Applied Mechanics and Materials Vols 592-594 (2014) pp 1972-1977
© (2014) Trans Tech Publications, Switzerland
doi:10.4028/www.scientific.net/AMM.592-594.1972

Submitted: 2014-04-23
Revised: 2014-05-08
Accepted: 2014-05-19
Online: 2014-07-15

TRANSIENT SIMULATION OF FLOW PAST SMOOTH CIRCULAR CYLINDER AT VERY HIGH REYNOLDS NUMBER USING OpenFOAM

Sangamesh.M.Hosur[1,a*], D.K.Ramesha[2,b], Suman Basu[3,c]

[1&2]Department of Mechanical Engineering, University Visvesvaraya College of Engineering, Bangalore University, K.R Circle Bangalore-560001(INDIA)
[3]Robert Bosch Engineering and Business Solutions Ltd, RMZ EcoSpace , Bellandur, Bangalore - 560 103(INDIA)
[a]sangamesh.hosur@gmail.com, [b]rameshadkuvce@gmail.com, [c] suman_basu@yahoo.com

Keywords : High Reynolds Number, Turbulent Flow, URANS, OpenFOAM, pimpleFoam

Abstract: Flow past a smooth circular cylinder at high Reynolds number (Re=3.6 x 10^6) which covers the upper-transition regime has been investigated numerically by using Open source Field Operation and Manipulation (OpenFOAM) package. OpenFOAM is a free, open source Computational Fluid Dynamics (CFD) software package. The numerical model has been set up as two dimensional (2D), transient, incompressible and turbulent flow. A standard high Reynolds number k-ε turbulence model is included to evaluate the turbulence. The objective of the present work is to set up the case using pimpleFoam solver which is an Unsteady Reynolds Averaged Simulations (URANS) model and to evaluate the model for its conformance with available literature and experiments. The results obtained are compared with experimental and numerical data.

Introduction

The flow is classified in to laminar and turbulent based on the Reynolds number (i.e $Re=U_\infty.D/\nu$) as for laminar Re< 2000, for turbulent Re>4000 and 2000<Re>4000 is transition region. Many experimental and numerical simulations have been carried out for laminar and turbulent flows. But very high Reynolds no flow conditions are very hard and expensive in an experimental set up requiring appropriate experimental facilities for minimizing human and instrument errors during measuring hydrodynamic quantities. Therefore an attractive alternative is to use Computational Fluid Dynamics (CFD) to obtain the essential hydrodynamic quantities needed for engineering design. To date, not many experimental and numerical simulations have been performed to predict very high Reynolds number flows (Re > 10^6) around a smooth circular cylinder due to the complexity of the flow. E. Achenbach [1] has done experimental investigation at Reynolds numbers, $6 \times 10^4 < Re < 5 \times 10^6$ and obtained data which allow one to define three states of the flow: the subcritical flow, where the boundary layer Separates laminarly; the critical flow, in which a separation bubble, followed by a turbulent reattachment, occurs; and the supercritical flow, where an immediate transition from the laminar to the turbulent boundary layer is observed at a critical distance from the stagnation point. Celik and Shaffer [2] showed that there are significant differences between calculated and measured mean flow parameters. Cantalano, Wang, Iaccarino and Moin [3] found Large Eddy Simulations (LES) are considerably more accurate than the RANS results but pretty expensive.

Hence the main objective of the present study is to evaluate whether the standard high Reynolds number k -ε model is applicable for engineering applications of high Reynolds number of the order of 10^6. The flow around a 2D smooth circular cylinder at Re=3.6X10^6 is investigated numerically, and these results are compared with available experimental data and the numerical results reported by Catalano et al. [3] and Singh and Mittal [4].

Numerical Simulation

Flow simulation is carried out by using OpenFoam. The physics of the flow is considered as 2D, transient, incompressible and turbulent flow. Hence we chose pimpleFoam as the solver for this case. pimpleFoam algorithm uses an inner PISO loop to get an initial solution, which is then under-relaxed and corrected using an outer SIMPLE loop. This method enables unsteady simulations at Courant number (Co) numbers larger than 1. In theory, very large Co numbers could be maintained if a large number of SIMPLE correction loops were applied along with large under-relaxation factors. But this has implications for the time-accuracy of the solution, because events that are shorter than the time step are missed completely by the solver. A too large time step generally smears the solution, regardless of the grid resolution. When using pimpleFoam one has to keep in mind that too large Co numbers risk reducing the accuracy of the solution. Hence time step is chosen in such a way that Co_{max} is less than 1.

$$Co = \frac{\Delta t\, U_\infty}{\Delta x} \tag{1}$$

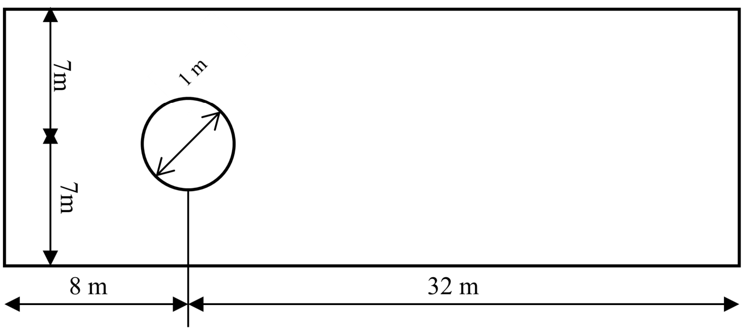
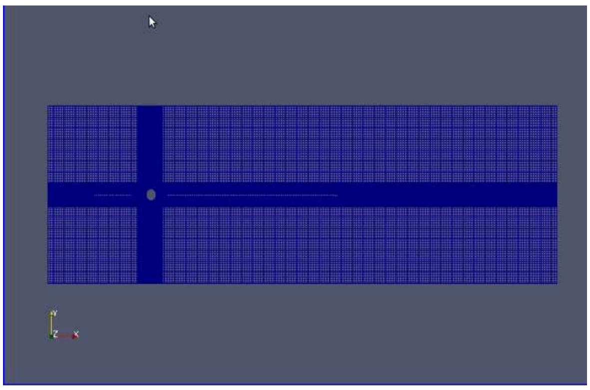


Fig 1: Domain Size for Simulation

Geometry and Mesh Generation : The geometric size of the rectangular computational is as shown in Fig1. To generate the geometry and mesh for this case we have to use blockMeshDict file. In this we have to specify vertices, blocks, edges, boundary patches and merge pairs the co-ordinates of the geometry and also discretization parameters. The blockMeshDict file is located in the constant/polyMesh directory of the case, where the information about the vertices, blocks, edges, the boundaries, and the mergePatchPairs are saved. The generated mesh is as shown in Figure 2 and 3.



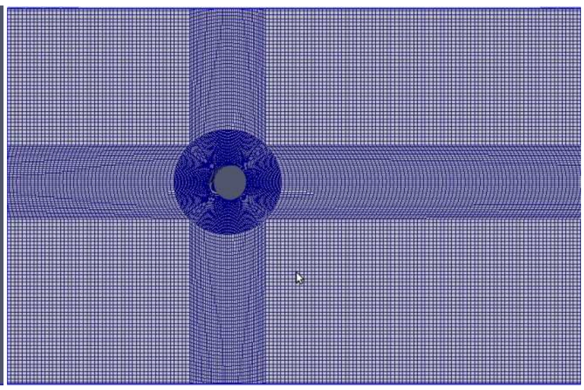


Fig 2 : Mesh pattern of the case Fig 3 : Zoomed view of Mesh pattern

Initial and Boundary Conditions : The 0 directory contains 4 sub-directories: p, U, k and ε whose initial values and boundary conditions must be set. The velocity has been calculated from the known Reynolds number as mentioned the literature [5] i.e 3.6×10^{6},

$$Re = U_\infty D/\nu \tag{2}$$

U_∞ is calculated as 3.6 m/s which is the free stream velocity.

The turbulence intensity, *I*, is a measure of the strength of the velocity fluctuations, *u'*, compared to the strength of the bulk velocity U_∞. By definition, *I* is equal to the ratio of the root-mean-square of the velocity fluctuations to the mean freestream velocity.

$$I = \frac{u'}{U_\infty} \tag{3}$$

The best way to get values for inlet turbulence quantities specification is to have experimental data, from which turbulence intensity can be calculated. When experimental data is not available, we can use the following formula to calculate *I*.

$$I = 0.16Re^{-1/8} = 3.22\% \tag{4}$$

$$\text{Turbulent Kinetic Energy} \quad k = \frac{3}{2}(U_\infty I)^2 = 0.01142 \tag{5}$$

$$\text{Turbulent Dissipation Rate} \quad \varepsilon = \frac{0.09k^2}{\beta \nu} = 0.001568 \tag{6}$$

Where β is Turbulent Viscosity ratio.

Running the Case: After making the necessary setting in controlDict, fvSchemes and fvSolutions, the case is ready to run. We need navigate to the case directory and then type pimpleFoam which starts the simulation. But the simulation results are written in a file only at a specific interval as specified at writeInterval in controlDict file. To get the log of the simulation we need to type pimpleFoam > log. This creates a file called log and writes all the info in to it.

Results And Discussion

The flow field around the cylinder is characterized by a fairly large wake behind the cylinder caused by separation at around mid-point of the upper and lower curvature of the cylinder. The point of separation is near the expected location.

Figures 4 & 5 show the flow streamlines at different times. The wake formation and its fluctuation from one side of the central axis to other side is in figure 5 and 6. The wake propagation has been shown in Figure 6.

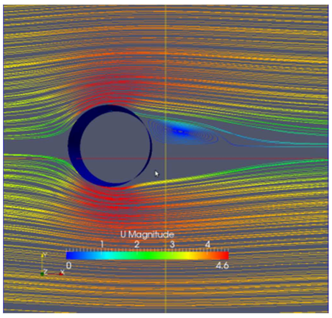


Fig 4: Streamlines at 3.0249 sec showing wakes

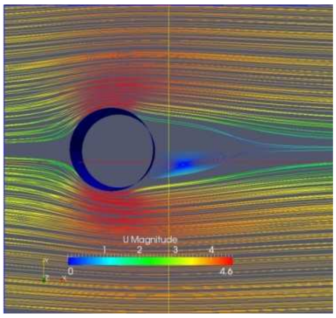


Fig 5: Streamlines at 3.79387 sec showing wakes

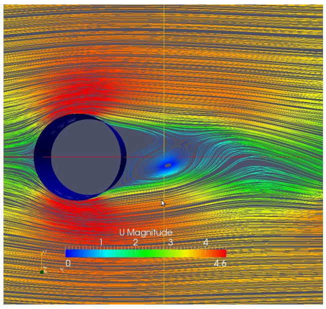


Fig 6: Streamlines at 27.4887 sec showing wakes

Velocity and pressure profiles for the time t=27.4887 are in Figures 7 and 8.

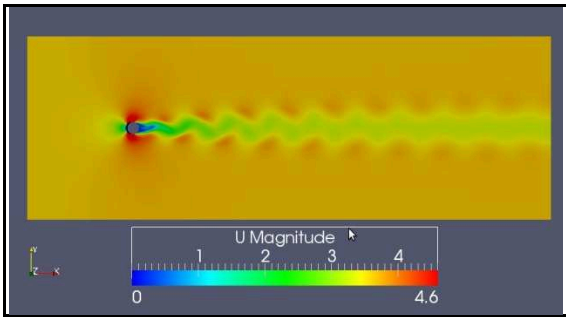


Fig7: Velocity contour at 27.4887 sec showing wakes propagation

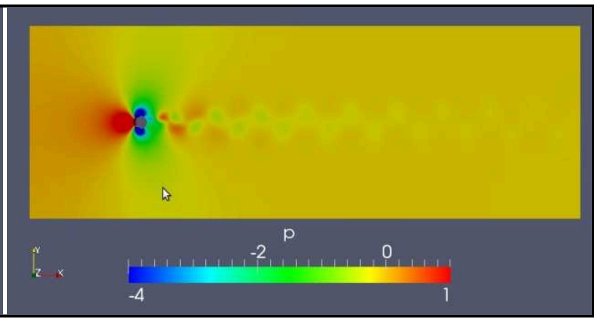


Fig 8: Pressure contour at 27.4887 sec showing wakes propagation

The computations have been performed at Re = 3.6 x10^6, which is the upper-transition flow regime. The objective is to evaluate the applicability of using a standard high Reynolds number k -ε model for engineering computations of flow around a smooth circular cylinder in upper-transition flow regime. Therefore, essential hydrodynamic quantities, such as C_D, C_L and St, have been predicted and compared with published experimental data [1, 6, 7, 8, 9] and numerical results [10, 4].

The following charts show the fluctuation of C_D, C_L and Y component of the velocity against the time.

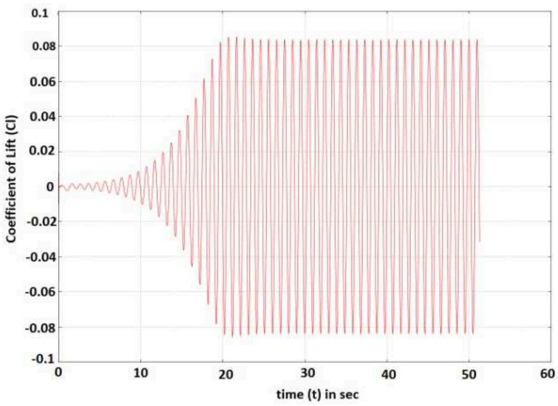


Fig 9: Fluctuation of Coefficient of Lift (C_L)Vs Time(t)

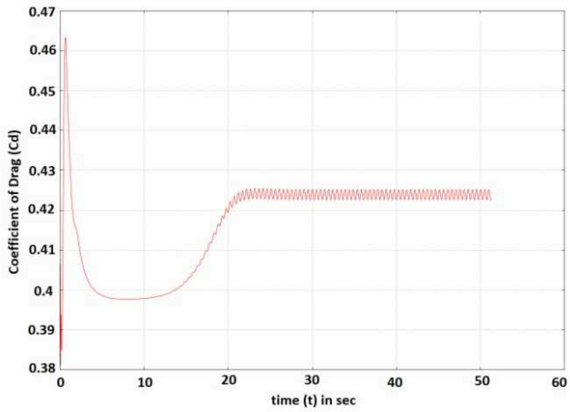


Fig 10: Fluctuation of Coefficient of Drag (C_D)Vs Time(t)

The probe has been inserted at a distance of 3.5 m from the centre of the cylinder behind it to calculate the velocity. The plot gives the fluctuation of the velocity in y-direction Vs time as shown if Figure 11.

Initially the fluctuation goes on increases but after some time period it stabilises and fluctuates between the constant values. By looking at the graph, we can say that the model satisfies the graph nature as per literature of standard k-ε model. But we can't say that simulation is write until unless we compare with the quantitatively with experimental results. This comparison has been done in the Table 1.

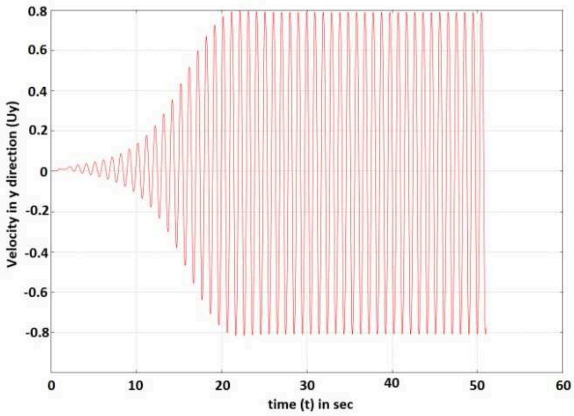


Figure 11: Fluctuation of Velocity(U_y) Vs Time(t)

The values of the parameters are in well agreement with the experimental and literature hence validation of the numerical model set in the OpenFOAM is correct. And this model can be used to solve the Industrial problems with proper care on mesh as the model is highly sensitive to mesh as of the fact that "Poor grid resolution, which becomes increasingly severe as the Reynolds number increases, and is the primary suspect"

Table 1: Comparison of the results

Re		C_D	C_L	St
3.6×10^6 Upper Transition regime	Muk Chen Ong [1]	0.4573	0.0766	0.3052
	Catalano et el [3]	0.46	-	-
	Published Experimental Data	0.36-0.75	0.06-0.14	0.17-0.29
	Present Simulation	0.4255	0.0853	0.2864

Conclusion

The flow around a 2D smooth circular cylinder has been computed for very high Reynolds number at the upper-transition flow regime, using the 2D RANS in conjunction with a standard high Reynolds number k -ε model. Although it has been shown earlier that this model gives less accurate predictions of flow with strong anisotropic turbulence, the present study shows that for engineering design purposes it gives satisfactory qualitative agreements with the published experimental data and numerical results in the supercritical and upper-transition flow regimes, i.e. Re $>10^6$. The present study should be reliable and useful as an engineering assessment tool for design work.

Acknowledgements
The authors are grateful to Robert Bosch Engineering and Business solutions Ltd for its simulation facilities and also thankful to Mr. Amine Abdelmoula (University of Stuttgart, Germany) for his valuable assistance.

Nomenclature

U_∞	Free stream velocity (m/s)
D	Diameter of the cylinder (m)
ν	Kinematic viscosity (m^2/s)
Δt	time step (s)
Δx	Minimum cell distance in x – direction (m)
k	Turbulent kinetic energy (m^2/s^2)
ε	Turbulent dissipation rate (m^2/s^3)
I	Turbulent Intensity (%)
β	Turbulent viscosity ratio
Co	Courant number
C_D	Co-efficient of drag
C_L	Co-efficient of lift
St	Strouhal No
U_y	Velocity in Y- direction (m/s)
OpenFOAM	Open source Field Operation and Manipulation
URANS	Unsteady Reynolds Averaged Simulations
CFD	Computational Fluid Dynamics
SIMPLE	Semi-Implicit Method for Pressure-Linked Equations
PISO	Pressure Implicit with Splitting of Operators
LES	Large Eddy Simulations
DNS	Direct Numerical Simulation

References

[1] Achenbach E. Distribution of local pressure and skin friction around a circular cylinder in cross-flow up to Re=5 x 10^6. J Fluid Mech 1968;34(4):625–39.

[2] Ismail Celik, Franklin D. Shaffer. *"Long time-averaged solutions of turbulent flow past a circular cylinder".* Journal of Wind Engineeringand Industrial Aerodynamics 56 (1995) 185-212

[3] Pietro Catalano, Meng Wang, Gianluca Iaccarino, ParvizMoin. Sunflower oil as a fuel for compression ignition engines. "*Numerical simulation of flow around a circular cylinder at high Reynolds numbers"*. International Journal of Heat and Fluid Flow 24 (2003) 463–469.

[4] Singh SP, Mittal S. Flow past a cylinder: shear layer instability and drag crisis. Int J Numer Meth Fluids 2005;47:75–98.

[5] Muk Chen Ong a, Torbjørn Utnes b, Lars Erik Holmedal a, Dag Myrhaug a, Bjørnar Pettersen, "*Numerical simulation of flow around a smooth circular cylinder at very high Reynolds numbers"*. Journal of Marine Structures 22 (2009) 142–153.

[6] Fung YC. Fluctuating lift and drag acting on a cylinder in a flow at supercritical Reynolds numbers. J Aeronautical Sci 1960; 27:801–14.

[7] James WD, Paris SW, Malcolm GV. Study of viscous cross flow effects on circular cylinders at high Reynolds numbers. AIAAJ 1980;18:1066–72.

[8] Jones GW. Unsteady lift forces generated by vortex shedding about large stationary and oscillating cylinders at high Reynolds numbers. In: ASME proceedings of the symposium on unsteady flow; 1968. Philadelphia, 68-FE-36.

[9] Zdravkovich MM. Flow around circular cylinders, vol. 1: fundamentals. New York: Oxford University Press; 1997

[10] Catalano P, Wang M, Iaccarino G, Moin P. Numerical simulation of the flow around a circular cylinder at high Reynolds numbers. Int J Heat Fluid Flow 2003;24:463–9.

Dynamics of Machines and Mechanisms, Industrial Research
10.4028/www.scientific.net/AMM.592-594

Transient Simulation of Flow Past Smooth Circular Cylinder at very High Reynolds Number Using OpenFOAM
10.4028/www.scientific.net/AMM.592-594.1972

DOI References
[1] Achenbach E. Distribution of local pressure and skin friction around a circular cylinder in cross-flow up to Re=5 x 106. J Fluid Mech 1968; 34(4): 625-39.
http://dx.doi.org/10.1017/S0022112068002120
[2] Ismail Celik, Franklin D. Shaffer. Long time-averaged solutions of turbulent flow past a circular cylinder,. Journal of Wind Engineeringand Industrial Aerodynamics 56 (1995) 185-212.
http://dx.doi.org/10.1016/0167-6105(94)00091-Q
[4] Singh SP, Mittal S. Flow past a cylinder: shear layer instability and drag crisis. Int J Numer Meth Fluids 2005; 47: 75-98.
http://dx.doi.org/10.1002/fld.807
[5] Muk Chen Ong a, Torbjørn Utnes b, Lars Erik Holmedal a, Dag Myrhaug a, Bjørnar Pettersen, Numerical simulation of flow around a smooth circular cylinder at very high Reynolds numbers,. Journal of Marine Structures 22 (2009) 142-153.
http://dx.doi.org/10.1016/j.marstruc.2008.09.001
[7] James WD, Paris SW, Malcolm GV. Study of viscous cross flow effects on circular cylinders at high Reynolds numbers. AIAAJ 1980; 18: 1066-72.
http://dx.doi.org/10.2514/3.50855
[10] Catalano P, Wang M, Iaccarino G, Moin P. Numerical simulation of the flow around a circular cylinder at high Reynolds numbers. Int J Heat Fluid Flow 2003; 24: 463-9.
http://dx.doi.org/10.1016/S0142-727X(03)00061-4