Effect of slit inclusions in drag reduction of Flow over square cylinders for low Reynolds number in the laminar regime.

***Rohit Bhattacharya[1], † Fausto Moreira-Izurieta[2],**

[1]School of Aerospace, Mechanical and Mechatronics Engineering. The University of Sydney, Australia.
[2] School of Aerospace, Mechanical and Mechatronics Engineering. The University of Sydney, Australia.

*Presenting author: rbha4656@uni.sydney.edu.au
†Corresponding author: andrew.mi8@gmail.com

Abstract

The flow over bluff bodies is separated compared to the flow over streamlined bodies. The investigation of the fluid flow over a circular cylinder with a streamwise slit has shown a reduction in the drag coefficient in the past for a very low Reynolds number. This work helps in understanding the fluid flow over bluff bodies in the laminar regime. An increase in the slit ratio is inversely proportional to the reduction in the drag coefficient resulting in a narrower wake which is a phenomenon seen in the turbulent regime, hence reducing the drag coefficient.

In this work two different approaches are used to simulate fluid flow over 2D cylinder of a square cross section and a comparison between the finite volume method and the Lattice Boltzmann Method is made. The width of the slits progressively increase from 10% all the way to 40% of the diameter of the cylinder. Reduction in the drag coefficients will be show for different values of Reynolds numbers as the width of the slit increases. The effect of slit inclusions on flow over cylinder for different values of Reynolds number is studied in further detail and discussed in this paper.

Keywords: CFD, Lattice-Boltzmann, Drag reduction, Flow over slit cylinder.

Introduction

Computational Fluid Dynamics (CFD) solves the Navier-strokes equations numerically for fluid flows using computers .The Lattice Boltzmann Method (LBM) is a promising method in simulation of flow in complex geometries [1].

The computational investigation of the fluid flow over a cylinder with a streamwise slit has been done before for cylinders of circular cross section where a reduction in drag coefficients have been seen for low Reynolds number (~10) [2]. This work shows a reduction of the drag coefficient by 7% and this was demonstrated for a slit/diameter ratio of 0.2 approximately. This work aims to highlight the effect of slit size on the flow over cylinders in laminar regime (Re=10). A comparative study between the finite volume method and the Lattice Boltzmann method will show flow patterns over the cylinders which will be used to measure the efficiency and accuracy of the two difference approaches.

CFD Simulation using Finite volume Analysis on ANSYS Fluent

The laminar model on ANSYS Fluent is used to carry out the 2D Simulations. A moving velocity to the wall is assigned which has the same speed of the fluid at the inlet of the computational fluid domain. Air is used as the fluid for the analysis that flows at a Reynolds number of 10. The following equation is used to calculate the Reynolds number.

$$\boldsymbol{Re} = \frac{\boldsymbol{\rho V D}}{\boldsymbol{\mu}} \tag{1}$$

Here is the ρ density of the fluid which is 1.225Kg/m^3,V is the velocity of the fluid which is 0.1477 m/s and μ is the viscosity of the fluid which is 0.0181 (mPa.s).

The drag force over the cylinder is obtained using ANSYS and this is used to calculate the drag coefficient which is given by the following equation.

$$\boldsymbol{C_d} = \frac{\boldsymbol{F_x}}{\boldsymbol{0.5\rho A V^2}} \tag{2}$$

Here C_d is the drag coefficient, F_x is the drag force, ρ is the density of the fluid, A is the projected area and V is the velocity of the fluid.

Lattice Boltzmann Method (LBM)

The Lattice Boltzmann Method algorithm was built on Python .The Velocity of the flowing fluid is discretised first which is obtained from the discretized Boltzmann transport equation [2].

$$\boldsymbol{f_\alpha(r + v_\alpha \Delta t, t + \Delta t) = f_\alpha(r,t) + C(f_\alpha, f_\alpha)} \tag{3}$$

Where f is the distribution function for velocity, subscript α stands for the direction, $\boldsymbol{r}$ is the position vector for each lattice node, t is time and c is the lattice speed of sound. The D2Q9 model is a two dimensional lattice model with 9 discrete nodes assigned with velocity vectors, one at the centre and eight others surrounding it. The most critical point here is to specify the number of cells on the x and y axis as this defines the computational fluid domain to carry out simulations using the Boltzmann equations.

The following is the formula for the distribution function

$$f^{eq} = w_\alpha \left(1 + \frac{\boldsymbol{v_\alpha} \cdot \boldsymbol{u}}{c_s^2} + \frac{(\boldsymbol{v_\alpha} \cdot \boldsymbol{u})^2}{2c_s^4} - \frac{|\boldsymbol{u}|^2}{2c_s^2} \right) \tag{4}$$

Here w_α is the weight function in the α direction.

LBM Algorithm

The macroscopic variable is defined based on the distribution function. The information of the molecular number density (n) can be found using equation 5 , and momentum density ($n\boldsymbol{u}$) can be found using equation 6.

$$n = \sum_\alpha f_\alpha \tag{5}$$

$$\boldsymbol{nu} = \sum_{\alpha} \boldsymbol{f}_{\alpha} \boldsymbol{v}_{\alpha} \tag{6}$$

The pressure distribution is given by the following equation.

$$\boldsymbol{p} = \boldsymbol{n}\boldsymbol{c}_s^2 \tag{7}$$

Here c_s is the speed of sound as lattice constant. Using the complementary ideal gas equations ($pV = NRT, p = nRT$), It can be shown that$RT = c_s^2$. The values of w (weight factor) and c_s depend on the specific choice of the lattice velocity model. Table 1 summarises values of w and c_s for the chosen lattice model D2Q9 [4].

Table 1: Weight factors used for the lattice nodes

Model	v_α	w_α	c_s^2
D2Q9	(0,0)	16/36	
	(±1,0) ,(0,±1)	4/36	1/3
	(±1,±1)	1/36	

The computational fluid domain is a 2D channel of a 12D × 7D Cross section where the square cylinder has a side of D (=1μm). The square cylinder is located in the centre of the computational fluid domain for carrying out the iterations. Fig.1 shows the schematic of the computational fluid domain. In previous studies where computational analysis was performed using MATLAB where the simulation time was 21 seconds. The same simulation was carried out using Python in 15 seconds which also shows improved efficiency and accuracy.

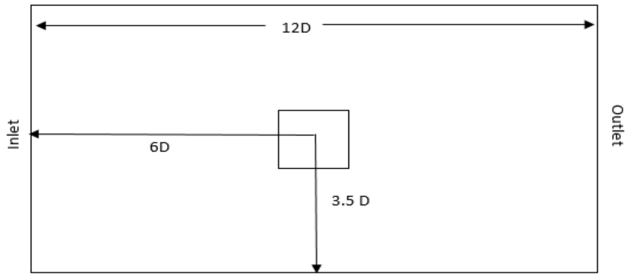


Figure 1: The schematic of the Computational fluid Domain

Table 2: Validation of numerical results obtained using CFD and LBM, against a variety of prior works in the literature at constant *Re*=10.

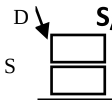
Literature C_d	C_d obtained by CFD	C_d obtained by LBM
7.29 (Durao et al) [5]	8.97	8.48
7.53 (Sohankar et al)[6]	8.97	8.48

The reported data indicate that the values of the drag coefficients obtained from LBM and CFD simulation are very close to prior works results. The drag coefficient results are especially close to that obtained by Durao [5] under Poiseuille flow conditions, suggesting the wall effects are not completely avoided in our work. Upon acceptable validation of our CFD model and LBM, further investigations are conducted for the slit cylinder. The boundary conditions remain the same, only the geometry in the computational fluid domain would change as a slit would be considered inside the cylinder.

Investigation of the effect of the slit

A slit is a narrow cut or opening in a bluff body. This section highlights the effect of the size of slits in the reduction of drag coefficient in bluff bodies. Cases where the slit to width (side of the square) ratio progressively increases by 10% up to 40% are considered and shown in Table 3.

Table 3: Drag coefficient and % reduction in comparison to slit free cylinder, calculated from CFD and LBM for various slit ratio and compared.

 S/D	CFD (ANSYS)		LBM (Python)		%Difference CFD vs. LBM
	C_d	%Reduction	C_d	%Reduction	
0	8.97	-	8.48	-	5.77
0.1	8.21	8.47	8.22	3.06	0.12
0.2	8.02	10.59	8.05	5.07	0.37
0.3	7.94	11.48	7.87	7.19	0.89
0.4	7.87	12.26	7.78	8.25	1.16

All the values obtained from ANSYS Fluent and LBM have been compared in terms of the percentage difference with respect to a slit-free cylinder. It can be seen from the results in the last column that the calculated drag coefficient from the two methods agree well and in most cases the difference is less than 2%.

Based on the results obtained we can conclude that both models are reliable and produce similar results for the flow field and drag coefficient. However there are some advantages in terms of algorithm simplicity and computational cost. The lattice Boltzmann method has a few advantages over the CFD method using ANSYS Fluent as outlined below

- The algorithm can be easily implemented on Python.
- Due to the regular lattice structure and because of the limited dynamic interaction that requires only one contact between each lattice node and its nearest neighbours for each iteration step.[7]
- Discretization of the macroscopic continuum equations is not needed. Hence, the LBM does not consider explicitly the distribution of pressure on interfaces of refined grids since the implicitly is already included in the computational scheme.[2]
- The computation time using LBM is considerably lower.

Conclusions

Numerical Investigation of fluid flow around cylinders of square cross sections were carried out with and without slits of varying sizes. The CFD method using ANSYS Fluent and the LBM Script using Python were used to simulate fluid flow and to calculate the drag coefficient. The numerical results agreed reasonably with available experiments at Reynolds number of 10 for a 2D cylinder. The incorporation of slits on such cylinders showed a considerable reduction on 12.3 % in drag coefficients.

References

[1] Liaw, K. (2005). *Simulation of flow around bluff bodies and bridge deck sections using CFD* (Doctoral dissertation, University of Nottingham).

[2] Bhattacharya, R., Moshfegh, A., & Jabbarzadeh, A. (2016). Effect of Slit Inclusions in Drag Reduction of Flow over Cylinders. In Applied Mechanics and Materials (Vol. 846, pp. 18-22). Trans Tech Publications.

[3] Dixon, A. G., Nijemeisland, M., & Stitt, E. H. (2006). Packed tubular reactor modeling and catalyst design using computational fluid dynamics. *Advances in Chemical Engineering*, *31*, 307-389.

[4] Zhang, H. (2008). Lattice Boltzmann method for solving the bioheat equation. Physics in medicine and biology, 53(3), N15.

[5] Durao, D. F. G., Heitor, M. V., & Pereira, J. C. F. (1988). Measurements of turbulent and periodic flows around a square cross-section cylinder. Experiments in Fluids, 6(5), 298-304.

[6] Sohankar, A., Norberg, C., & Davidson, L. (1998). Low-Reynolds-number flow around a square cylinder at incidence: study of blockage, onset of vortex shedding and outlet boundary condition. International journal for numerical methods in fluids, 26(1), 39-56.

[7] Abhijeet, T (2011). Introduction to the Lattice Boltzmann Method. 10th Indo-German Winter Academy 2011, IIT Kharagpur, India.