Simulation of Unsteady Flow around a Cylinder

Ridha Alwan Ahmed
Assistant .Lecturer
Al-Mustansiriyah University
College of Engineering
Mechanical dept.
Ridha_sjmo@yahoo.com

Abstract

In this paper, the phenomena of vortex shedding from the circular cylinder surface has been studied at several Reynolds Numbers (40≤Re≤ 300).The 2D, unsteady, incompressible, Laminar flow, continuity and Navier Stokes equations have been solved numerically by using CFD Package FLUENT. In this package PISO algorithm is used in the pressure-velocity coupling.

The numerical grid is generated by using Gambit program. The velocity and pressure fields are obtained upstream and downstream of the cylinder at each time and it is also calculated the mean value of drag coefficient and value of lift coefficient .The results showed that the flow is strongly unsteady and unsymmetrical at Re>60. The results have been compared with the available experiments and a good agreement has been found between them.

Keywords:-vortex shedding, cylinder, unsteady state, drag coefficient.

محاكاة التدفق الغير مستقر حول أسطوانة

الخلاصة

في هذا البحث تم دراسة ظاهرة سير الدوامة من سطح اسطوانة دائرية لعدة قيم من أرقام رينولدز تتراوح قيمها من 40 الى 300 .الجريان ذو البعدين ،غير المشتق، غير الانضغاطي، الطباقي ومعادلات ألاستمرارية ومعادلات نايفر ستوك تم حلها عدديا بواسطة برنامج فلونت لذلك استخدمت طريقة بيزو الرياضية في مزدوج السرعة والضغط.

العقد العددية تم توليدها باستخدام برنامج گامبت . توزيع السرعة وتوزيع الضغط تم الحصول عليها أمام ألأسطوانة وخلف ألأسطوانة وكذلك تم حساب معامل مقاومة الهواء ورفع الهواء. النتائج تبين أن الجريان غير مستقر بقوة وغير متماثل عندما يكون رقم رينولد أكبر من 60 .وقد تم مقارنة النتائج مع التجارب المتاحة ووجد توافق جيد بينهما.

الكلمات الدالة:- سير الدوامة، أسطوانة، حالة غير مستقرة، معامل المقاومة

1-Introduction

In fluid dynamics, vortex shedding is an oscillating flow that takes place when a fluid such as air or water flows past a bluff (as opposed to streamlined) body at certain velocities, depending on the size and shape of the body. In this flow, vortices are created at the back of the body and detach periodically from either side of the body [1]. The flow past a two-dimensional cylinder is one of the most common studies of aerodynamics. It is relevant to many engineering applications. The flow pattern and drag on a cylinder are functions of the Reynolds number, based on the cylinder diameter and the undisturbed free-stream velocity [2]. In addition, the phenomenon of vortex shedding from bluff bodies is very important in various engineering situations. Its control leads to the reduction in the unsteady forces acting on the bluff body and can significantly reduce its vibrations [3]. In reference [4] the authors presented calculations of unsteady 2D-flow past rectangular cylinders at incidence. The Reynolds numbers are low so that the flow presumably is laminar. Experimental results, at these low Reynolds numbers, are rather scarce. The influence of cylinder side ratio (B/A=1, 2, 4) at various angels of incidences ($\acute{\alpha}=0^0$-90^0) and for Re=100,200 was investigated. A number of quantities such as Strouhal number lift and drag coefficients and various surface pressure coefficients are also calculated.

In reference [5] the measurements show that , at a Reynolds number of 60, the Strouhal number for the shedding of vortices from a rotating cylinder is only weakly dependent on the value of α, the ratio of the cylinder's peripheral speed to its translational speed, up to the α value at which shedding was suppressed. Authors in [6] experimentally investigated the flow behavior of vortex shedding downstream a dual equilateral triangle-section of bluff body using oscillating outlet boundary condition. The motivation was to analyze the behavior of flow meter in unsteady flow. In [7] the optimal control of flow around a rotating cylinder is formulated and governed by the unsteady Navier-Stokes equations. In this work, the main objective consists of suppressing Karman vortex shedding in the wake of the cylinder by controlling the angular velocity of the rotating body, which can be constant in time or time-dependent. Since the numerical control problem was ill-posed, regularization was employed.

Reference [8] focused on the two-dimensional numerical simulation of the unsteady laminar flow past a circular cylinder in a channel, mimicking the effect of the tunnel wall. This study confirms a decrease in wake length and a shift in flow separation further downstream at smaller gaps between the tunnel walls and cylinder.
In [9] an investigation of the laminar flow past an elliptical cylinder confined in a channel. The Lattice Boltzmann (LB) method is used to simulate flow in two dimensions. The LB method with the used boundary conditions is validated in simulations of the incompressible flow past a circular cylinder.

Recently, D.C. Lobaol [10] employed FEM to discretize the governing equations for a viscous incompressible fluid flow around a circular cylinder inside a 2D channel. The fluid flow is described by the Navier–Stokes equations. To tackle these equations minding computational speed the choice is for a simple method called Chorin's projection method for discretizing the Navier-Stokes equations.

The aim of this work is to use CFD package FLUENT to find out the pressure distribution and velocity distribution at upstream and downstream of the cylinder at each time and also to find out mean drag coefficient and lift coefficient at several Reynolds numbers. A Fluent package is used to analyze and calculate these variables.

2-Case Study

Consider a uniform viscous fluid flow past an infinitely cylinder whose a unit diameter. The Reynolds number Re is based on the incident velocity (U) and cylinder diameter (D), where the direction of the free stream flow is normal to the cylinder axis. The user flow in this case is air as shown in Fig. (1).

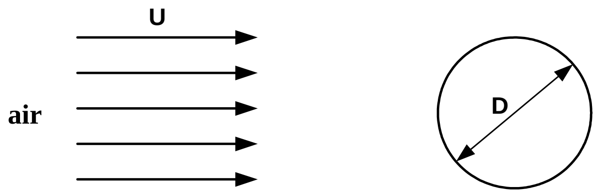


Fig. (1) Flow over a Cylinder

3-Geometrical Model

The flow field around the cylinder is modeled in two dimensions with the axis of the cylinder perpendicular to the direction of flow. A flow domain of Fig.(1) is created surrounding the cylinder. The upstream length is 15 times the radius of the cylinder, and the downstream length is 40 times the radius of the cylinder. The width of the flow domain is 50 times the radius of the cylinder. To facility meshing, a square with a side length of six times the radius of the cylinder is created the cylinder [11]. The square is split into four pieces as shown in Fig. (2). the geometry and dimension of the flow field is chosen to accurately predict the contours of velocity and pressure distribution and the lift and drag coefficients.

4-Mathematical Model

It is considered the following assumption:-

The flow is assumed 2D, unsteady state, laminar and incompressible. Using these assumptions the mass and momentum equations can be written as [12]:-

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \quad \text{.......} \quad (1)$$

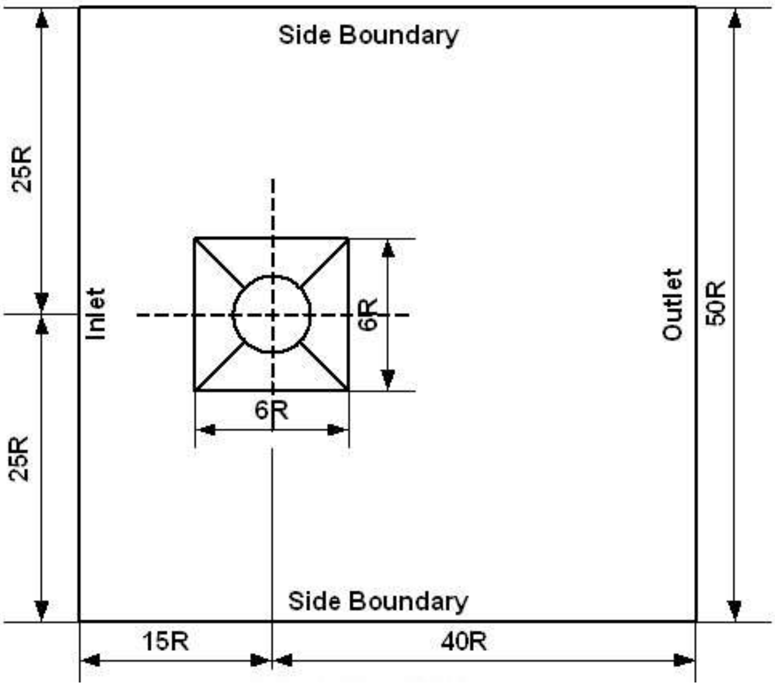


Fig. (2) Flow Domain

$$\rho\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{\partial p}{\partial x}+\mu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \quad \text{.......} \quad (2)$$

$$\rho\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{\partial p}{\partial y}+\mu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \quad \text{.......} \quad (3)$$

The above equations are solved by using FLUENT package after selecting the correct options from the windows of this package.

5- Boundary Conditions

At the inlet boundary (upstream of the cylinder), uniform axial velocity is assumed (U). A constant atmospheric pressure boundary is imposed downstream at the outlet which was placed far from the cylinder (p=101325 Pascal). No slip conditions are used at the rigid walls.

6-Numerical Solution

Gambit program is used to create and mesh the geometry of Fig. (2) Once preprocessing in complete, fluent will be used to solve the flow problem. The mesh generation of Fig.(2) using Gambit program is illustrated in Fig. (3).The 2D unsteady Navier –Stokes was solved by means of a commercial CFD Fluent package based on the finite volume method.

The first-order accurate differencing scheme for time derivative and the PISO algorithm for pressure-velocity coupling is used to allow the use of higher time step size without affecting the stability of the solution. The unsteady formulation is first order implicit. Second order upwind was used for momentum and the allowable error in FLUENT package for continuity, x-velocity and y-velocity is 0.001.

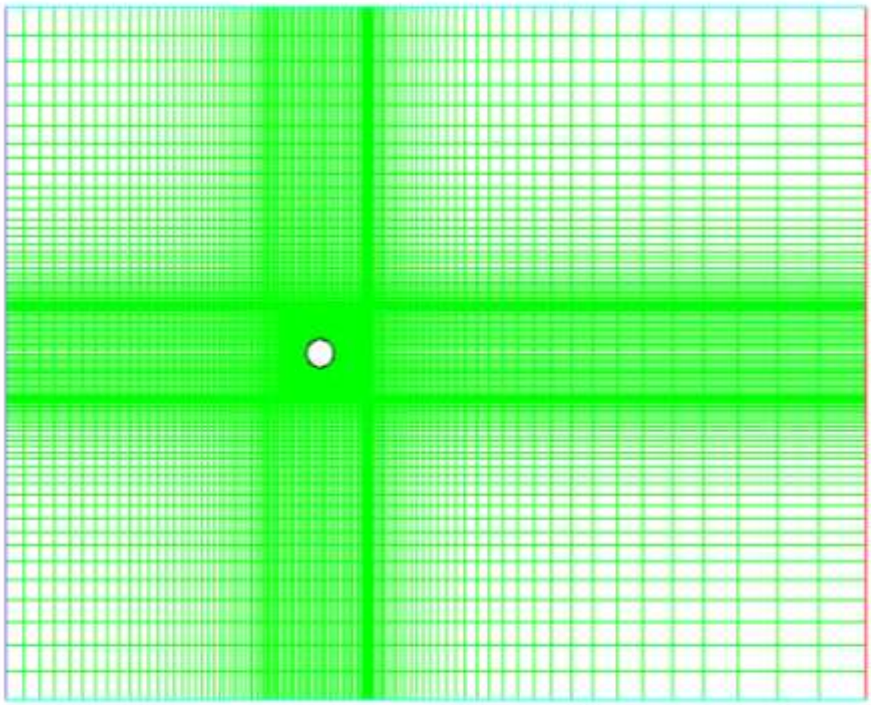


Fig.(3) Meshed Flow Domain Around the Cylinde

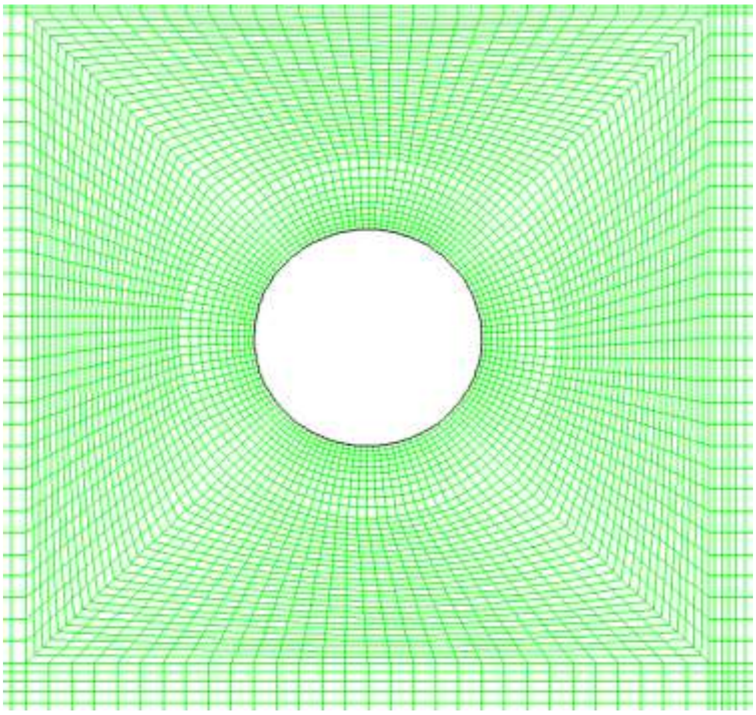


Fig.(4) Mesh Generation Around the Cylinder – Zoom- in View

7-Results and Discussions:

In general, altering the value of Re will be reflected on the flow properties. This is associated with the phenomena of vortex shedding. When Re is sufficiently low (Re≤40) the flow is symmetric and steady. When the Reynolds number is slightly larger, Re<60, the trailing vortex street becomes unstable and develops an unsteady wavy pattern. For Reynolds numbers 60<Re< 300, the Karman vortex shedding occurs in the near wake behind a cylinder due to the flow instability accompanying a large fluctuating pressure and, thus, a periodically oscillating lift force. At higher Reynolds numbers (i.e. Re>300) the flow becomes more turbulent and vortex shedding also occurs.

The drag force on the cylinder is resulting from low-pressure region downstream of the cylinder created by the flow-separation process and generation a wake due to its lower pressure compared with the flow upstream of the body, this wake produces a large pressure differential.

When Re=40, it is found that the flow is symmetric and steady as shown in Fig.(5) and the pressure distribution is uniform as shown in Fig.(6). It is also found that the flow is unsymmetrical and unsteady at Re values of (100, 125, 150and 300) as shown in Figs (7-14).

As the flow progresses upstream of the cylinder, the pressure would decrease and then increase downstream of the cylinder as shown in Figs (15-22), resulting in an increase in free-stream velocity upstream of the cylinder and a decrease downstream.

Fig. (23), Fig. (24), Fig. (25) and Fig. (26) show the relation between the mean drag coefficient and the time at Re=100, Re=125, Re=150 and Re=300respectively and the mean values of drag coefficient at Re=150 is compared with experimental work in [12] to make sure that the use of FLUENT package was true and the agreement is reasonable as shown in Table (1). From these figures the mean values of drag coefficient at Re=100, Re=125, Re=150 and Re=300 are illustrated in table (2), which shows a decrease in drag coefficient when Re is increased.

The values of lift coefficient at each time for Re=100, Re=125, Re=150 and Re=300 are illustrated in Fig. (27), Fig. (28), Fig. (29) and Fig. (30) respectively and these figures can be used to compute the correct value of Strouhal Number ($S_t = \frac{fD}{U}$).

From Fig.(29) the shedding cycle time is 6.2 sec. at Re=150 then f=1/(shedding cycle time)=1/6.2=0.161 Hz ,non-dimensionalized is used in Fluent package (i.e. the diameter is equal 1 & the velocity is equal 1) then S_t=0.161 and this value is compared with experimental work in [12] and the agreement is reasonable as shown in Table (1).

The drag force on the cylinder is resulting from low-pressure region downstream of the cylinder created by the flow-separation process and generation a wake due to its lower pressure compared with flow upstream of the body, this wake produces a large pressure differential.

Table(1): comparison of fluent results and experimental results[12] for unsteady laminar flow over a circular cylinder at re=150

	S_t	C_D
Experimental	0.18	1.1 to 1.4
Fluent	0.161	1.19

Table(2): mean values of C_D at several Re by using Fluent package

Re	100	125	150	300
C_D	1.27	1.22	1.19	1.08

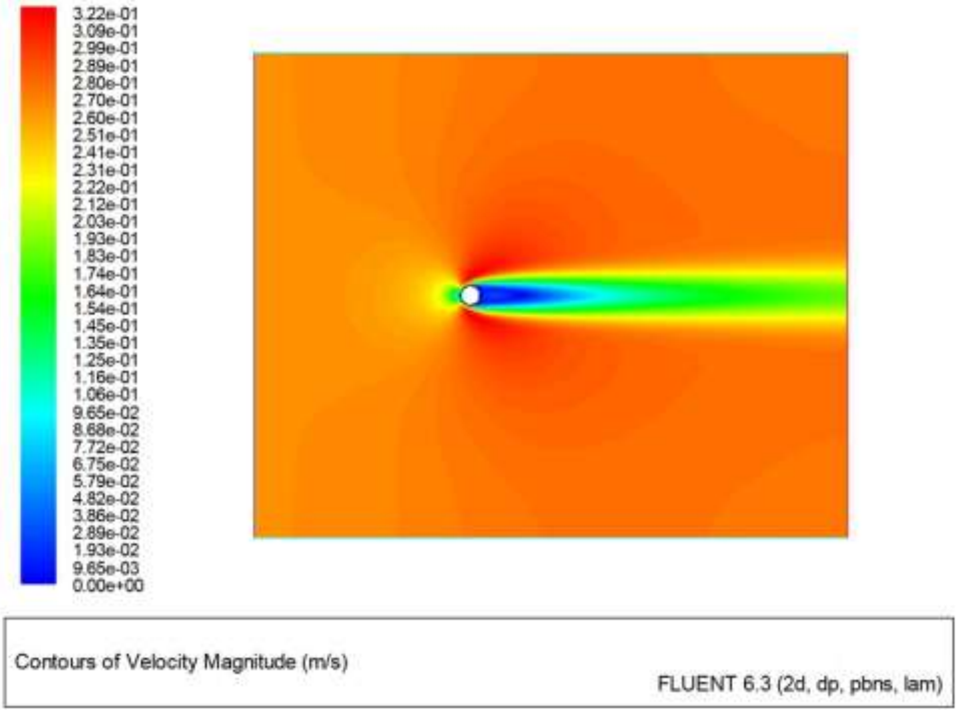


Fig.(5) Contour of Velocity Magnitude(m/s) at Re=40

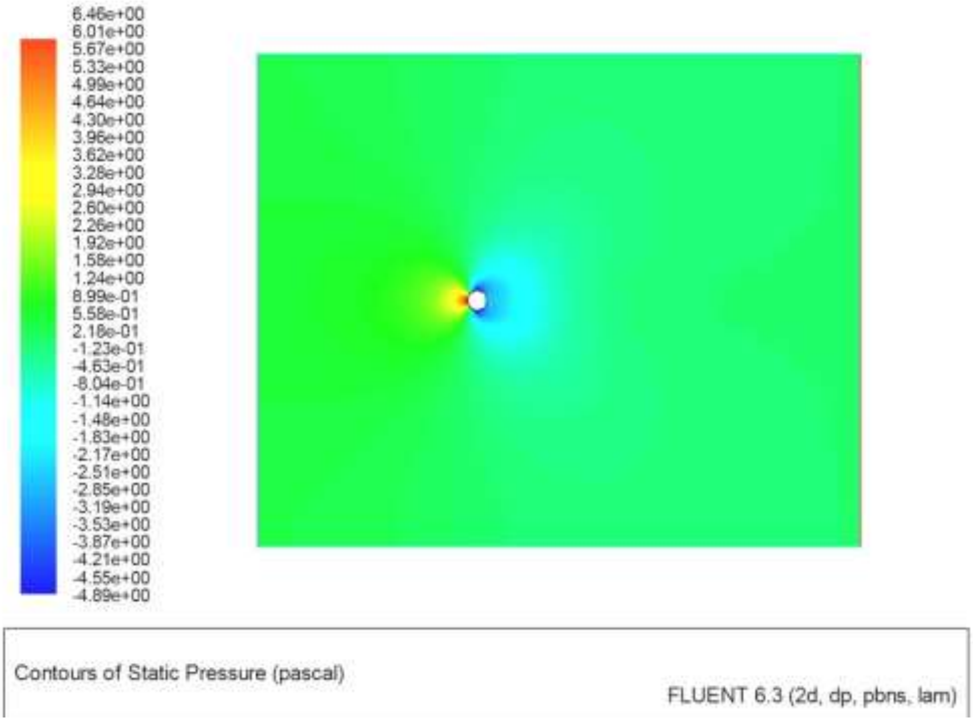
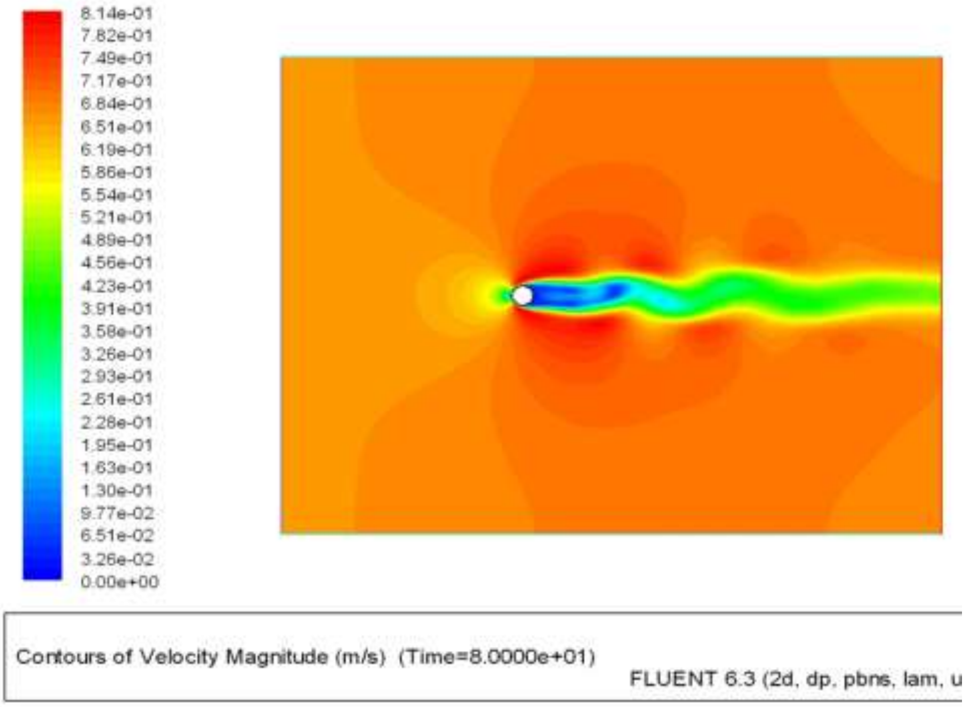
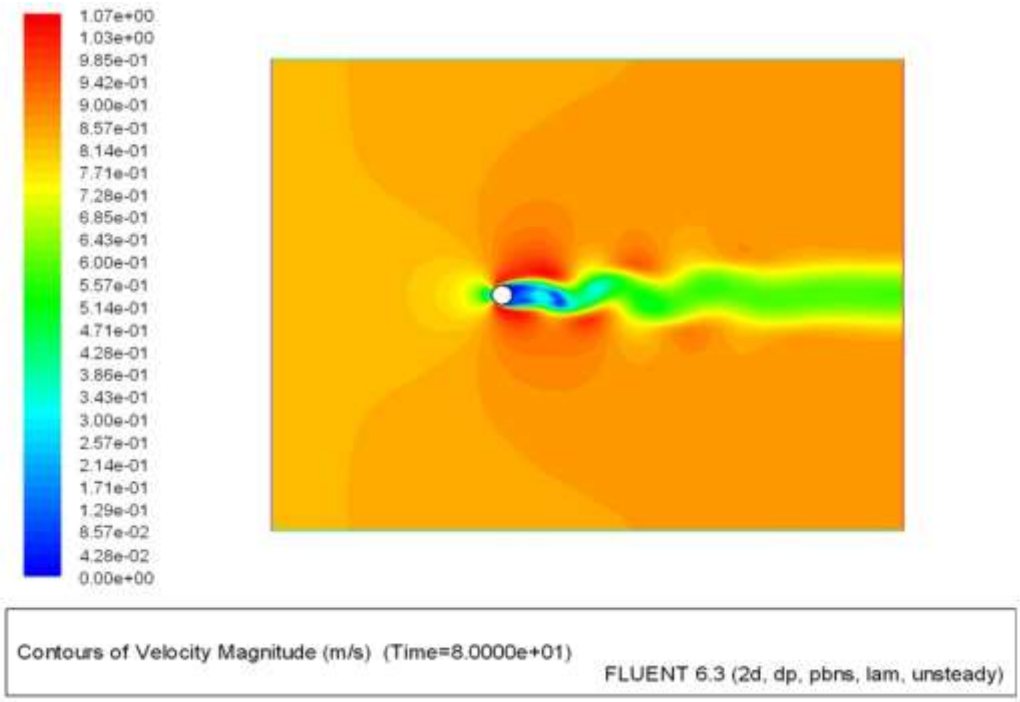


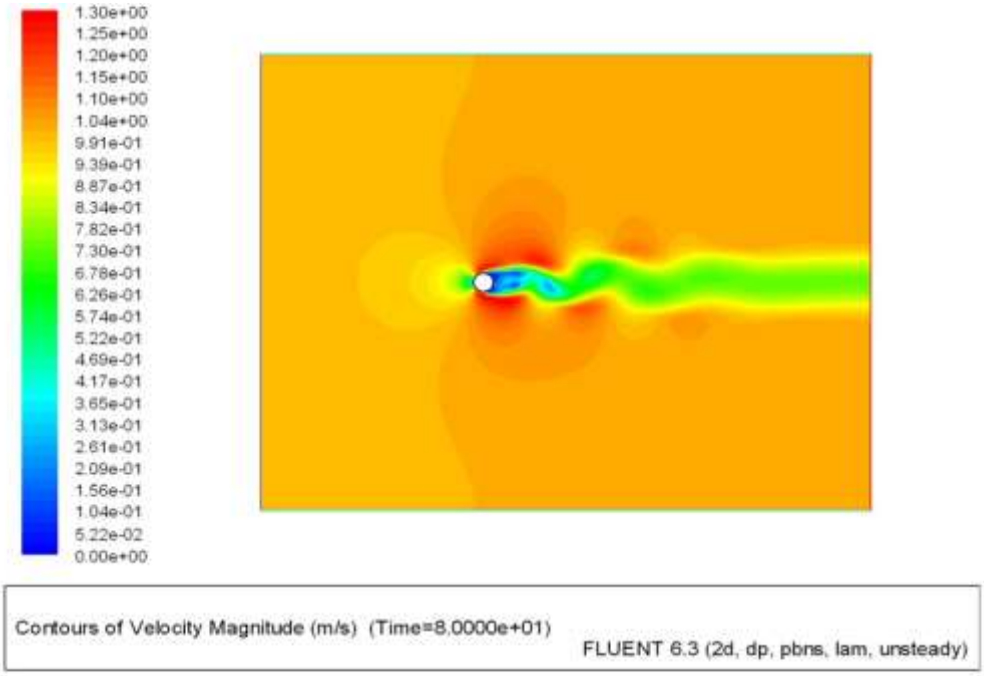
Fig.(6) Contour of Static Pressure Magnitude(pas) at Re=40



Fig(7) Contour of Velocity Magnitude (m/s) at Re=100 & time=80Sec



Fig(8) Contour of Velocity Magnitude (m/s) at Re=125 & time=80 Sec



Fig(9) Contour of Velocity Magnitude (m/s) at Re=150 & time=80 sec

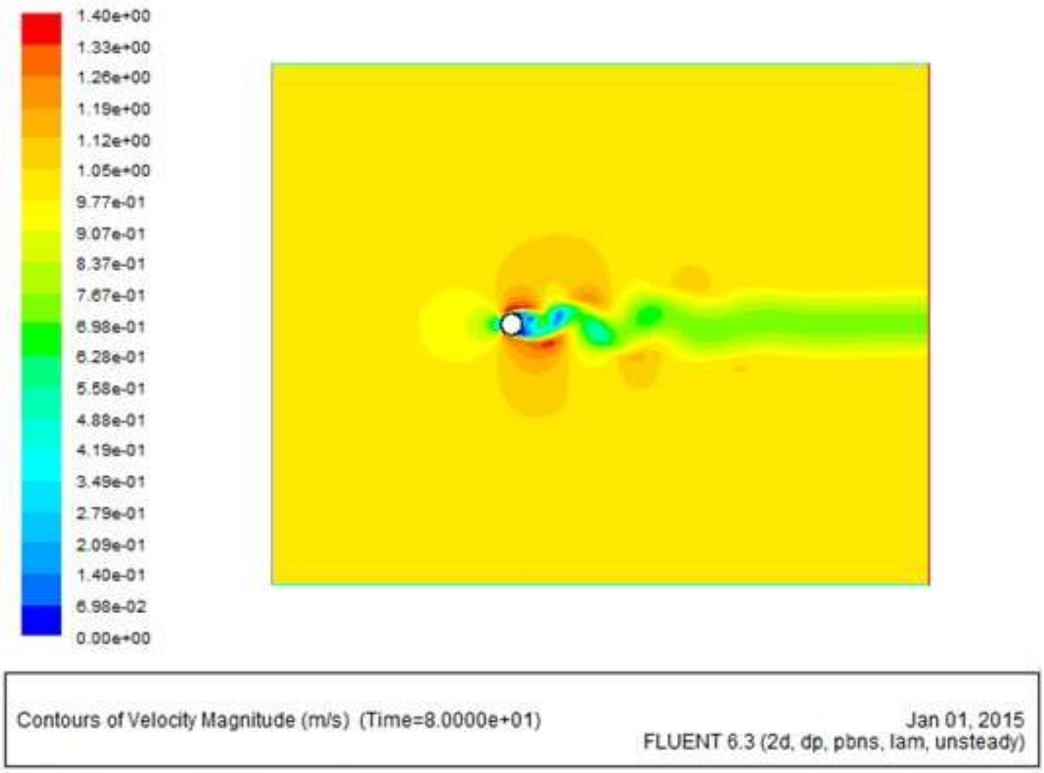


Fig (10) Contour of Velocity Magnitude (m/s) at Re=300 & time 80Sec

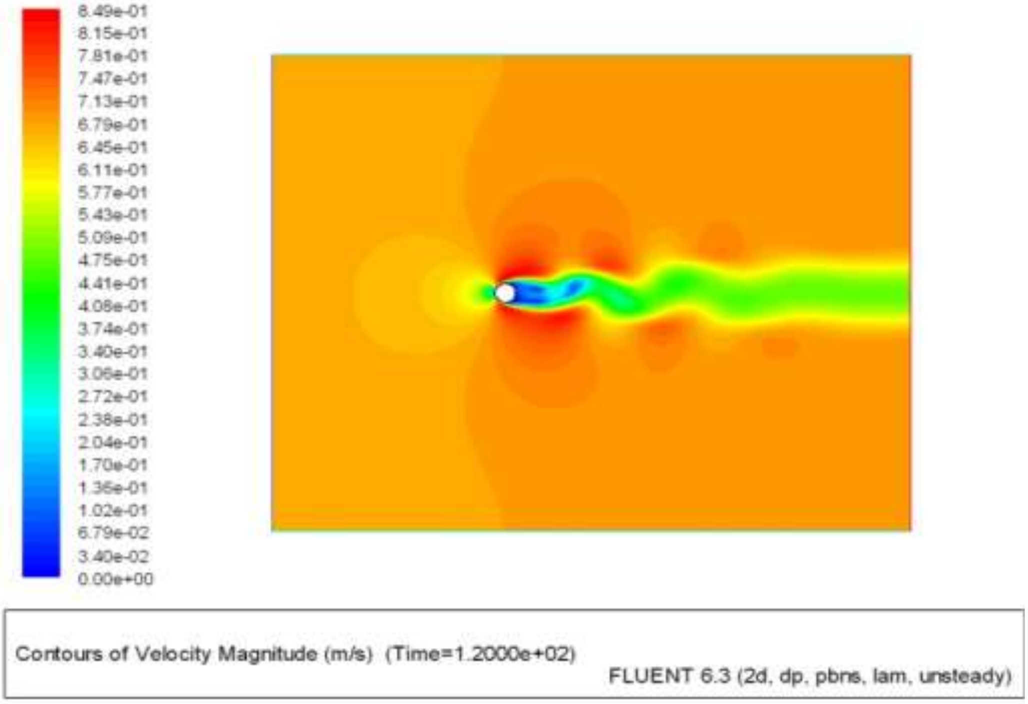


Fig (11) Contour of Velocity Magnitude (m/s) at Re=100 & time 120 Sec

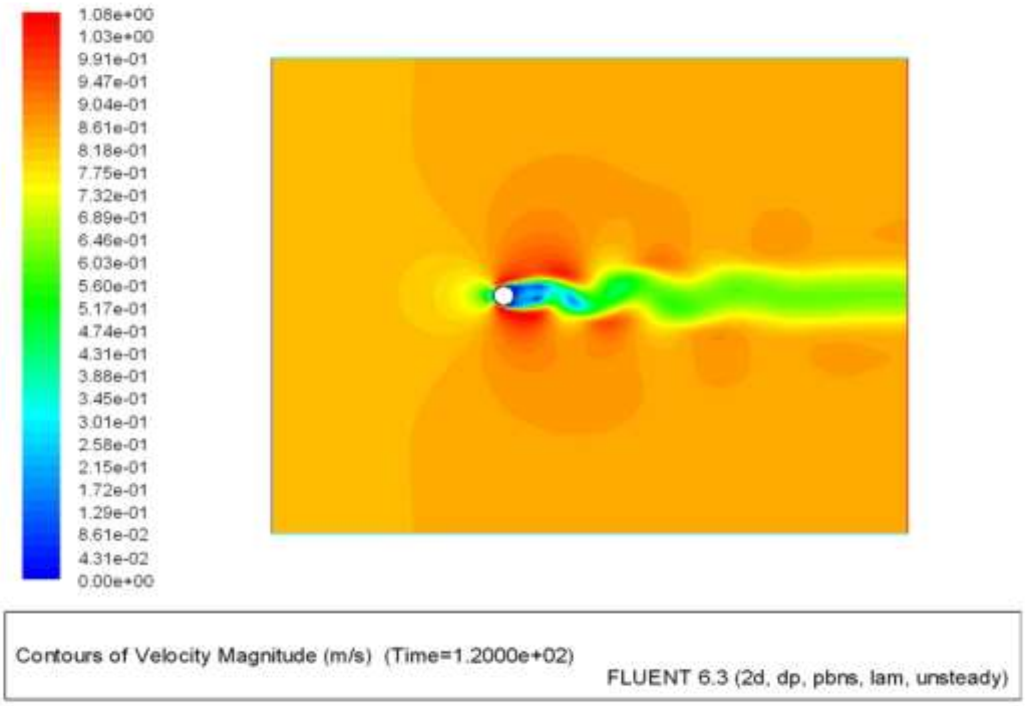


Fig (12) Contour of Velocity Magnitude (m/s) at Re=125 & time 120 Sec

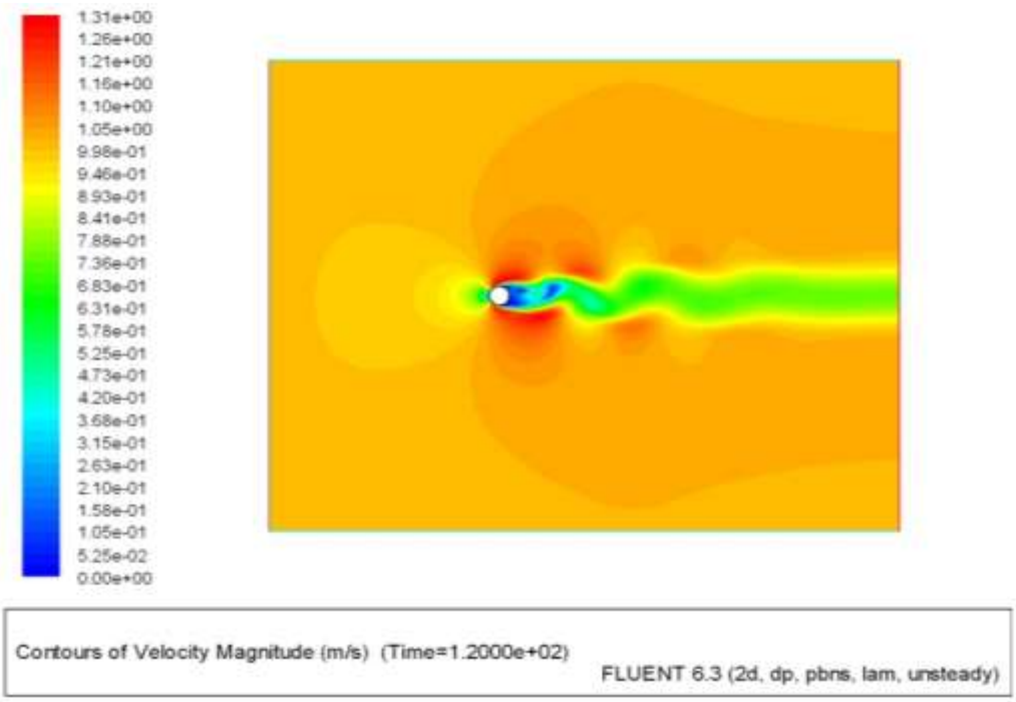


Fig (13) Contour of Velocity Magnitude (m/s) at Re=150 & time 120 Sec

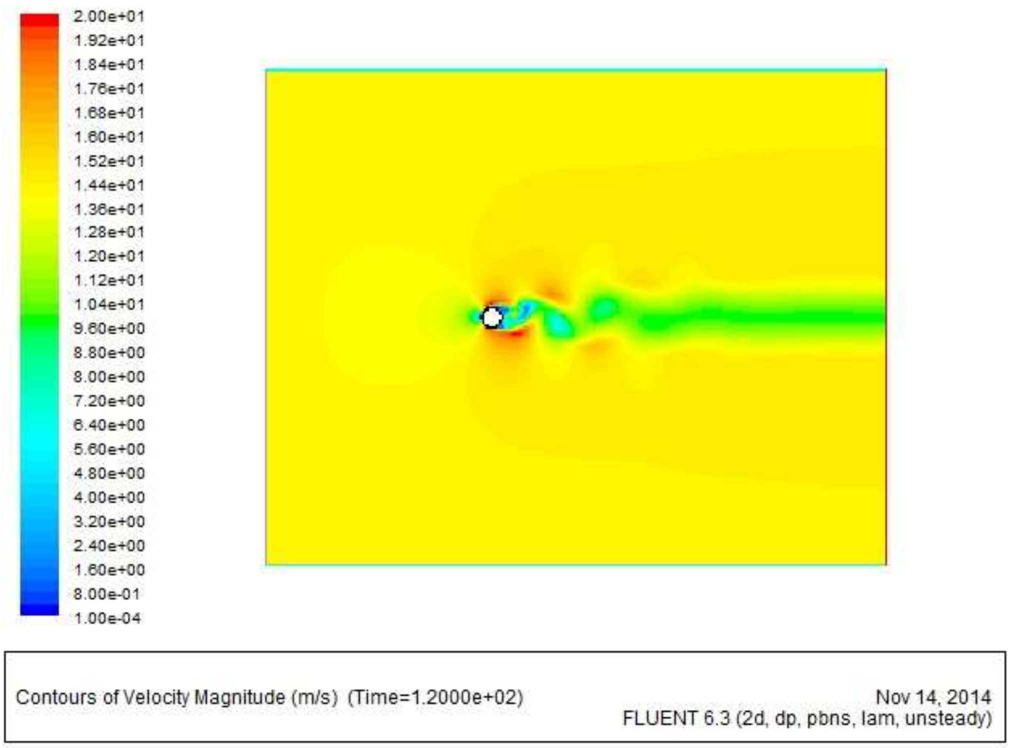


Fig (14) Contour of Velocity Magnitude (m/s) at Re=300 & time 120 Sec

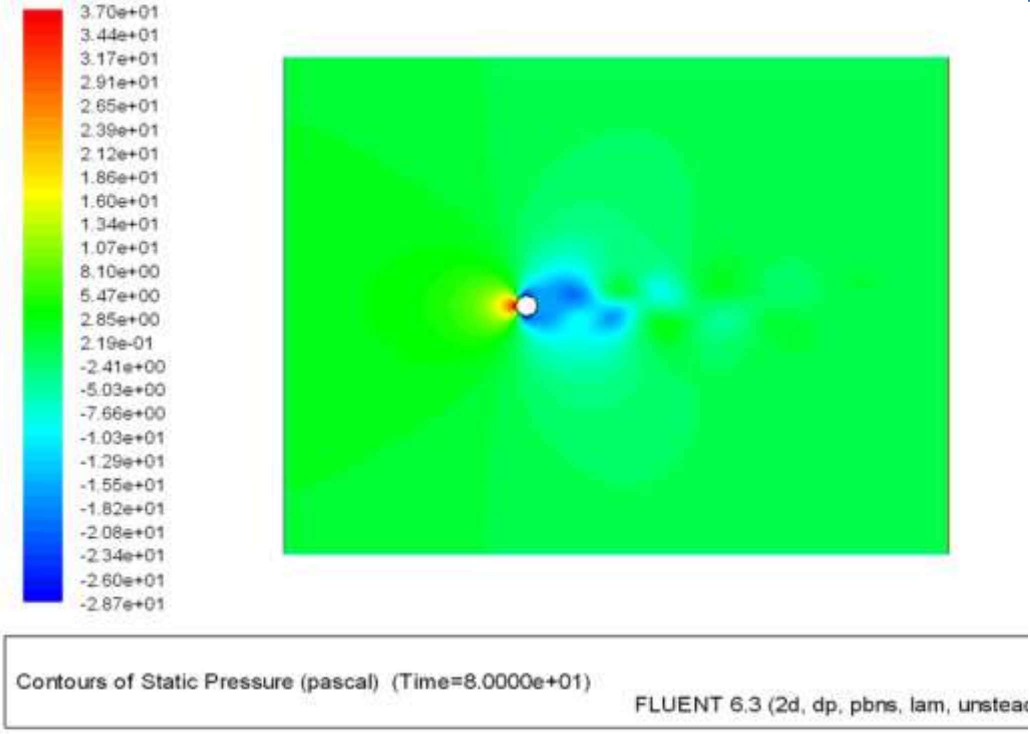


Fig (15) Contour of Static Pressure Magnitude (pas) at Re=100 & time 80 Sec

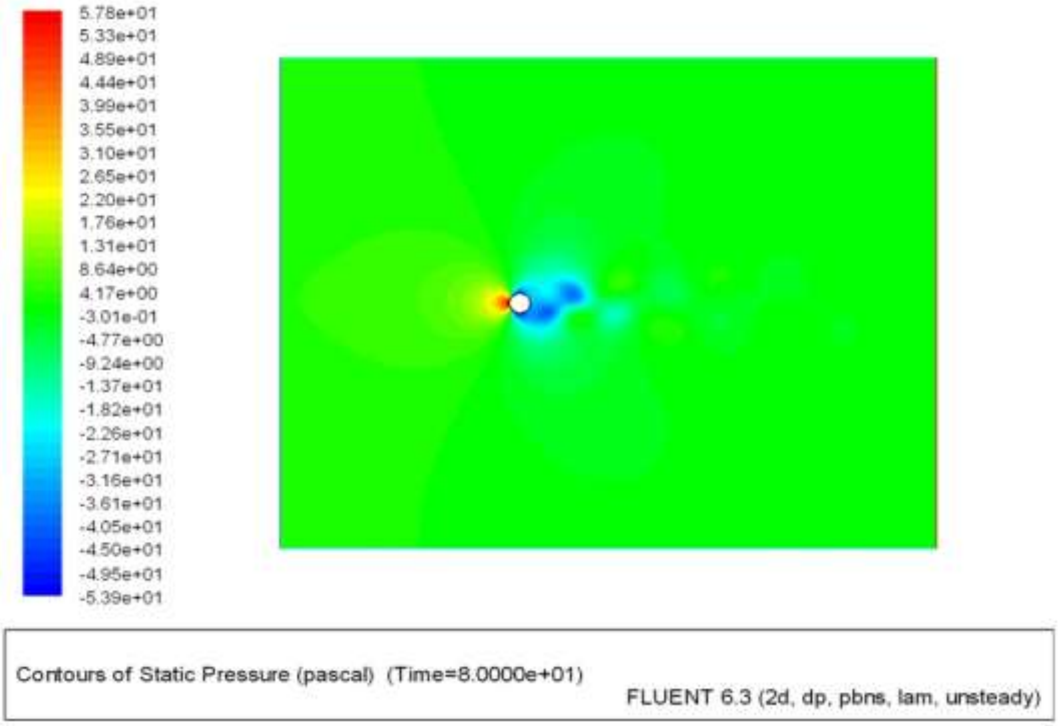


Fig (16) Contour of Static Pressure Magnitude (pas) at Re=125 & time 80 Sec

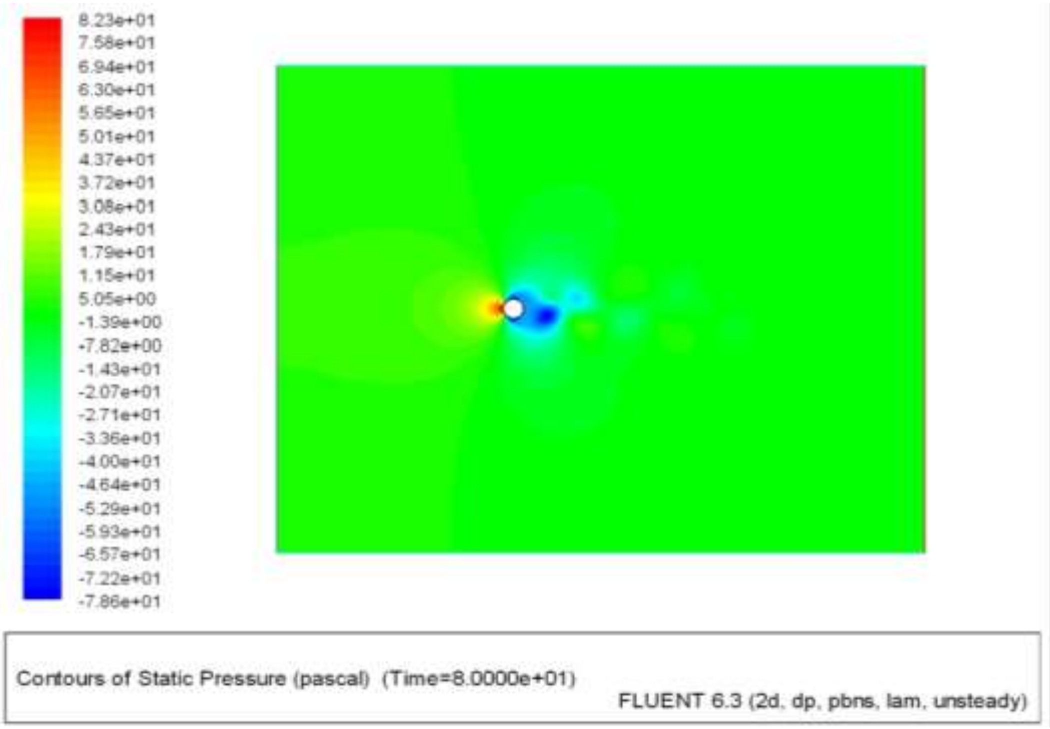


Fig (17) Contour of Static Pressure Magnitude (pas) at Re=150 & time 80 Sec

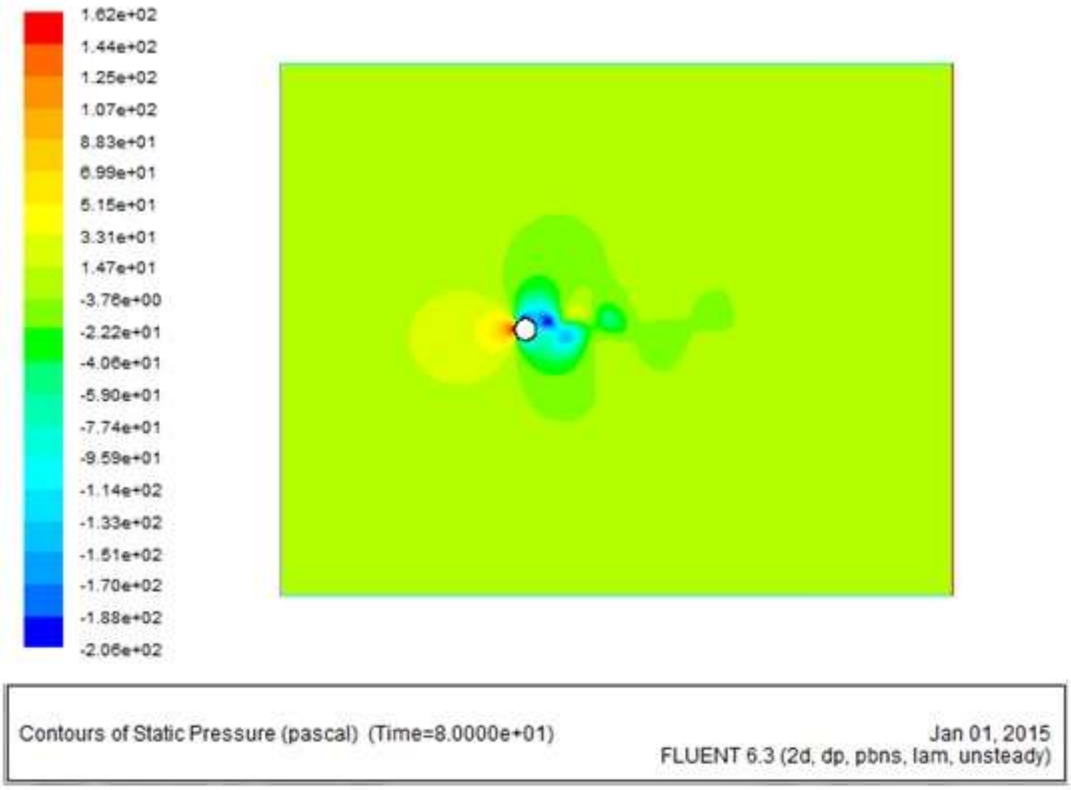


Fig (18) Contour of Static Pressure Magnitude (pas) at Re=300 & time 80 Sec

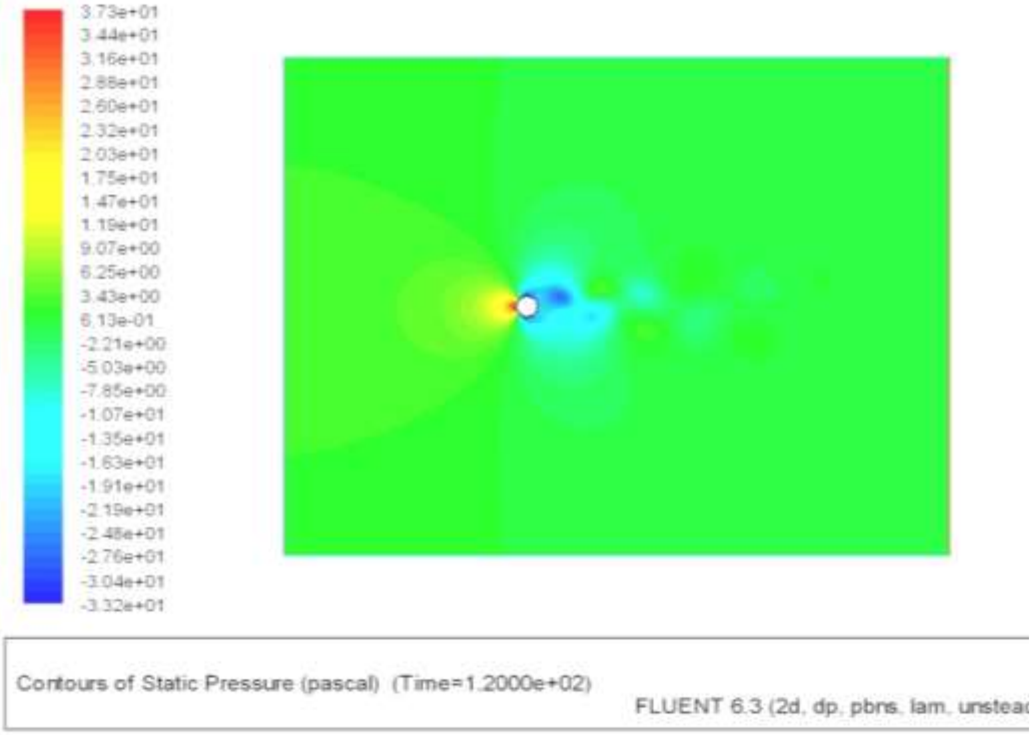


Fig (19) Contour of Static Pressure Magnitude (pas) at Re=100 & time 120 Sec

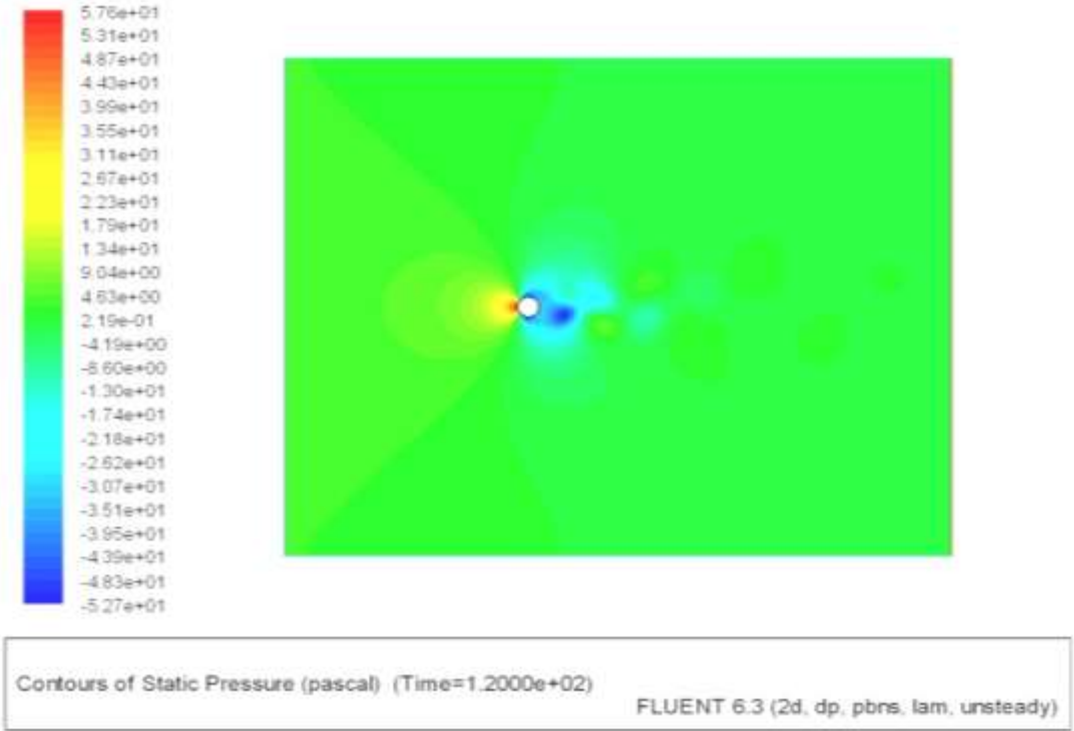


Fig (20) Contour of Static Pressure Magnitude (pas) at Re=125 & time 120 Sec

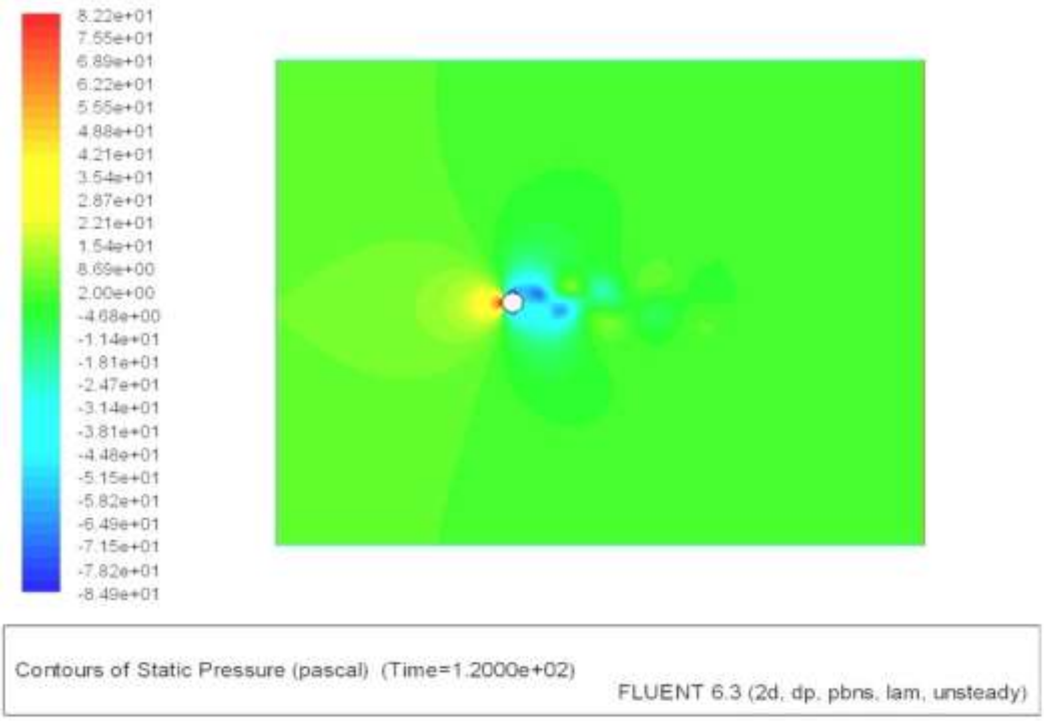
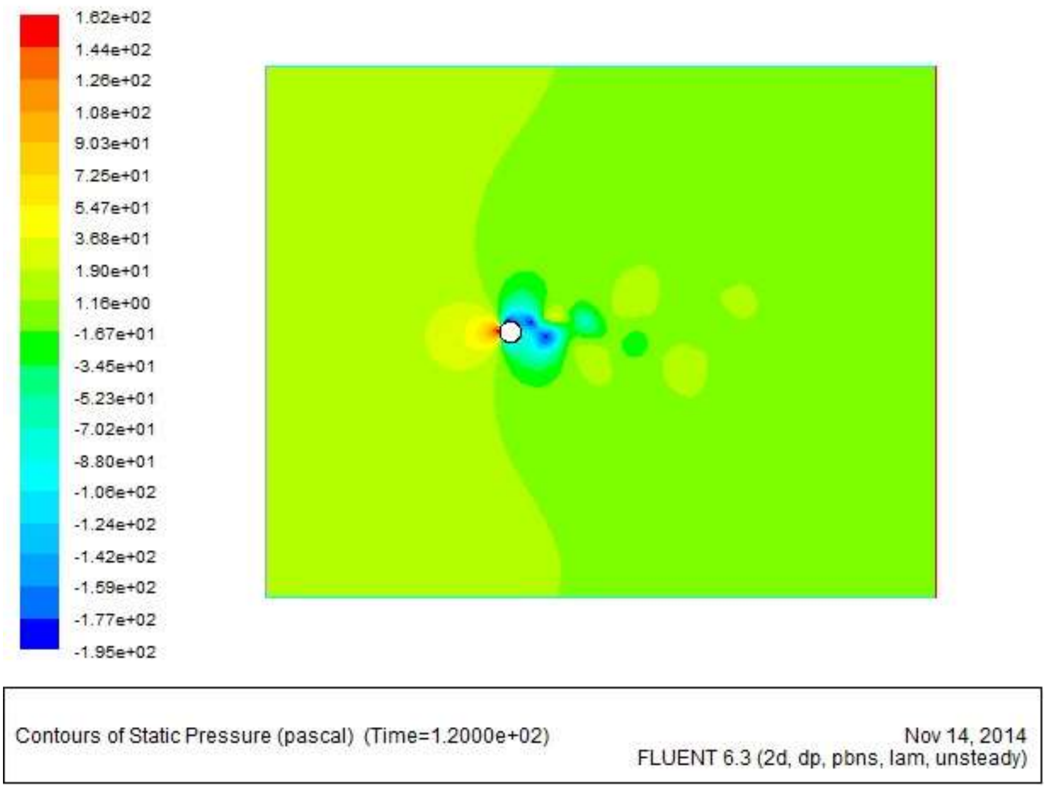


Fig (21) Contour of Static Pressure Magnitude (pas) at Re=150 & time 120 Sec



(22) Contour of Static Pressure Magnitude (pas) at Re=300 & time 120 Sec

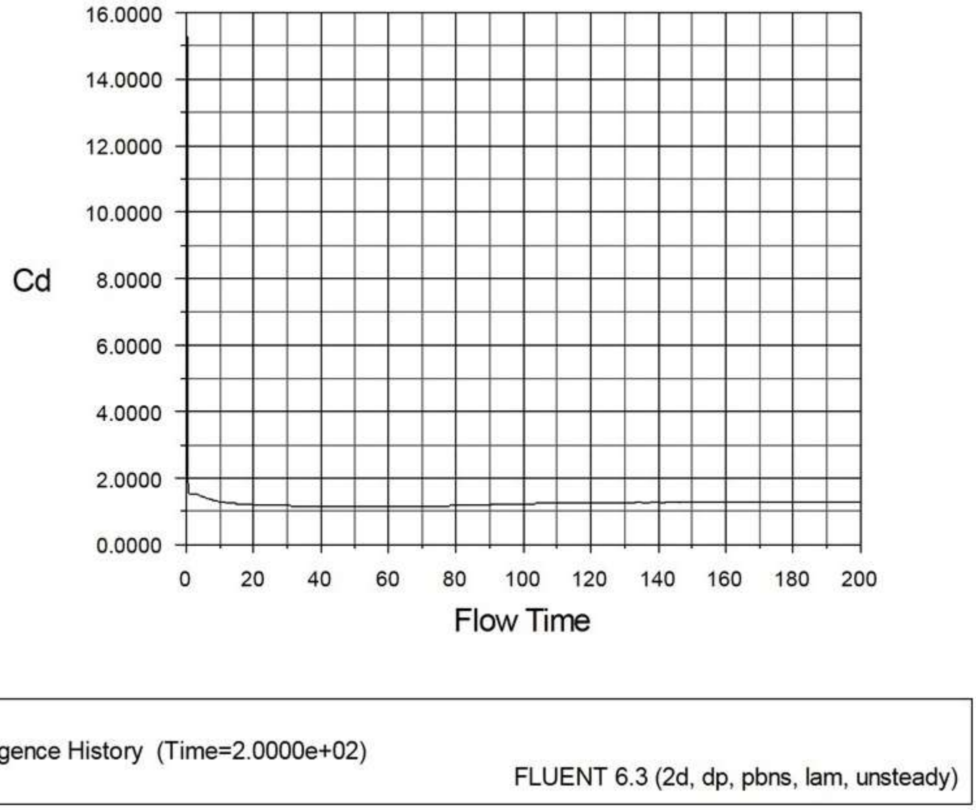


Fig (23) Drag Coefficient history with Time at Re=100

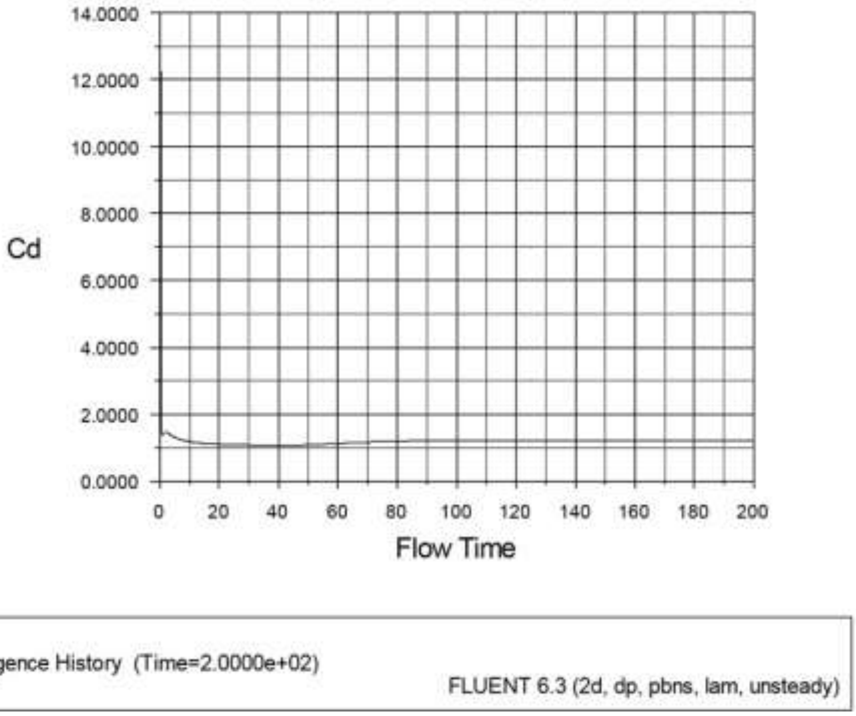


Fig (24) Drag Coefficient History with Time at Re=125

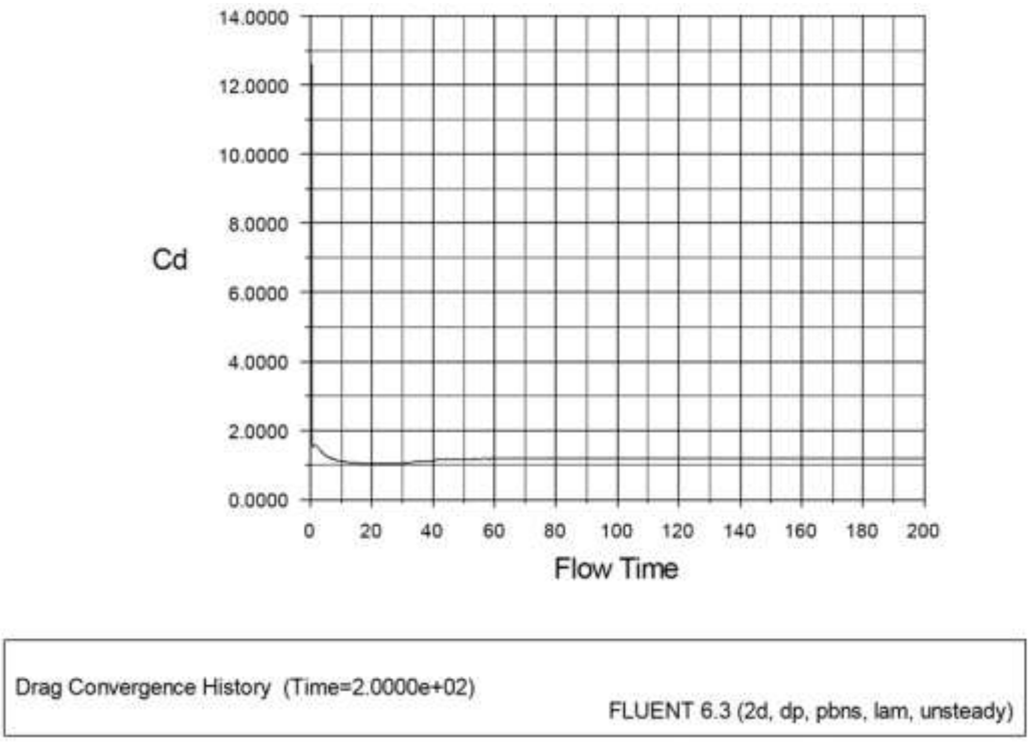


Fig (25) Drag Coefficient History with Time at Re=150

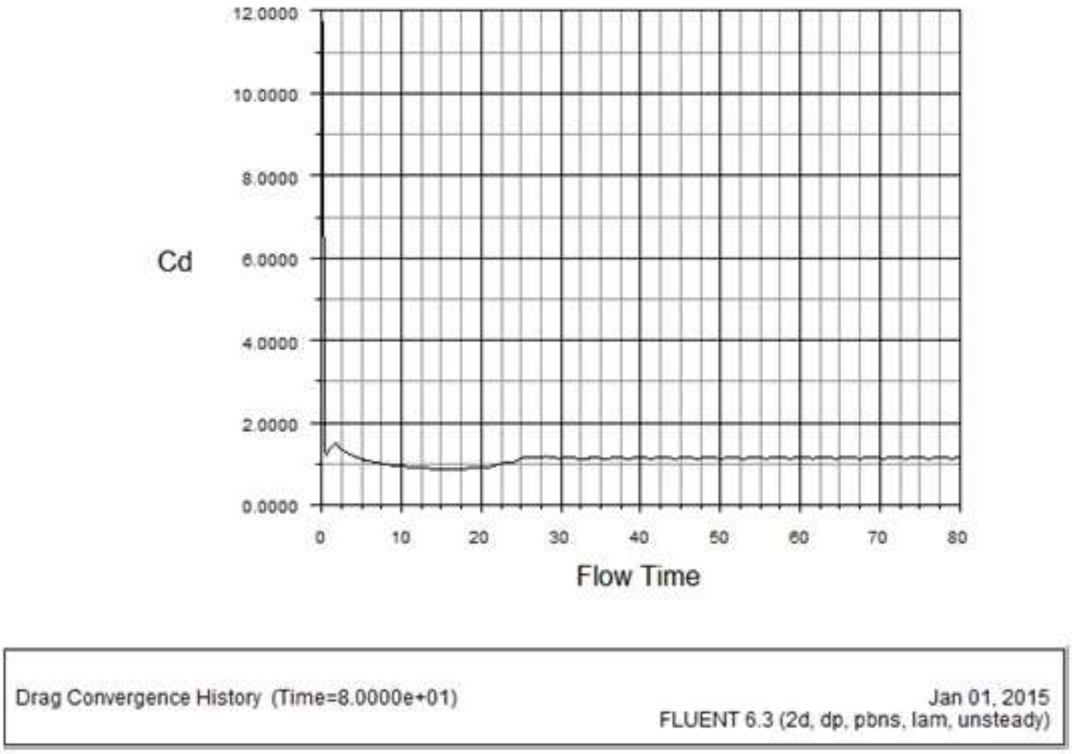


Fig (26) Drag Coefficient History with Time at Re=300

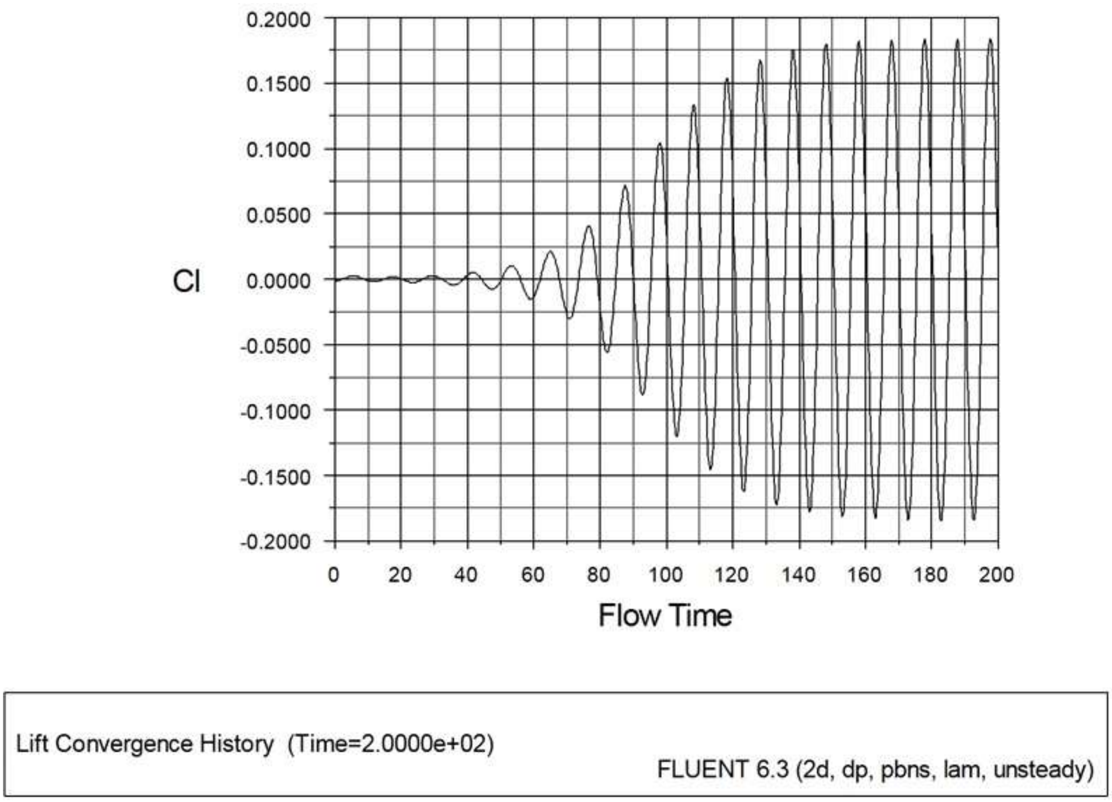


Fig (27) Lift Coefficient History with Time at Re=100

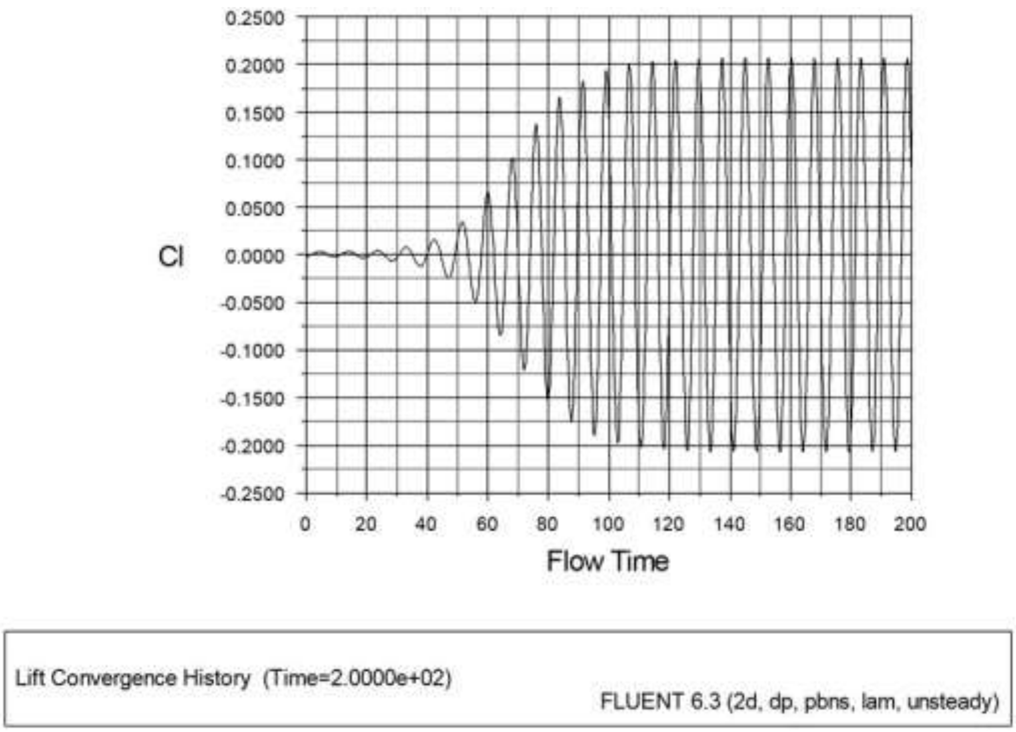


Fig (28) Lift Coefficient History with Time at Re=125

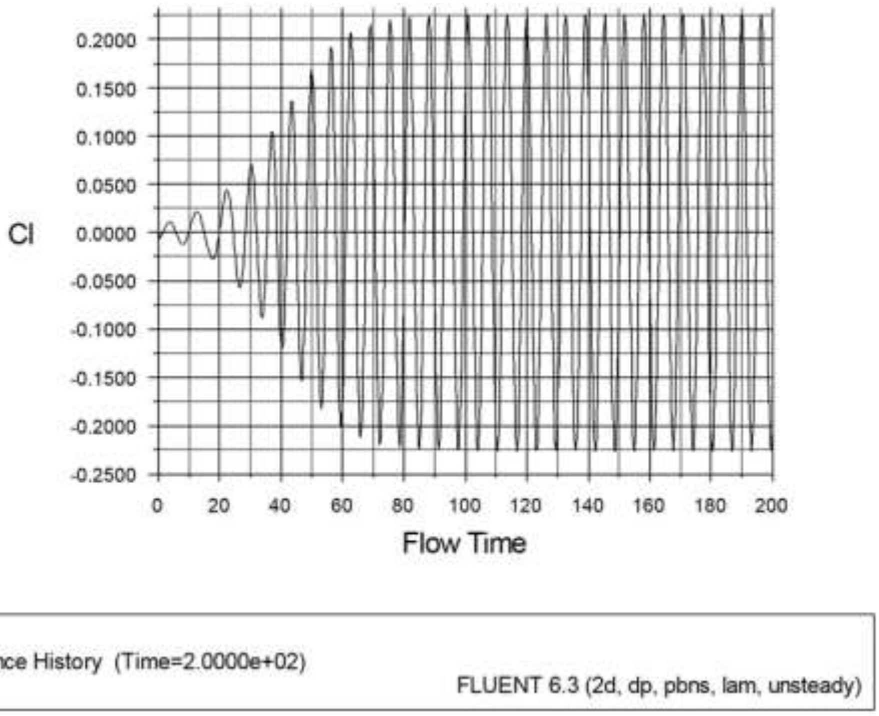


Fig (29) Lift Coefficient History with Time at Re=150

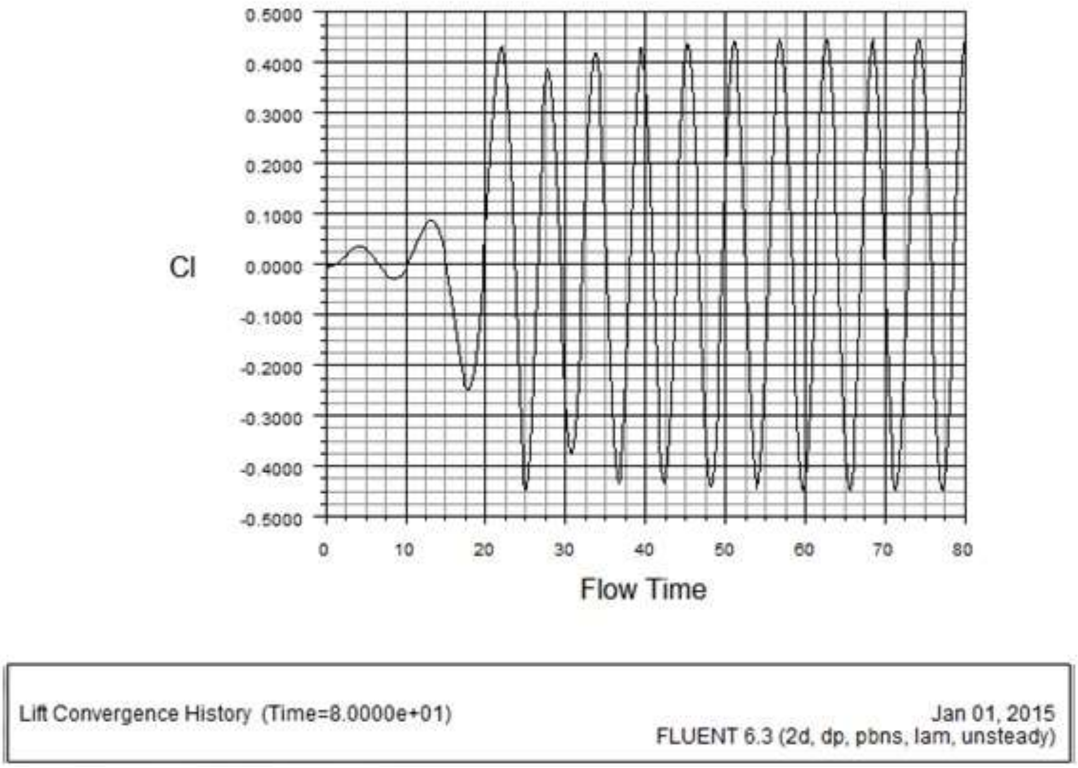


Fig (30) Lift Coefficient History with Time at Re=300

8-Conclusions:

In this paper, flow around a circular cylinder simulated using FLUENT software. The results agree well with the available experiment results of other investigators. Flow parameters such as drag and lift coefficient, pressure and velocity contours are also investigated. The results of FLUENT software was shown that the upstream cylinder is found higher compared with downstream cylinder. As Re is increased, the frequency of the vortex shedding, the amplitude of the oscillation of the lift coefficient is increased. In the other hand, the drag coefficient is slightly decreased as Re is increased. Also the flow is strongly unsteady and unsymmetrical at Re>60.

9-Nomnchrature

Symbol	*Description*	*Unit*
A	The Shortest Side of the Cylinder	m
B	The Longest Side of the Cylinder	m
C_D	Drag Force Coefficient	-------
C_L	Lift Force Coefficient	-------
D	Diameter of Cylinder	m
F	Vortex shedding Frequency	Hz
P	Pressure	pas
R	Radius of Cylinder	m

Symbol	*Description*	*Unit*
Re	Reynold Number	-------
S_t	Strouhal Number	-------
T	Time	Sec
U	Velocity in x-direction	m/Sec
V	Velocity in y-direction	m/Sec
µ	Viscosity	Kg/m.Sec
P	Density	Kg/m^3
A	Angle of incidence	Degree

10-References

1- Wikipedia, the Free Encyclopedia "**Vortex Shedding**", 2015, http://www.wikipedia.org.

2- William,J.,"**Experiment 3-flow past a Circular Cylinder** ",AOE 3054,Expeimental Methods, Course Manual, Department of Aerospace and Ocean Engineering,2007,http://www.aoe.vt.edu.

3- Sanjay,M.,"**Suppression of vortex Shedding Using Control Cylinder**", The Seventh Asian Congress of Fluid Mechanics, Department of Aerospace engineering, Dec 18-12-1997,Chennai (Mardas),P.315-318.

4- Sohankar,A.,Norberg,C. and Davidson, L."**A Numerical Study of Unsteady Two-Dimensional Flow Around Rectangular Cylinders at Incidence**", Internal Report ,Nr.96/25,Department of Thermo and Fluid Dynamics, Chalmers University of Technology ,Gothenburg, Sweden ,1996.

5- F.,Barners,"**Vortex shedding in The Wake of a Rotating Circular Cylinder at Low Reynolds Number**",J.Phys.D:Appl.Phys.33,L141-L144,2000.

6- Scholl,F."**Effect of Pressure Unsteadiness on Vortex Shedding Frequency from Dual Bluff Body**", Project is Supported by SRF for ROCS, SEM and Zhejiang Provincial Natural Science Foundation (599086), 2002,http://www.fluid.power.net.

7- C.H., I.M. and Z. L.”**Suppression of Vortex Shedding for Flow Around a Circular Cylinder Using Optimal Control**”, Int. J. Number .Meth. Fluids; 38:34-69, 2002.

8- Rajani B.N, R.V.P Gowda And P. Ranjan "**Numerical Simulation of Flow past a Circular Cylinder with Varying Tunnel Height to Cylinder Diameter at Re 40**",International Journal Of Computational Engineering Research (ijceronline.com) Vol. 3 Issue. 1, January, 2013.

9- *M. Taeibi-1,2Rahni, 3V. Esfahanian and 2M.Salari*"**Investigation of Flow Around a confined Elliptical Cylinder Using the Lattice Boltzmann Method**”, Middle-East Journal of Scientific Research 15 (1): 08-13, IDOSI Publications, 2013.

10- D.C. Lobão1"**FINITE ELEMENT MODELING EXPERIMENTS OF UNSTEADY FLOW AROUND A CIRCULAR CYLINDER"** Blucher Mechanical Engineering Proceedings, vol. 1, num. 1, May 2014. www.proceedings.blucher.com.br/evento/10wccm

11- PhilonNet Engineering Solutions” **Flow Over a Cylinder"**, April 16, 2007, http://www.philonnet.gr.

12- Yunus A. Cengel and John M. Cimbala**” Fluid Mechanics Fundamentals and Applications”**, McGraw-Hill Publishing Company,2006.