.: **Simulation of vortex shedding including blockage by the random-vortex and other methods**. Int. J. for Numerical Methods in Fluids 17 (1993) 1003-1013.

[7] ZDRAVKOVICH, M.M.: **Flow around circular cylinders. Vol.1: Fundamentals**. Oxford University Press, Oxford/New York, 1997.

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