

# VISUALIZING TRANSIENT FLOW OVER A RIGHT CIRCULAR CYLINDER

**Kamal S Kumar**

Fourth Year B.Tech

Swami Vivekanand University

## Abstract:

This simulation is to study the changes in flow properties over the surface of a right circular cylinder placed in a laminar flow. The main objective of this simulation is to collect data on the variation of flow parameters (pressure and velocity) and force coefficients ( $C_d$  and  $C_l$ ) about the cylinder at different given mediums and inlet velocities as well as to visualize Kármán Vortex Street generated due to such interactions. This study is carried out using various FLOSS software.

*Keywords: Laminar flow, Karman Vortex Street, cylindrical body*

## 1. Introduction

If the flow parameters of a flow field depend not only on the position in the coordinate system used to describe it but also on time, such a flow is said to be transient or unsteady. Such a flow field can be defined as:

$$p = p(x, y, z, t)$$

$$U = U(x, y, z, t)$$

Unlike steady flow (which is time independent), unsteady flows show fluctuation of flow field with time thus exhibiting certain characteristic phenomena.

One such phenomenon can be observed when a circular cylinder (or a blunt body) is placed in a fluid ( $Re > 90$ ), we obtain a repeating pattern of swirling vortices, caused by a process known as vortex shedding, which

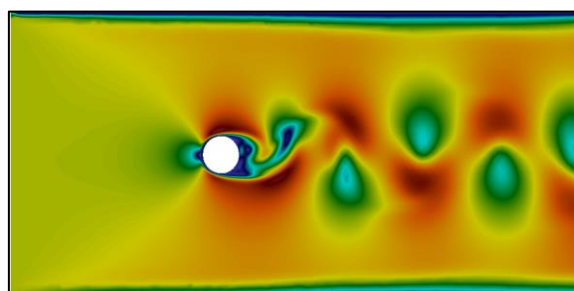


Fig.1 Kármán vortex street

is responsible for the unsteady separation of flow field about it. This phenomenon is known as Kármán vortex street (named after Aerospace Engineer Theodore von Kármán, father of supersonic flight).

## 2. Problem Statement

The objective of this case study is to simulate a Kármán vortex street which occur for Reynolds number between 180 and 200.

## 3. Governing Equations

This case study is based on Reynolds-averaged Navier–Stokes equations (or RANS equations or RAS equations), are time-averaged equations of motion for fluid flow. For a stationary flow of an incompressible Newtonian fluid, these equations can be written in Einstein notation in Cartesian coordinates as:

$$\rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho \bar{f}_i + \frac{\partial}{\partial x_j} \left[ -\bar{p} \delta_{ij} + \mu \left( \frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_i} \right) - \rho \overline{u'_i u'_j} \right]$$

The Navier-Stokes equation is supplemented with the incompressibility condition:

$$\nabla \cdot \mathbf{U} = 0$$

## 4. Simulation Procedure

A total of 10 cases are to be simulated by changing the inlet/freestream velocity and the kinematic viscosity of the working fluid, rest of the parameters are common throughout the cases/subcases. As mentioned in the transport properties, the simulation is divided into three cases (each case has distinct kinematic viscosity) each with 3-4 subcases (each subcase has distinct inlet/freestream velocity).

### 4.1 Geometry and Mesh

In this study the geometry and meshing of the domain is designed using Gmsh software. The geometry consists of a rectangular boundary region with dimension  $4u \times 3u$ , with a circle of radius  $0.1u$  placed in it. This geometry is given a depth of  $0.1u$ .

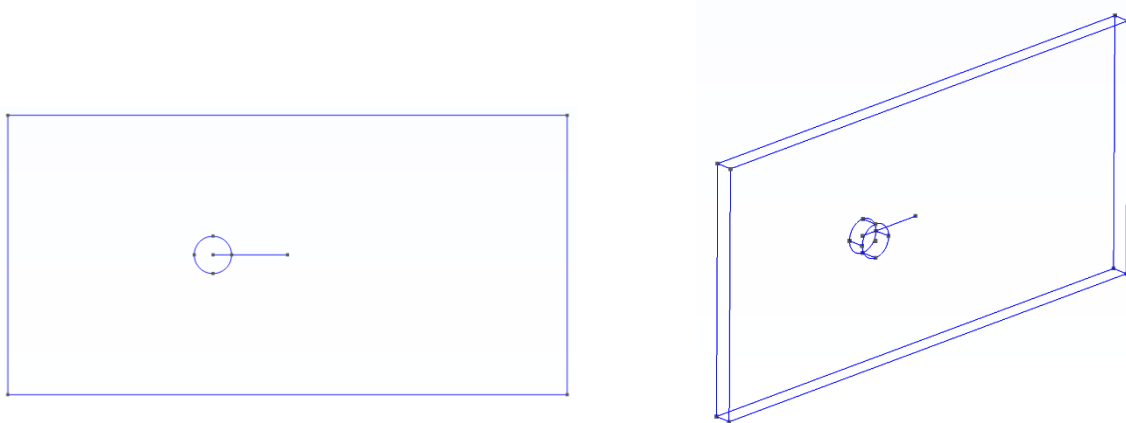


Fig.2 Front and Isometric view of domain geometry.

An unstructured mesh is generated in the domain, with attractor set to refine the region around the cylinder. The mesh generated in Gmsh is converted into OpenFOAM usable format using the command

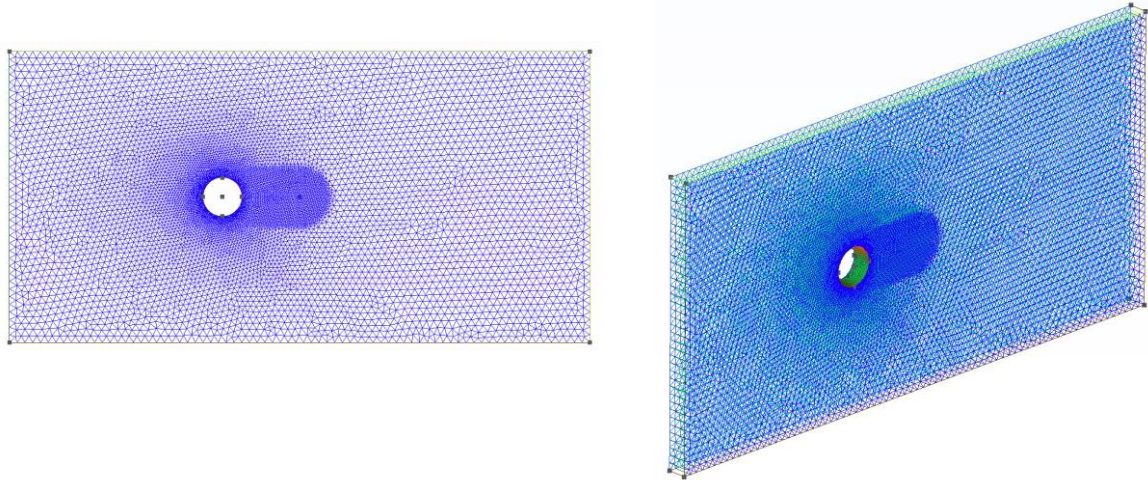


Fig.3 Front and Isometric view of domain mesh.

“gmshToFoam”. This creates the “polyMesh” folder, in the “constant” directory and the necessary mesh files are generated.

## 4.2 Initial and Boundary Conditions

- Boundary Condition/Initial Condition (BC/IC) Dictionaries

Boundary	Pressure (p)	Velocity (U)
Inlet	zeroGradient	fixedValue
Outlet	fixedValue	zeroGradient
Top and Bottom Walls	zeroGradient	fixedValue
Front and Back	empty	empty
Cylinder Wall	zeroGradient	fixedValue

- Transport Properties

Transport model is set as Newtonian, with kinematic viscosity ( $\nu$ ) set as:

CASE I -  $1.00 \times 10^{-6} \text{ m}^2 \text{ s}^{-1}$

CASE II -  $5.555 \times 10^{-4} \text{ m}^2 \text{ s}^{-1}$

CASE III -  $1.18 \times 10^{-3} \text{ m}^2 \text{ s}^{-1}$

- Turbulence Properties

Simulation type and RAS model is set as laminar.

## 4.3 Solver

Since this study demands for transient incompressible flow field, the pisoFoam solver is used to simulate the case. The solver uses the PISO (Pressure-Implicit with Splitting of Operators) algorithm to solve the continuity equation:

$$\nabla \cdot \mathbf{U} = 0$$

and momentum equation:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla \cdot (\mathbf{u}\mathbf{u}) - \nabla \cdot (\nu \nabla \mathbf{u}) = -\nabla p$$

## 4.4 Result extraction

The plots are generated using GNUPLLOT and the visualization of simulation is done using PARAVIEW.

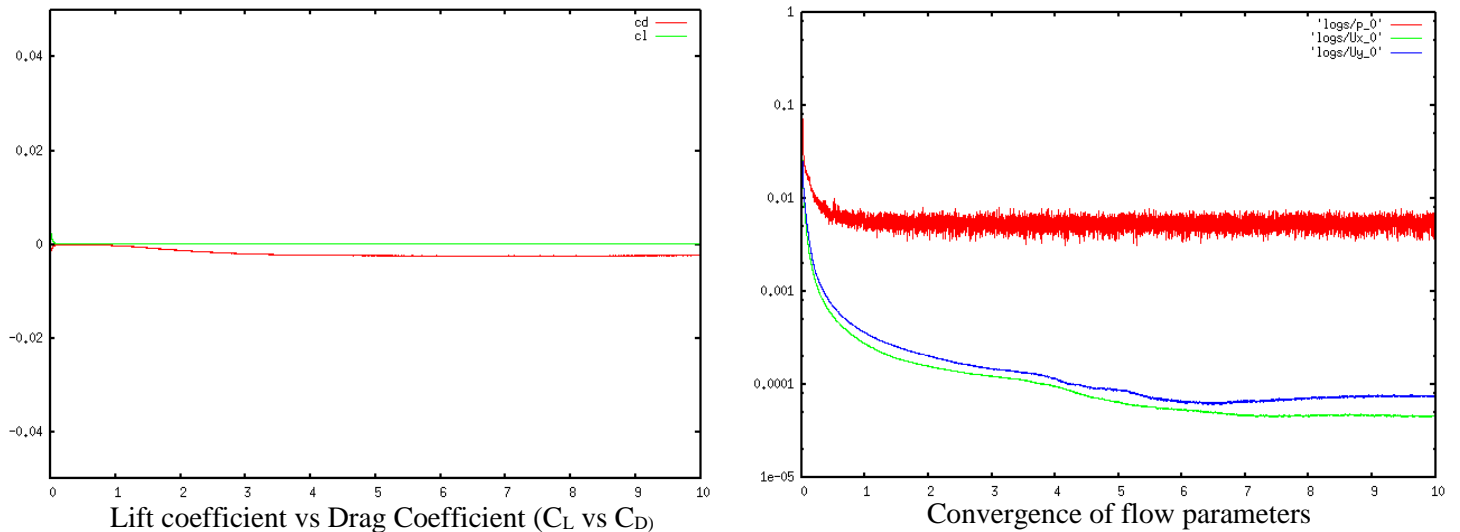
## 5. Result

### CASE I

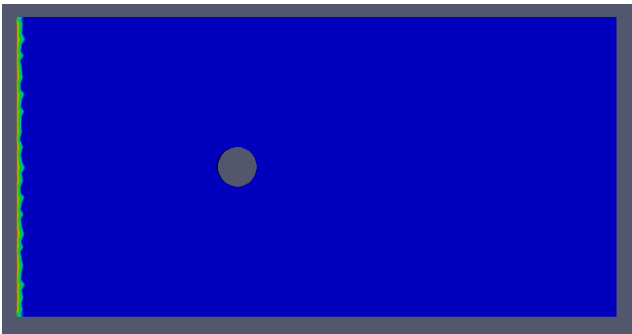
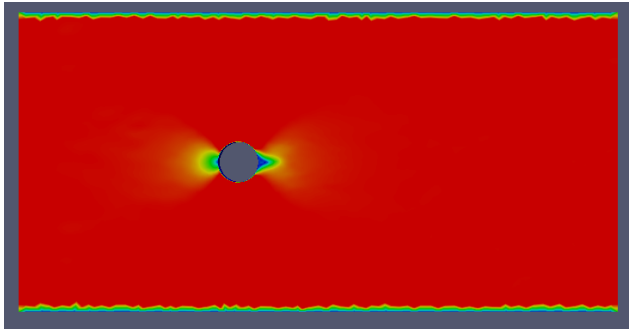
For the first case the kinematic viscosity is set at  $1.00 \times 10^{-6} \text{ m}^2 \text{ s}^{-1}$ . The fluid inlet velocity is set at  $0.5 \text{ m s}^{-1}$ ,  $1.0 \text{ m s}^{-1}$  and  $1.2 \text{ m s}^{-1}$ . The simulation results for each subcase is as follows:

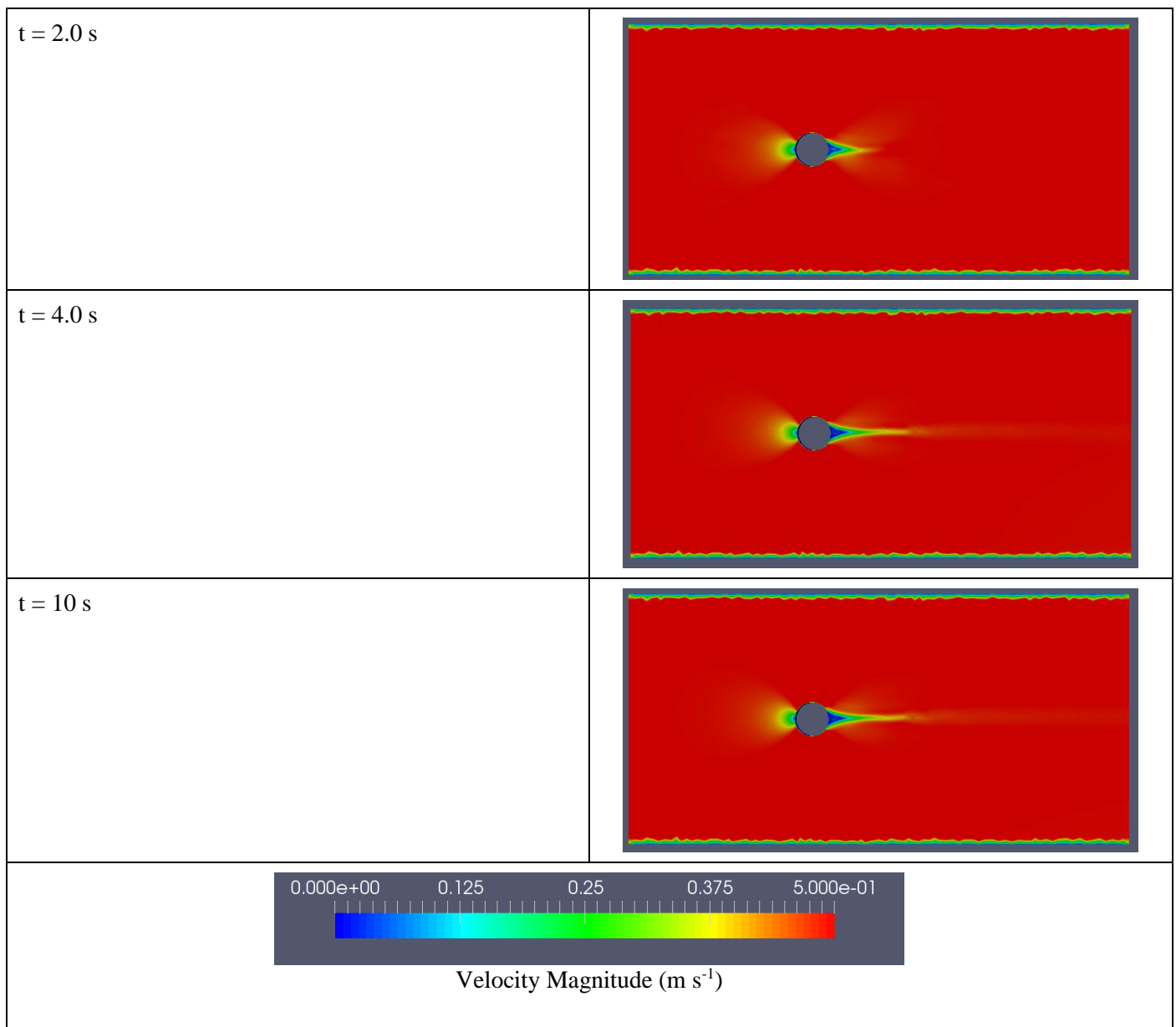
- At inlet velocity  $0.5 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

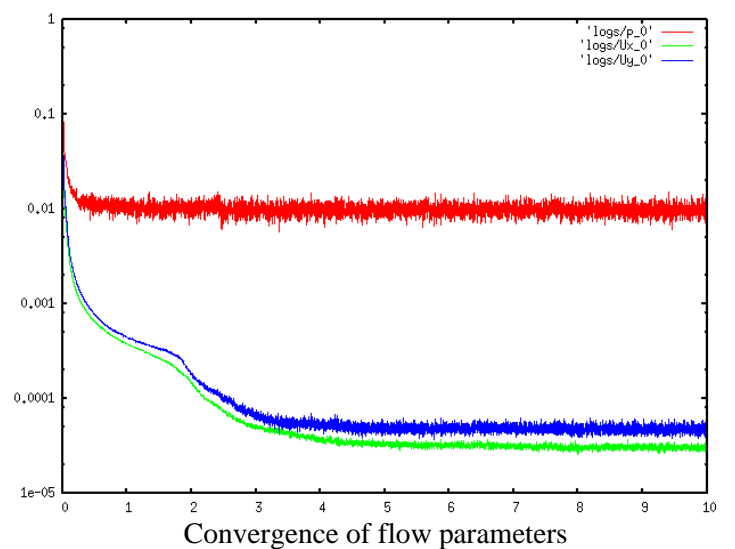
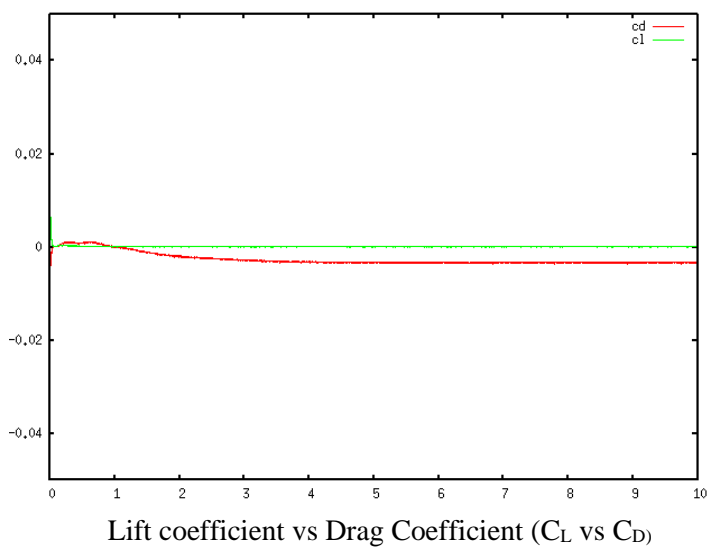
Time t (s)	Result
t = 0 s	
t = 0.5 s	



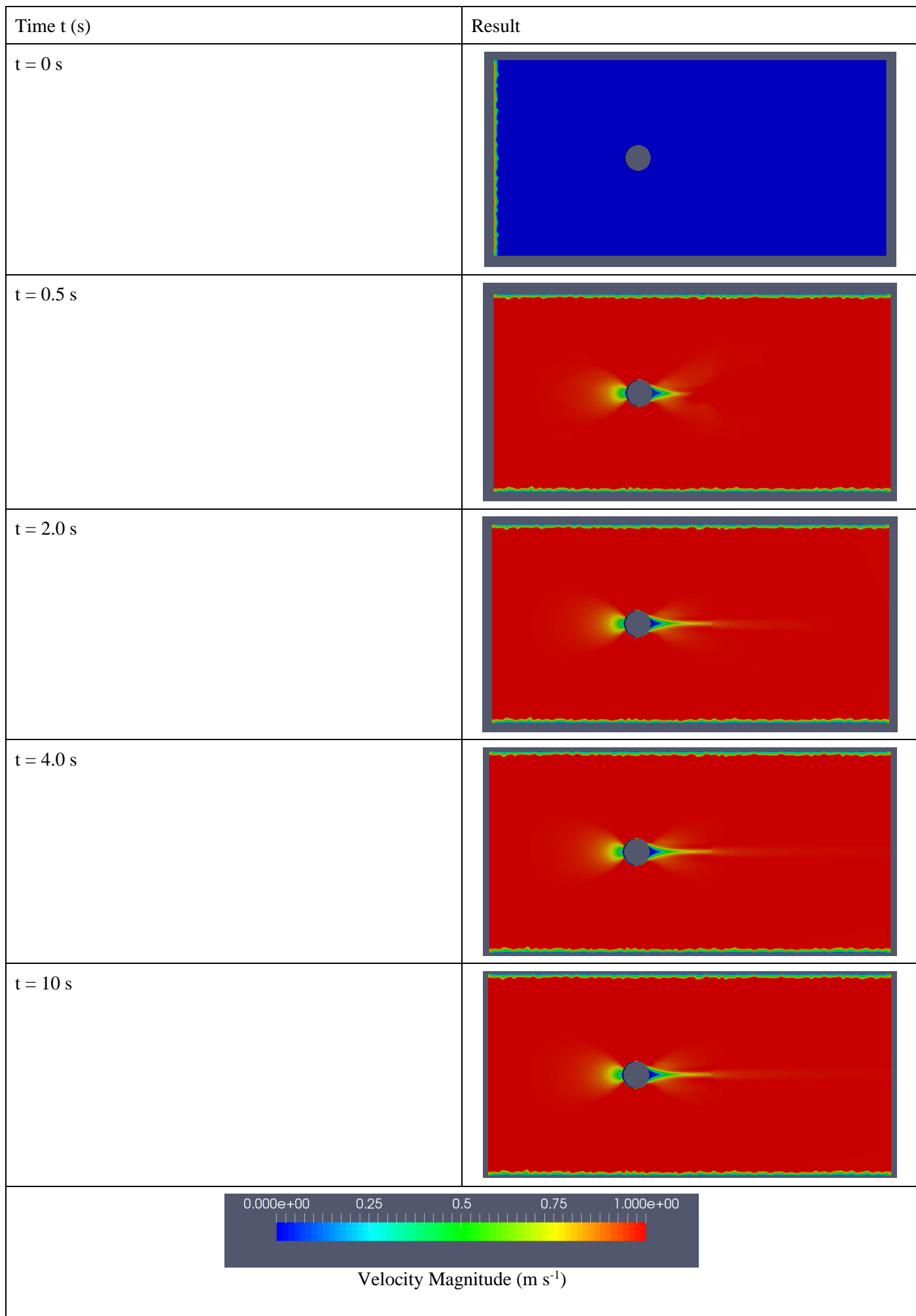
At  $0.5 \text{ m s}^{-1}$  the fluid converges without showcasing any vortex shedding pattern.

- At inlet velocity  $1.0 \text{ m s}^{-1}$

Plots:



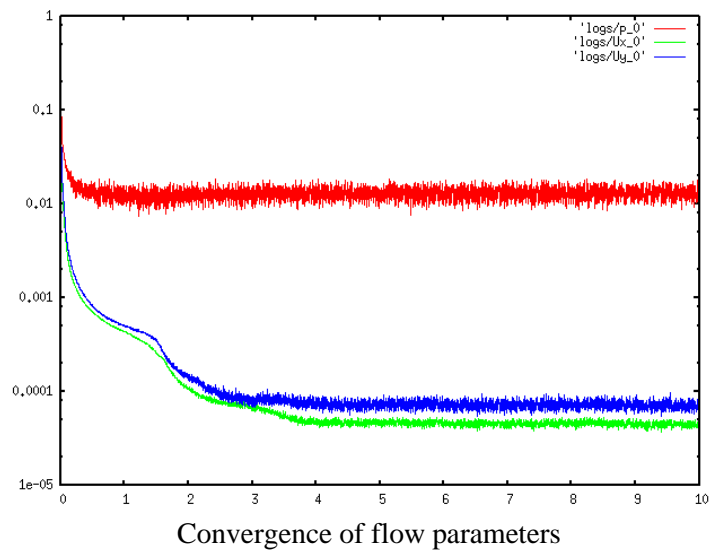
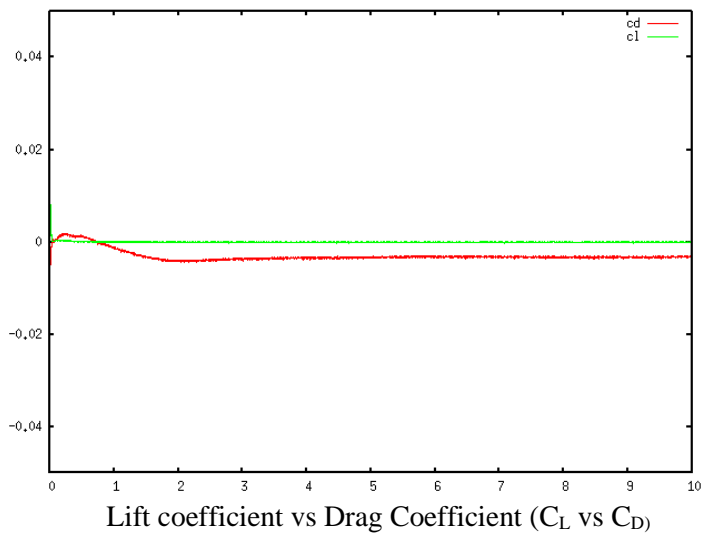
Simulation results for velocity profiles at different time intervals:



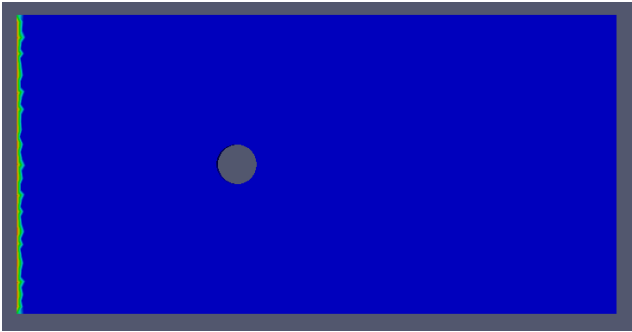
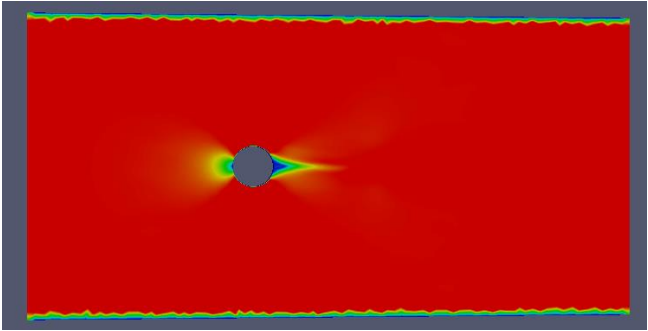
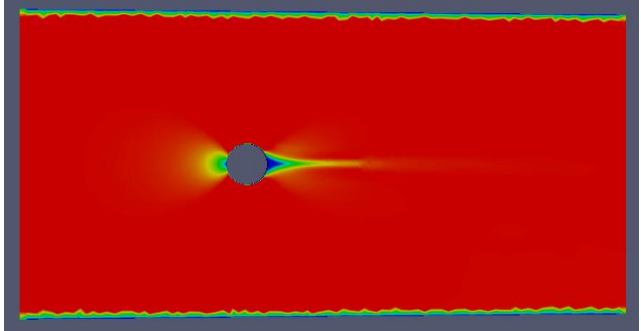
At  $1.0 \text{ m s}^{-1}$  the fluid converges without showcasing any vortex shedding pattern.

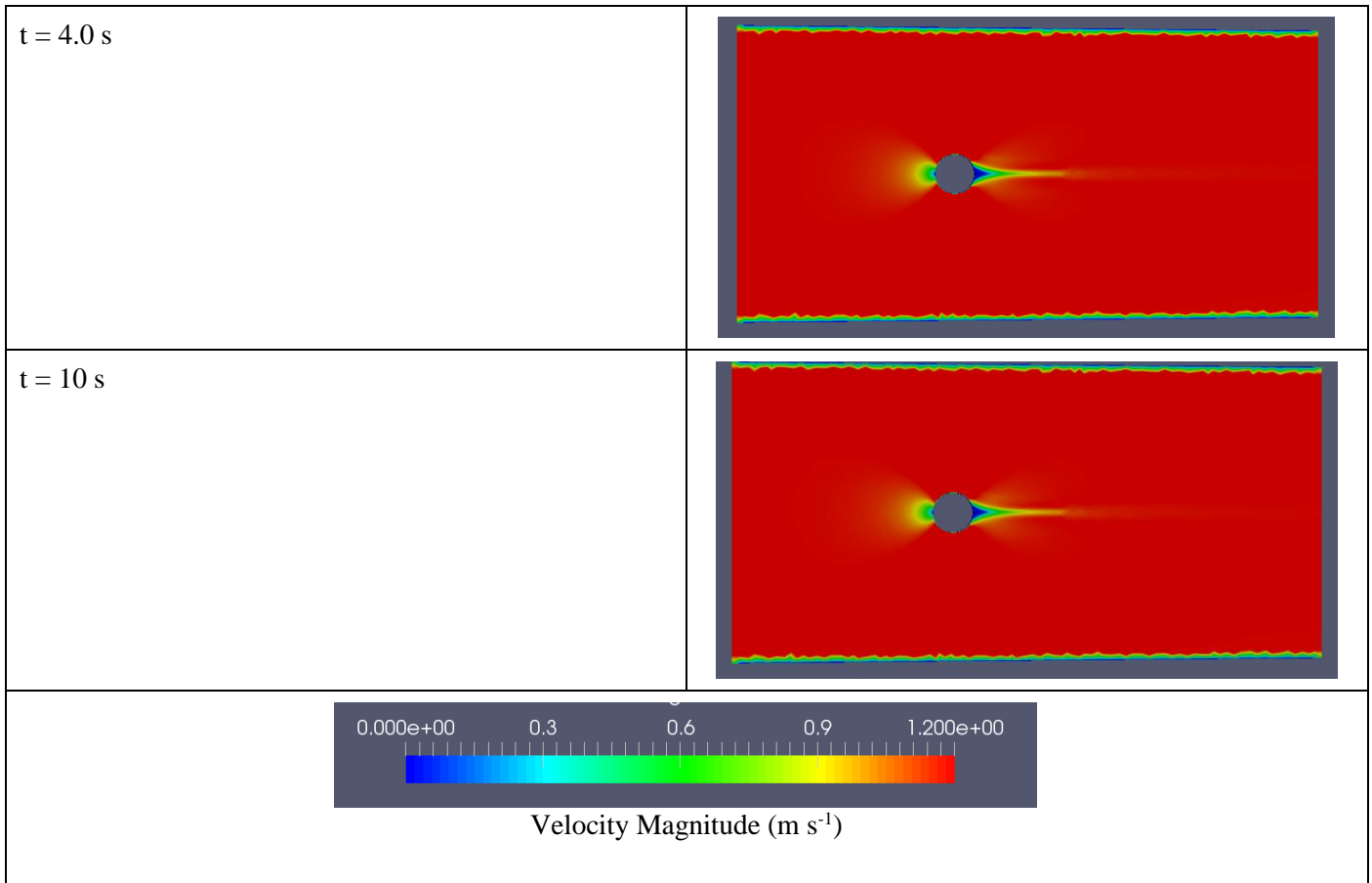
- At inlet velocity  $1.2 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

Time t (s)	Result
t = 0 s	
t = 0.5 s	
t = 2.0 s	



At  $1.2 \text{ m s}^{-1}$  the fluid converges without showcasing any vortex shedding pattern.

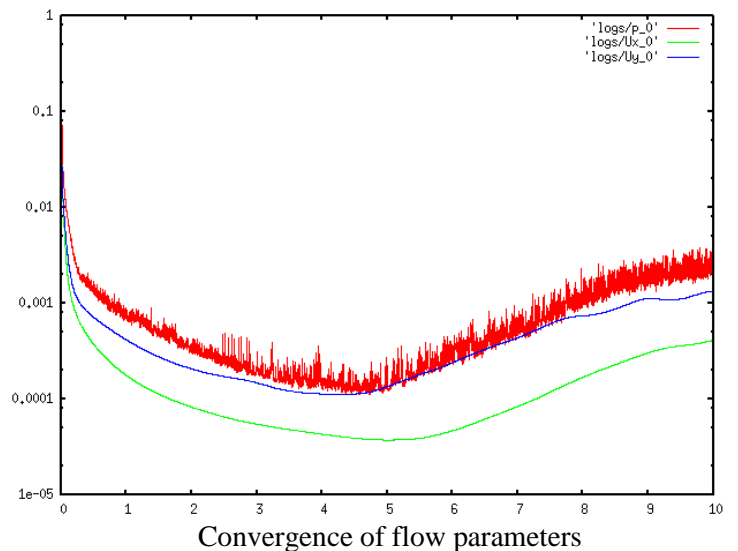
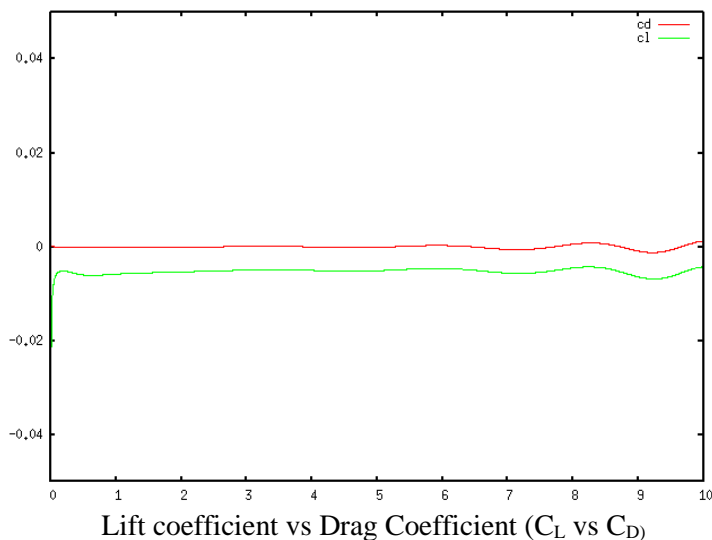
The working fluid in Case I does not possess enough viscosity to undergo a boundary layer separation or a pressure difference between the top and bottom of the body. Though it can be noticed that the fluid parameters converge at a faster rate with increase in inlet velocity.

## CASE II

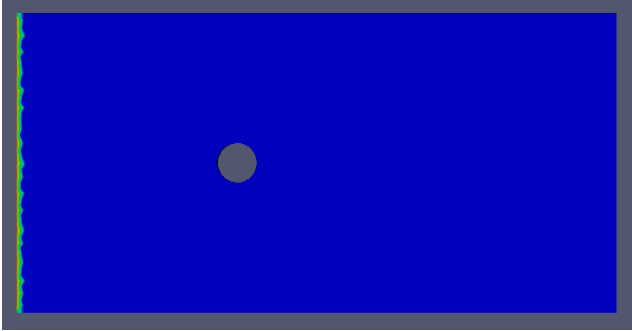
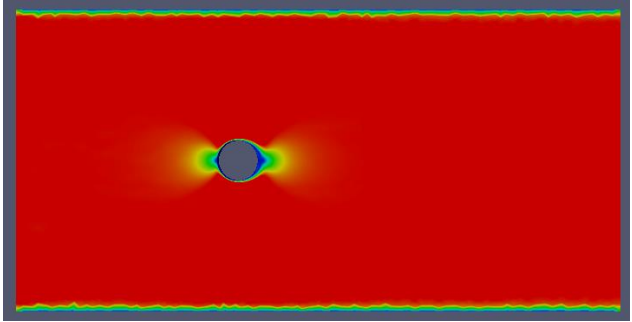
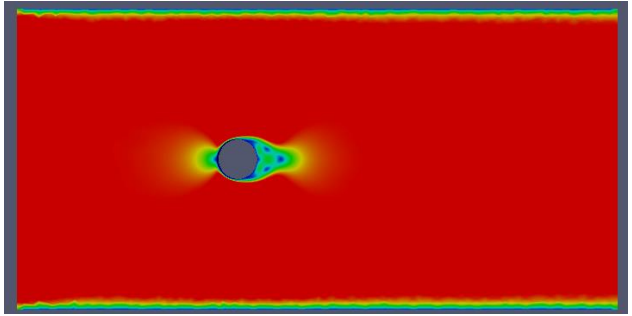
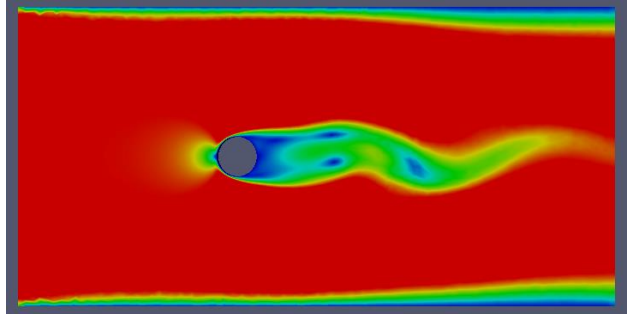
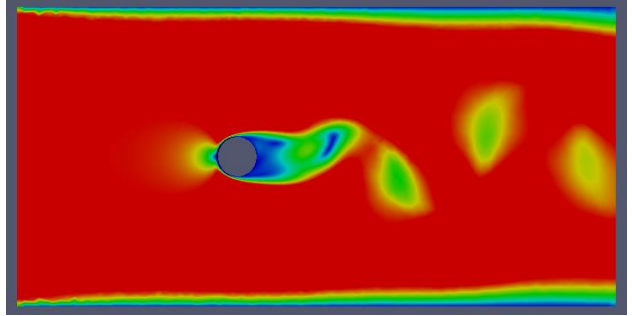
For the second case the kinematic viscosity is set at  $5.555 \times 10^{-4} \text{ m}^2 \text{ s}^{-1}$ . The fluid inlet velocity is set at  $0.5 \text{ m s}^{-1}$ ,  $1.0 \text{ m s}^{-1}$  and  $1.5 \text{ m s}^{-1}$ . The simulation results for each subcase is as follows:

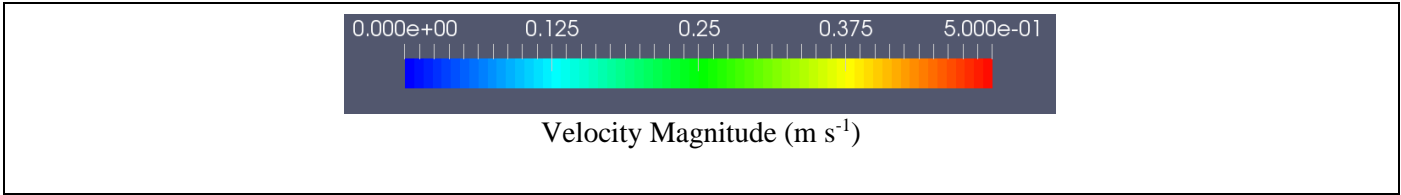
- At inlet velocity  $0.5 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

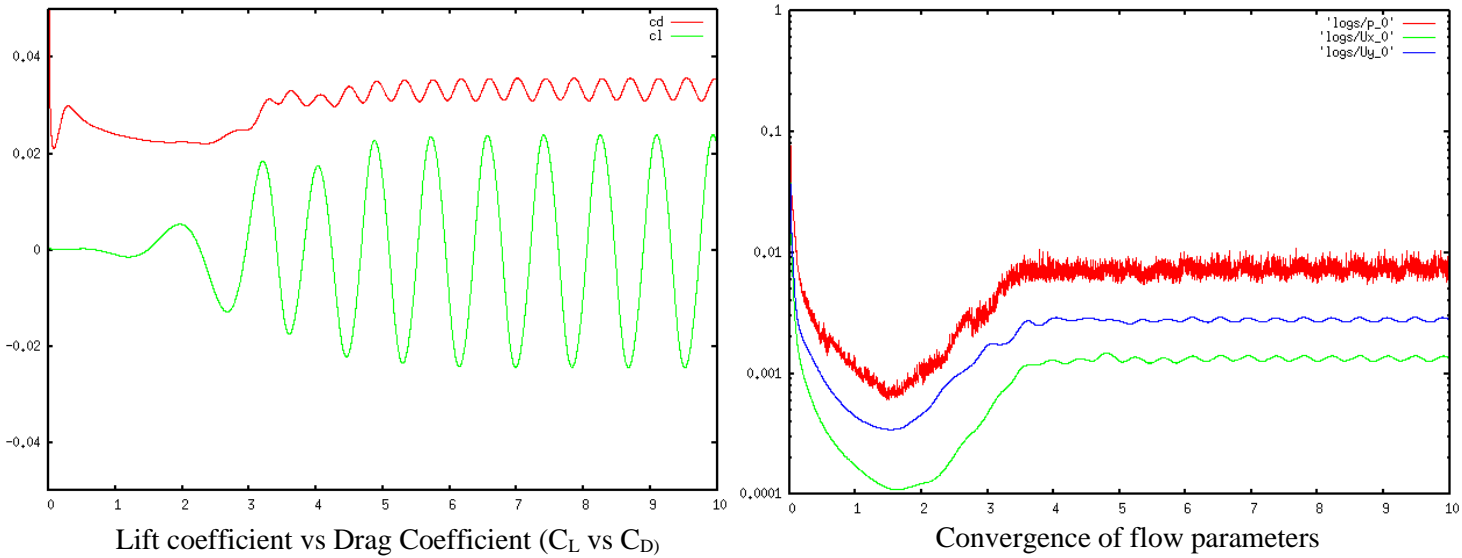
Time t (s)	Result
t = 0 s	 A rectangular domain with a blue background. A gray circle is centered in the domain. The left boundary is marked with a dashed green line.
t = 2.0 s	 The domain is mostly red. A gray circle is centered. A small, bright yellow-green oval is centered on the circle, indicating a localized velocity perturbation.
t = 6.0 s	 The domain is mostly red. A gray circle is centered. A larger, elongated yellow-green oval is centered on the circle, indicating a growing velocity perturbation.
t = 8.0 s	 The domain is mostly red. A gray circle is centered. A long, wavy yellow-green structure is centered on the circle, indicating a developing vortex or wake.
t = 10 s	 The domain is mostly red. A gray circle is centered. A long, wavy yellow-green structure is centered on the circle, with several smaller, bright yellow-green spots appearing downstream, indicating a more complex flow structure.



At  $0.5 \text{ m s}^{-1}$  the fluid converges with vortex shedding pattern occurring after 8 seconds into simulation.

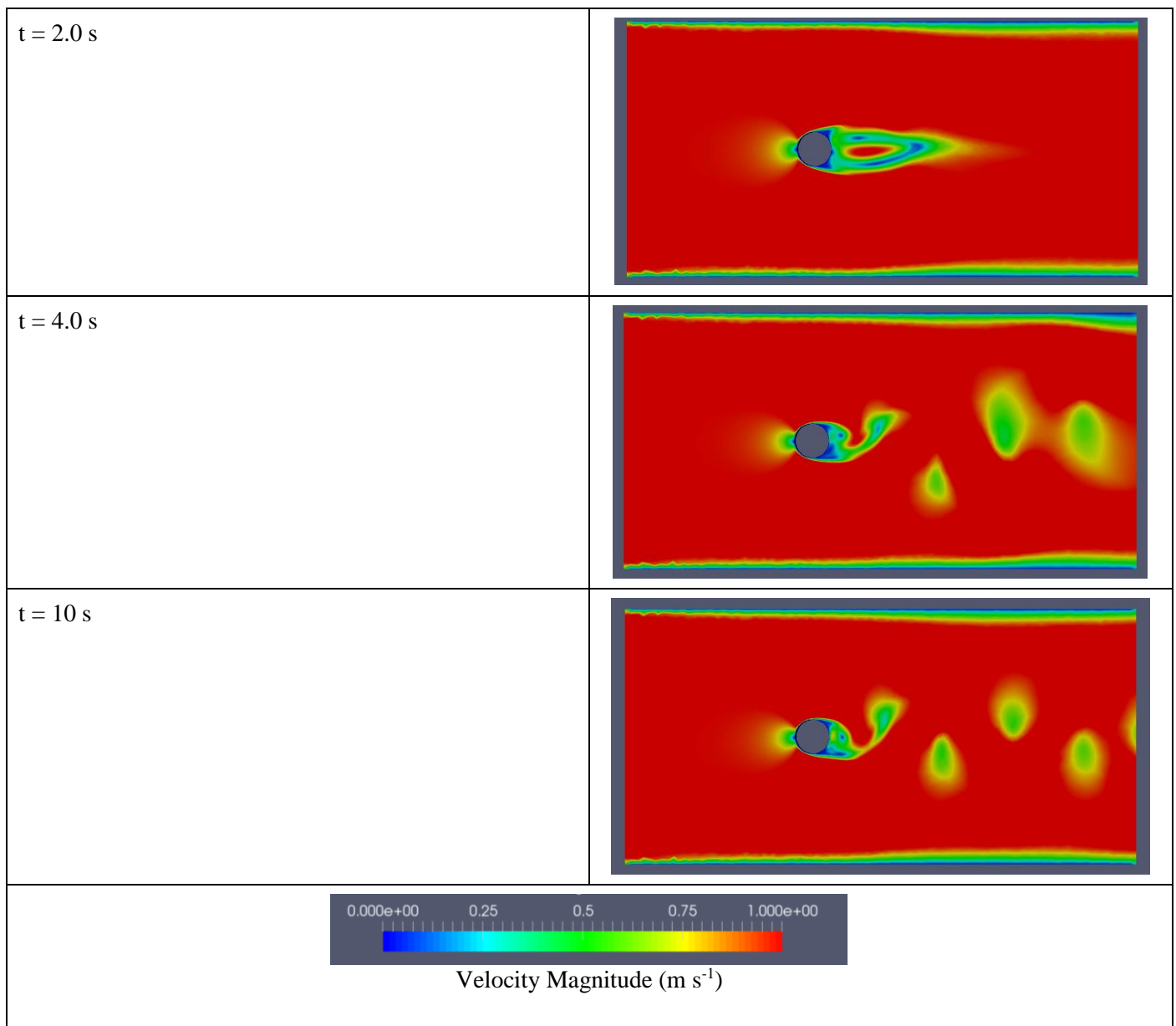
- At inlet velocity  $1.0 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

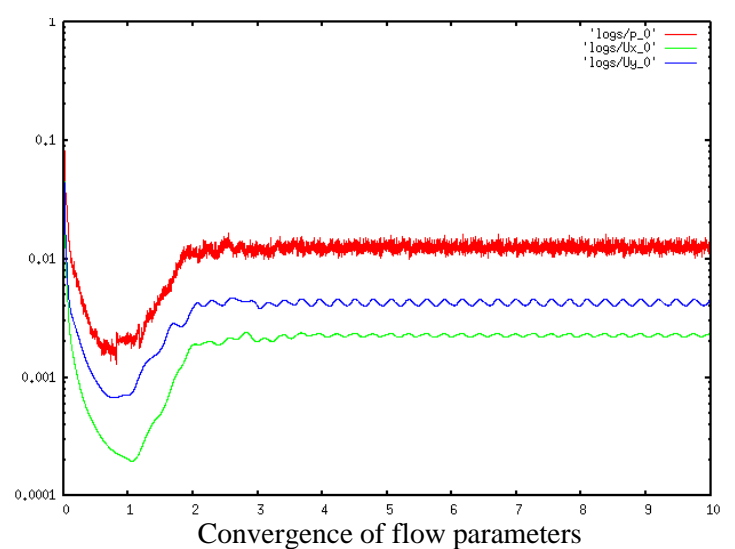
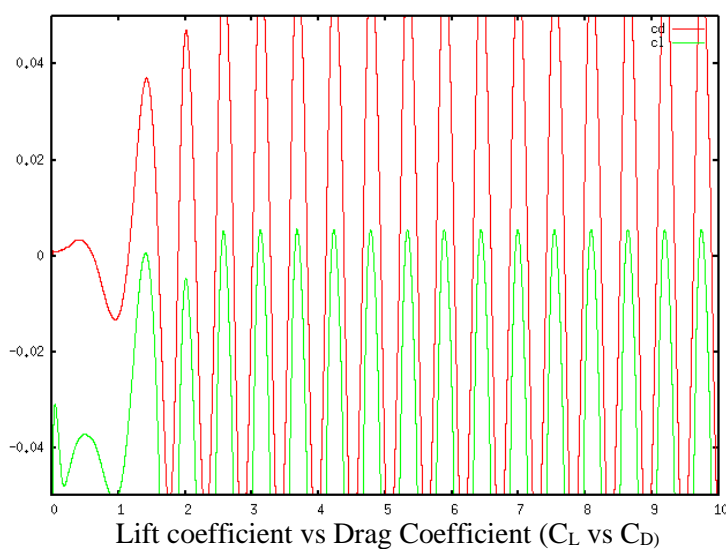
Time t (s)	Result
t = 0 s	
t = 0.5 s	



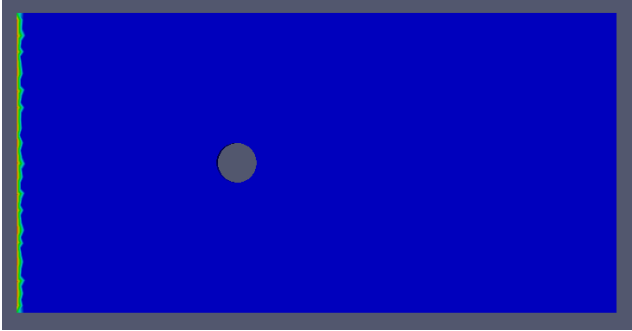
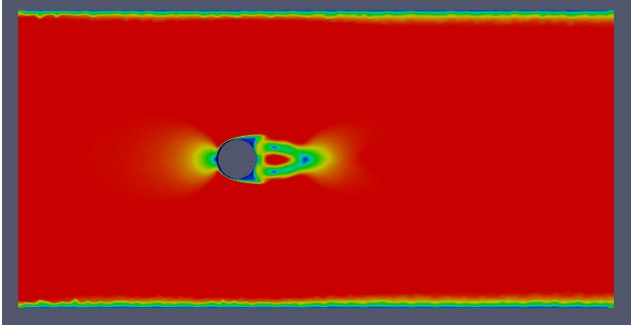
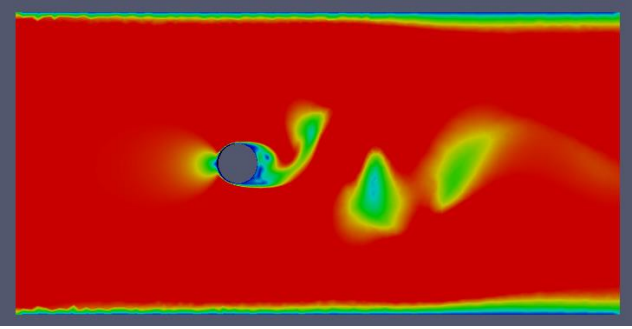
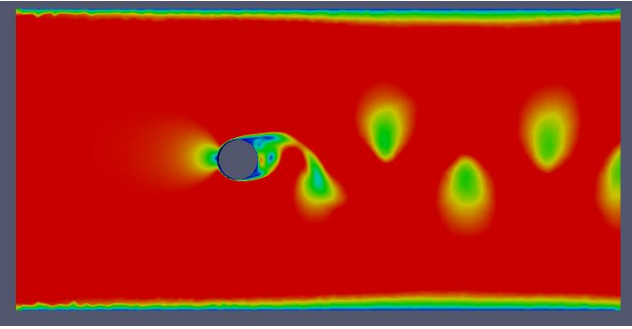
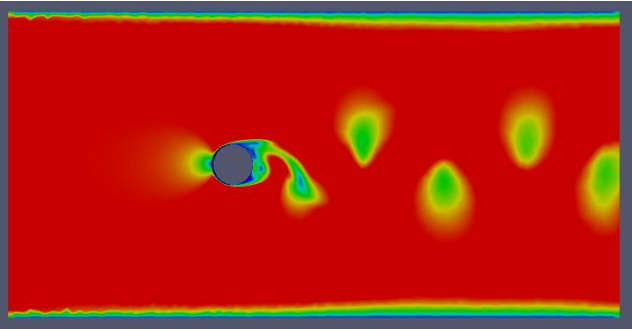
At  $1.0 \text{ m s}^{-1}$  the fluid converges with vortex shedding pattern occurring after 3 seconds into simulation.

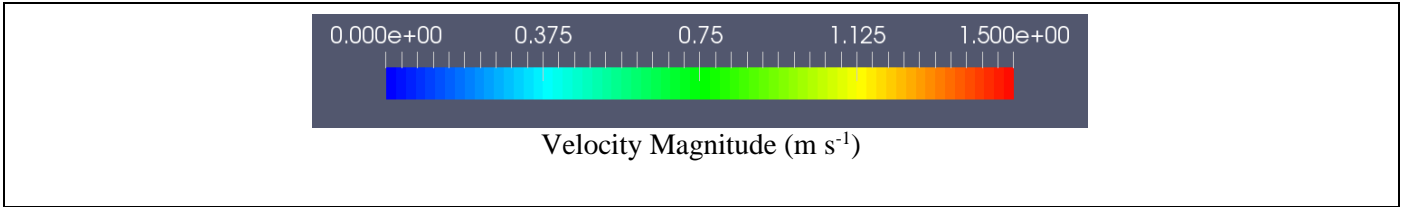
- At inlet velocity  $1.5 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

Time t (s)	Result
t = 0 s	 A rectangular domain with a blue background. A small gray circle is centered in the domain. The left and bottom boundaries are marked with a dashed green line.
t = 0.5 s	 The domain is mostly red. A small gray circle is centered. A small, elongated, yellow-green region is visible near the center, indicating a developing velocity profile.
t = 2.0 s	 The domain is mostly red. A small gray circle is centered. A larger, more complex yellow-green region is visible near the center, indicating a developing velocity profile.
t = 4.0 s	 The domain is mostly red. A small gray circle is centered. A larger, more complex yellow-green region is visible near the center, indicating a developing velocity profile.
t = 10 s	 The domain is mostly red. A small gray circle is centered. A larger, more complex yellow-green region is visible near the center, indicating a developing velocity profile.



At  $1.5 \text{ m s}^{-1}$  the fluid converges with vortex shedding pattern occurring after 2 seconds into simulation.

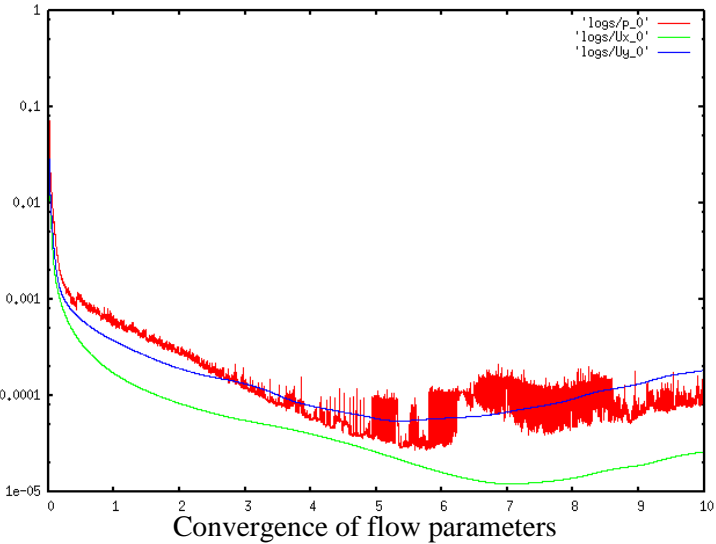
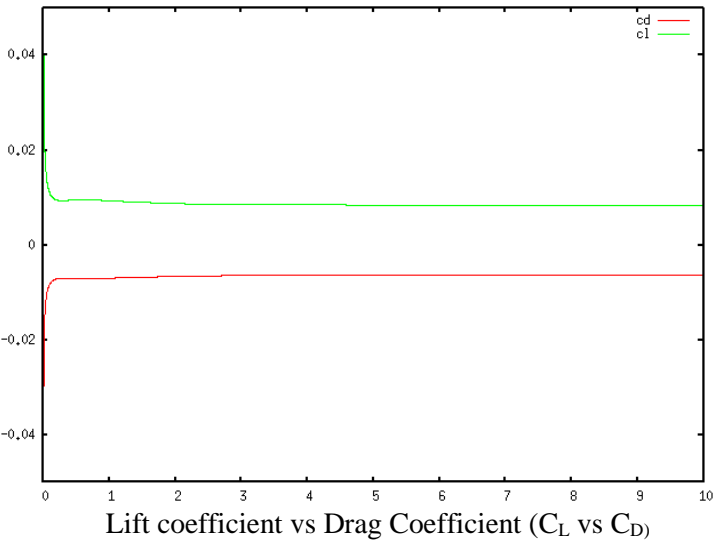
The working fluid in Case II possess enough viscosity to undergo a boundary layer separation and pressure difference between the top and bottom of the body thus, forming a vortex shedding pattern even at low inlet velocity. The fluid parameters converge at a faster rate when inlet velocity is increased.

### CASE III

For the third case the kinematic viscosity is set at  $1.18 \times 10^{-3} \text{ m}^2 \text{ s}^{-1}$ . The fluid inlet velocity is set at  $0.5 \text{ m s}^{-1}$ ,  $1.0 \text{ m s}^{-1}$ ,  $1.5 \text{ m s}^{-1}$  and  $2.0 \text{ m s}^{-1}$ . The simulation results for each subcase is as follows:

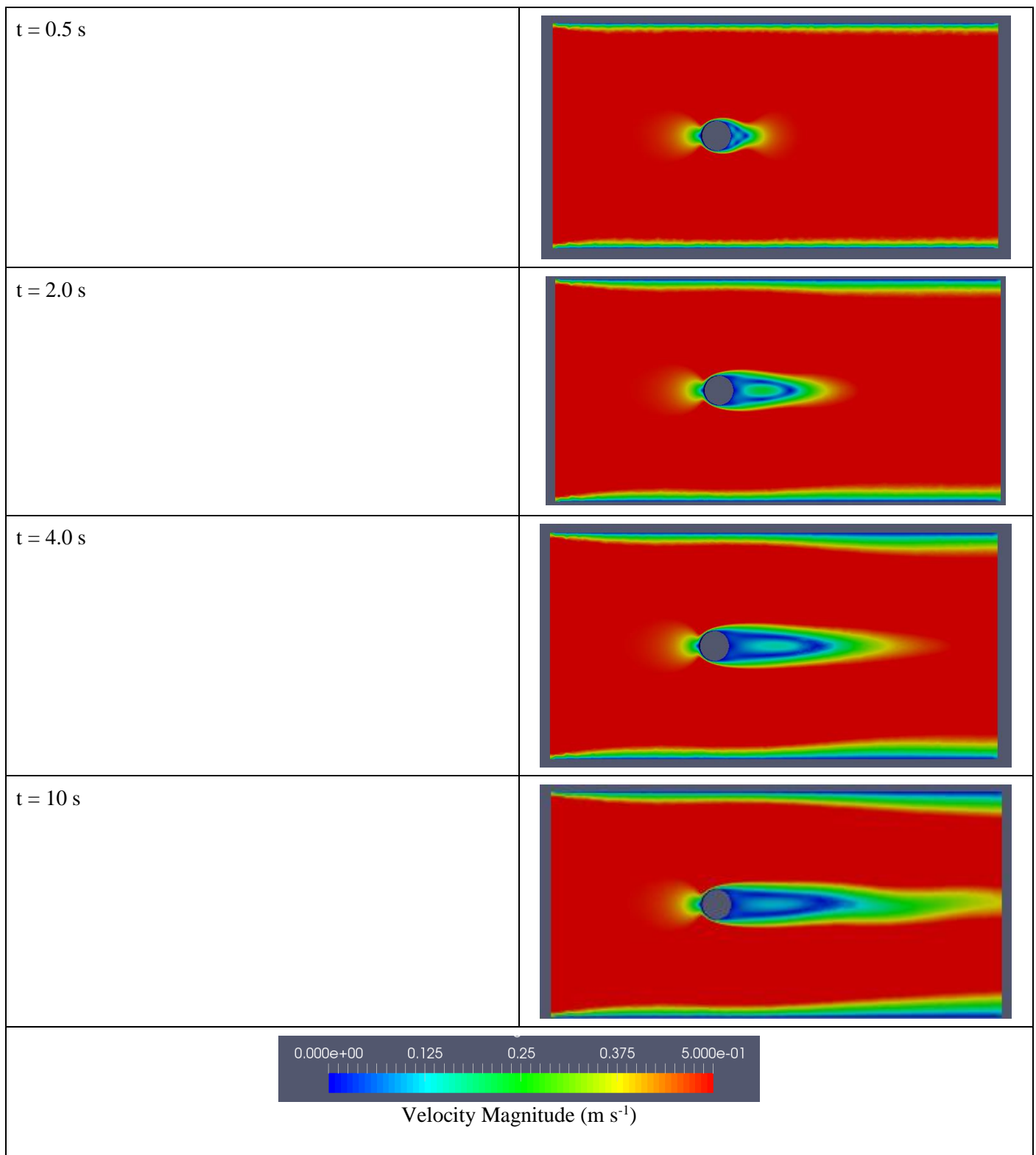
- At inlet velocity  $0.5 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

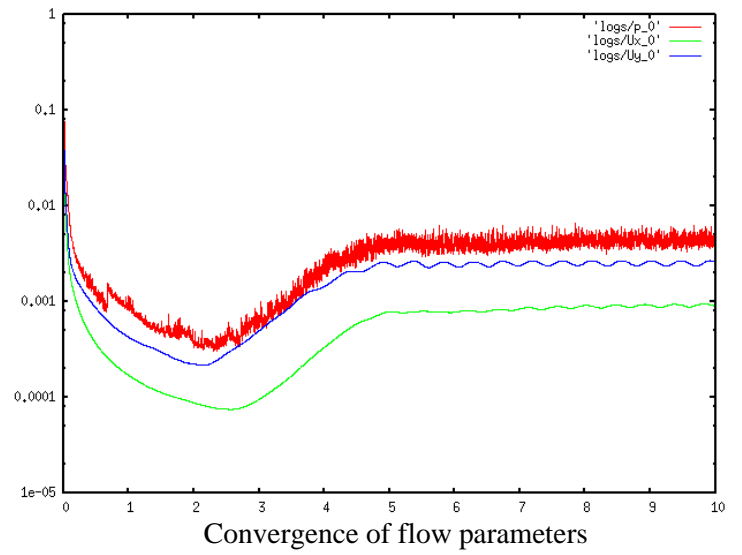
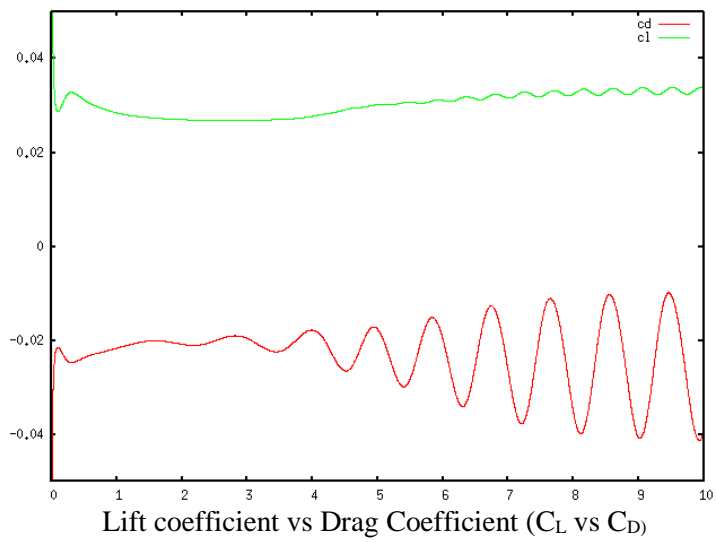
Time t (s)	Result
t = 0 s	



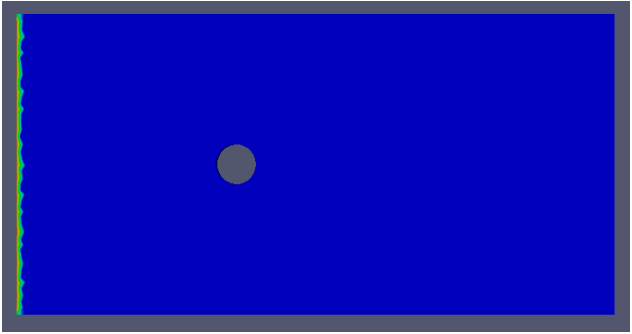
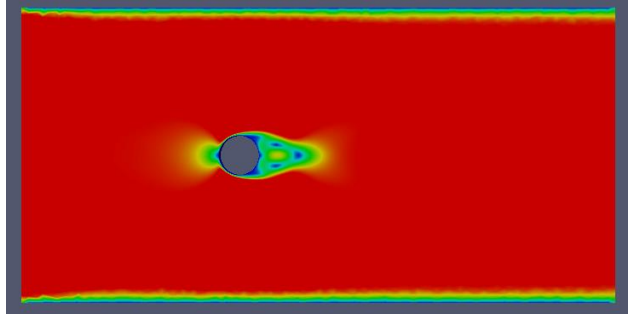
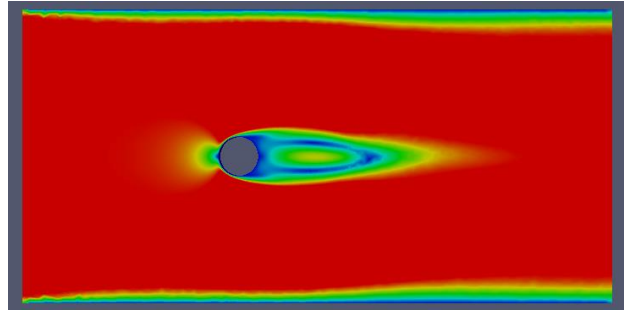
At  $0.5 \text{ m s}^{-1}$  the fluid parameters do not converge over the 10 second time period of the simulation, however it does show the tendency to converge in near future. The simulation ends without showcasing any vortex shedding pattern.

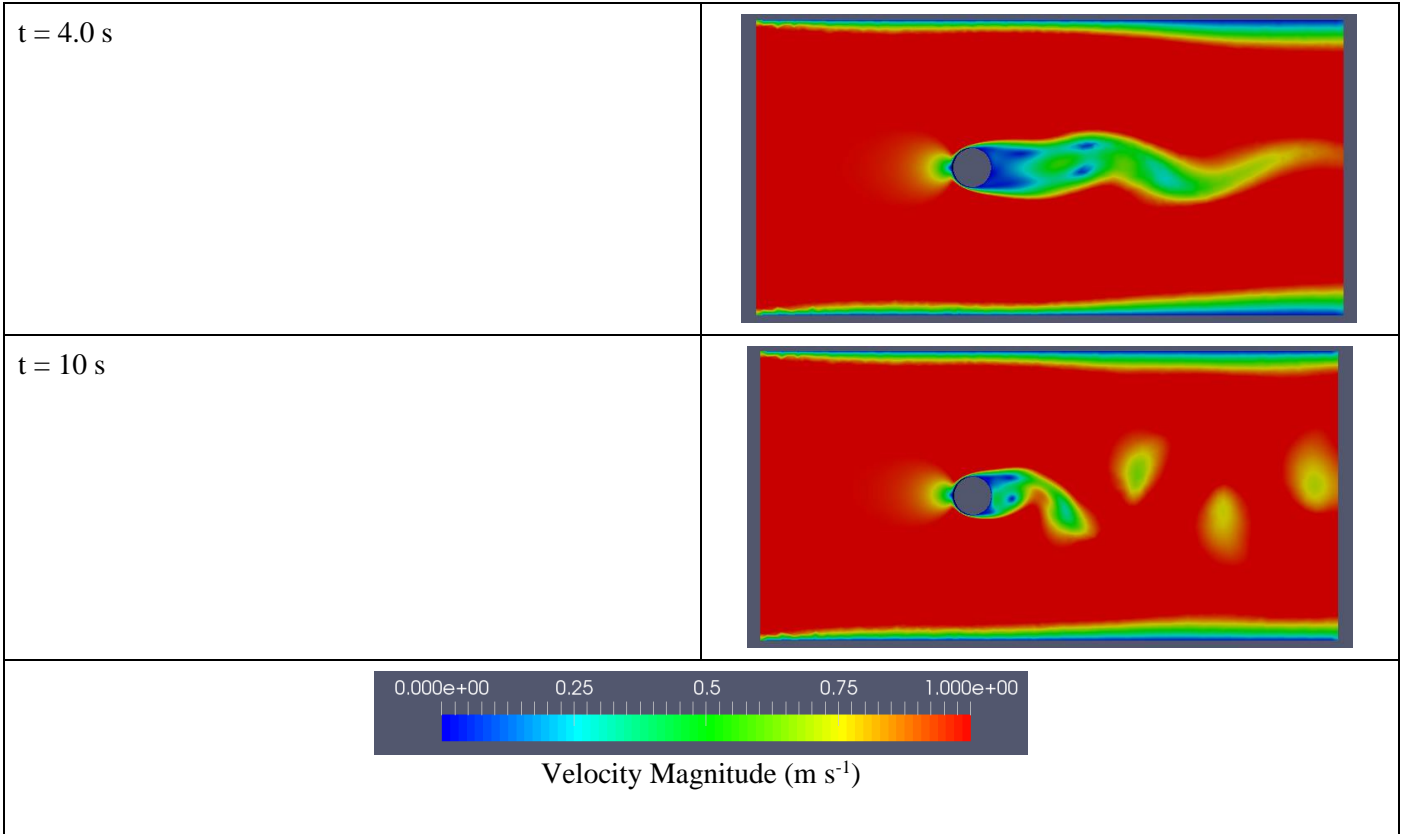
- At inlet velocity  $1.0 \text{ m s}^{-1}$

## Plots:



## Simulation results for velocity profiles at different time intervals:

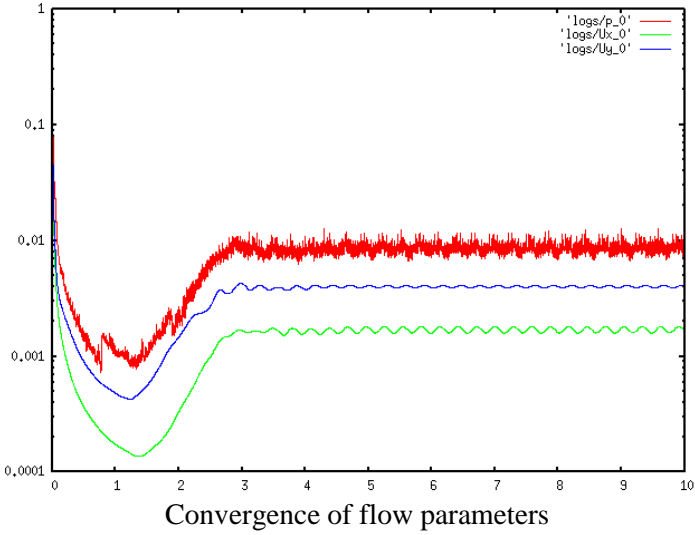
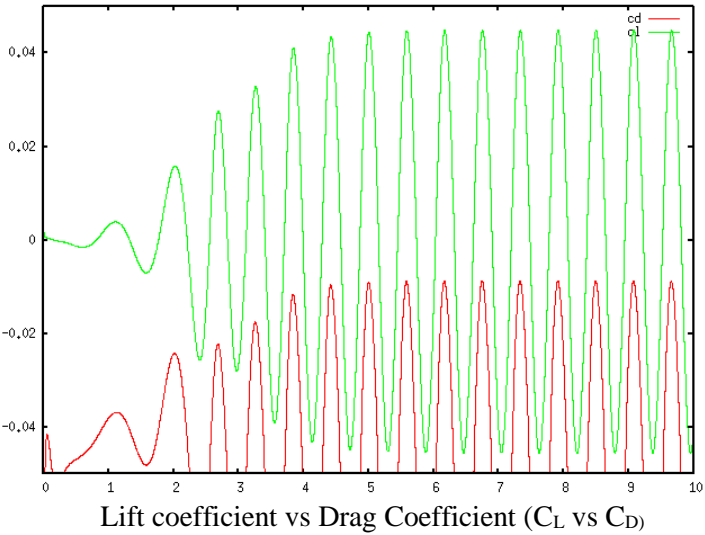
Time t (s)	Result
t = 0 s	
t = 0.5 s	
t = 2.0 s	



At  $1.0 \text{ m s}^{-1}$  the fluid parameters converge with showcasing vortex shedding patterns, occurring after 5 seconds into simulation.

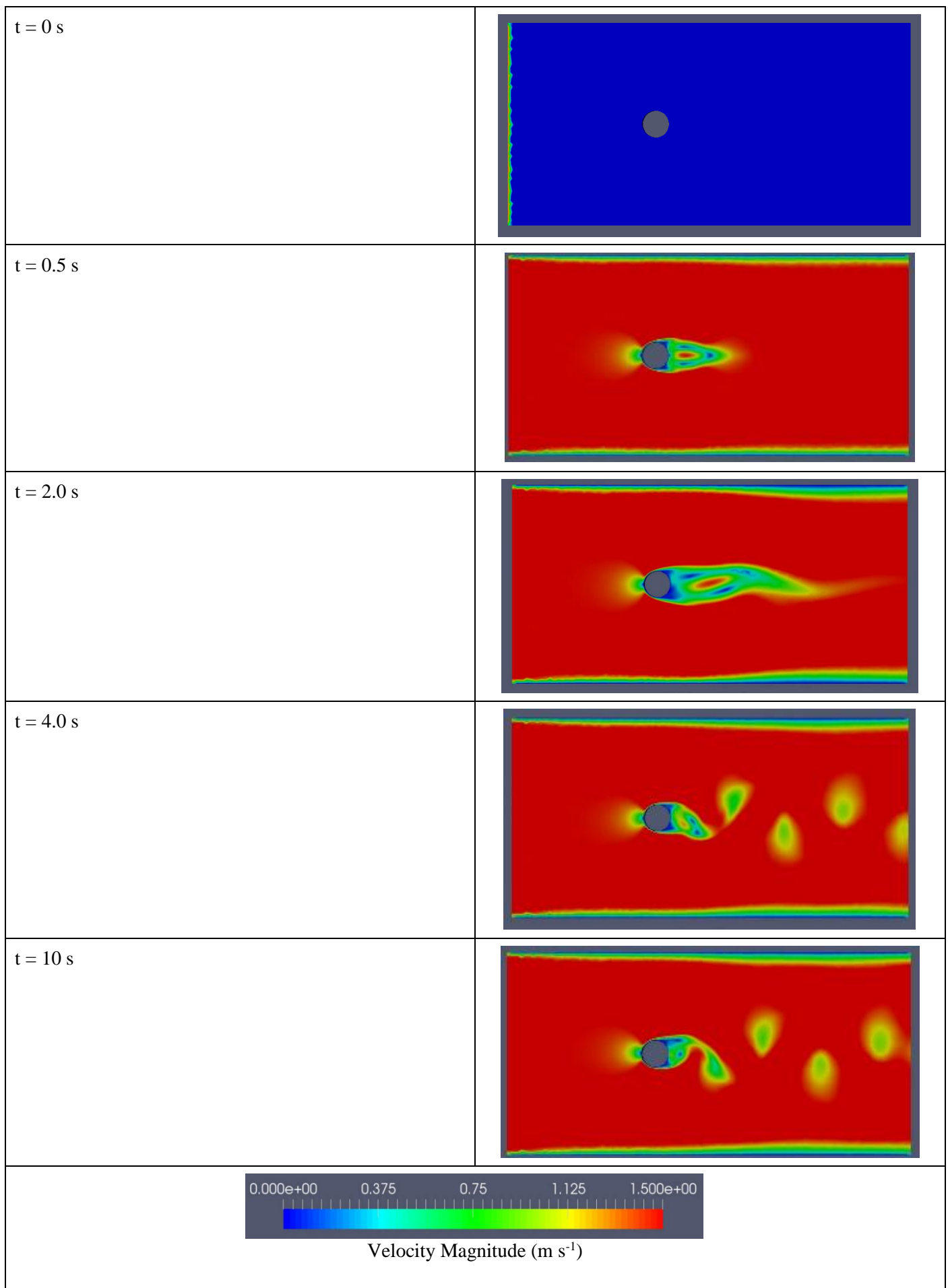
- At inlet velocity  $1.5 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

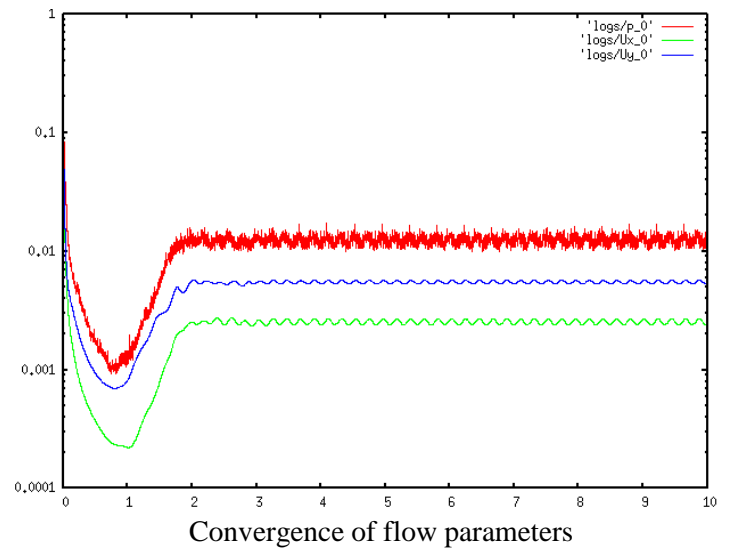
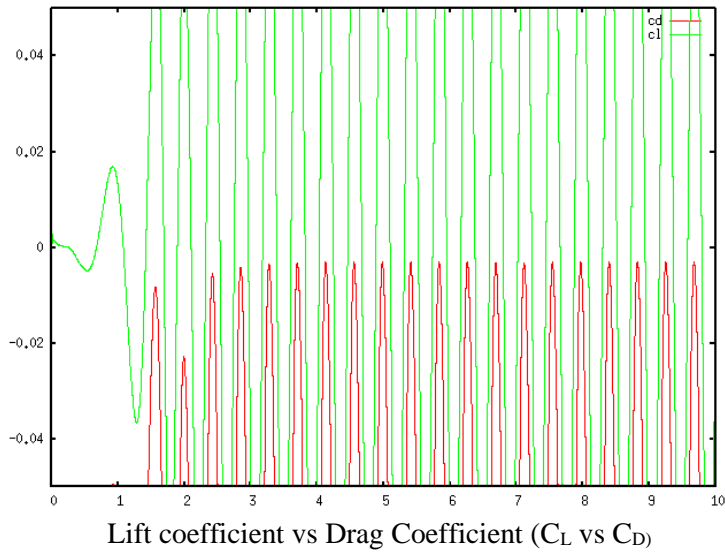
Time t (s)	Result
------------	--------



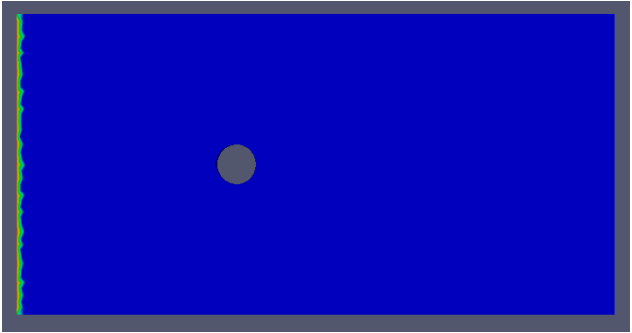
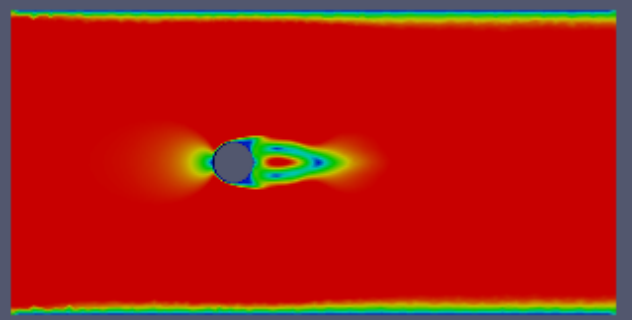
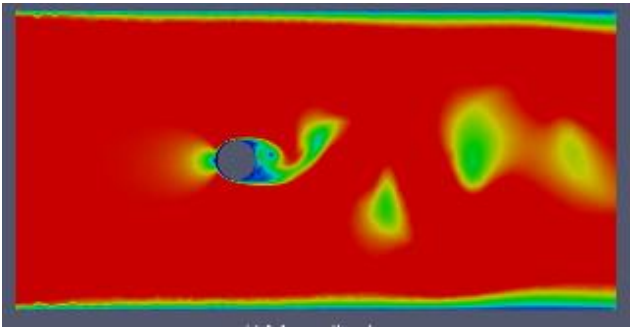
At  $1.5 \text{ m s}^{-1}$  the fluid parameters converge with showcasing vortex shedding patterns, occurring after 3 seconds into simulation.

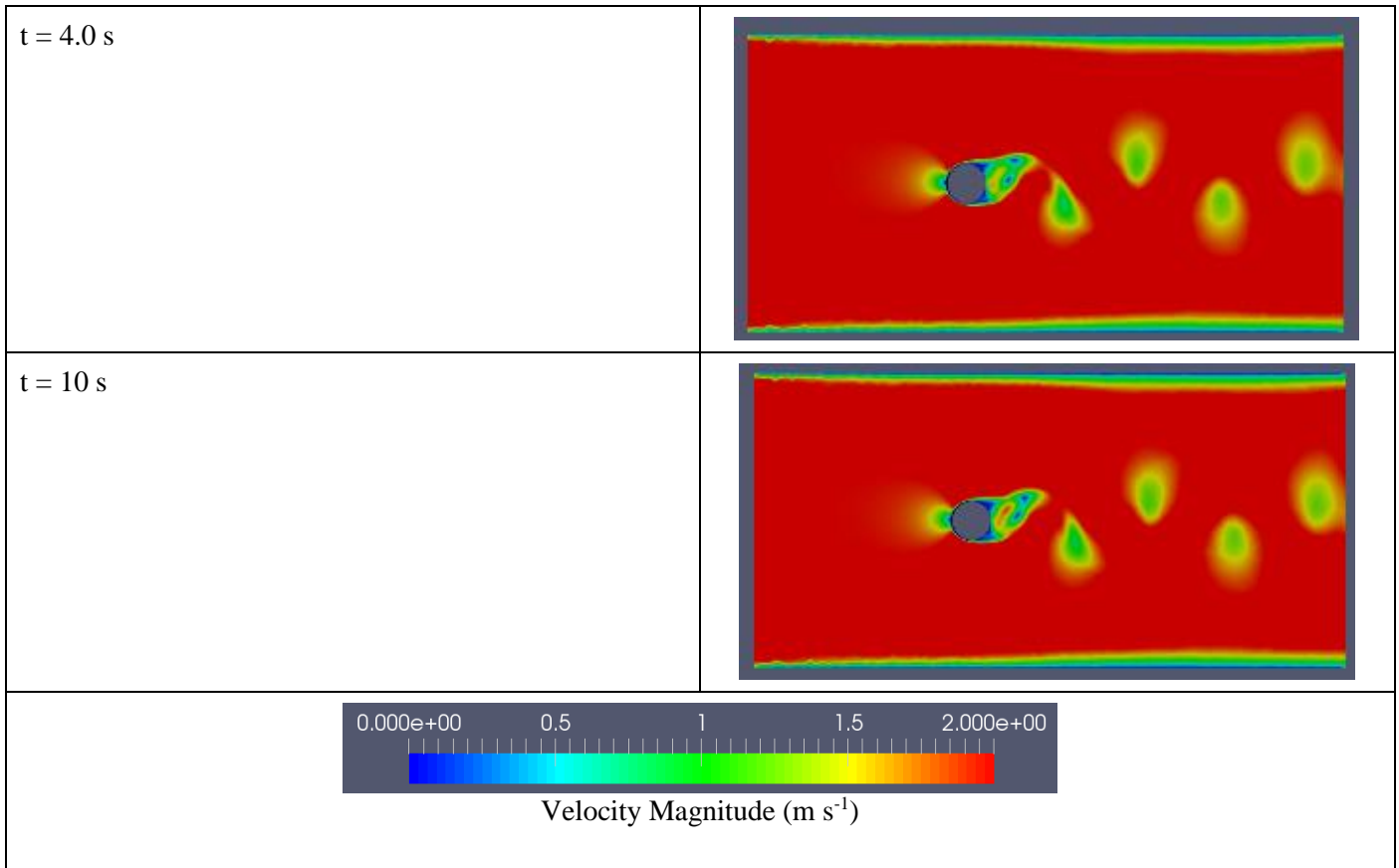
- At inlet velocity  $2.0 \text{ m s}^{-1}$

Plots:



Simulation results for velocity profiles at different time intervals:

Time t (s)	Result
t = 0 s	
t = 0.5 s	
t = 2.0 s	



At  $2.0 \text{ m s}^{-1}$  the fluid parameters converge with showcasing vortex shedding patterns, occurring after 2 seconds into simulation.

The working fluid in Case III also possess enough viscosity to undergo a boundary layer separation and undergo pressure difference between the top and bottom of the body thus, forming a vortex shedding pattern, even at low inlet velocities. The fluid parameters converge at a faster rate when inlet velocity is increased.

## 6. Conclusion

It is evident from the above results, that the formation of Kármán vortex street phenomenon is dependent on the fluid viscosity as well as the flow velocity. It can be observed that the possibility of vortex street formation is more likely in a fluid with higher kinematic viscosity, which in turn can be expressed as the ratio of dynamic viscosity and density. With increase in fluid/inlet/freestream velocity, the rate of convergence of fluid parameters and frequency of the vortex street (swirling vortex patterns) increases, such effects are dormant in a fluid with low kinematic viscosity. It can also be observed that the initiation of vortex street can be delayed by decreasing inlet velocity and vice versa.

While comparing two distinct working fluid (for instance take Case II and III) it can be observed that at same inlet velocity, the fluid whose kinematic viscosity is higher, takes more time to generate vortex shedding pattern. It is due to the fact that a higher kinematic viscosity results in more friction at the contact surface of blunt body and fluid, thus taking longer for flow separation.

## Reference

1. Masanori Hashiguchi & Kunio Kuwahara; Two-Dimensional Study of Flow past a Circular Cylinder  
[<http://hdl.handle.net/2433/60759>]
2. <https://www.openfoam.com/documentation/>
3. <https://spoken-tutorial.org/>
4. <https://www.ksb.com/centrifugal-pump-lexicon/transient-flow/328110/>
5. [https://en.wikipedia.org/wiki/Kármán\\_vortex\\_street](https://en.wikipedia.org/wiki/Kármán_vortex_street)
6. [https://en.wikipedia.org/wiki/PISO\\_algorithm](https://en.wikipedia.org/wiki/PISO_algorithm)
7. [https://en.wikipedia.org/wiki/Reynolds-averaged\\_Navier–Stokes\\_equations](https://en.wikipedia.org/wiki/Reynolds-averaged_Navier–Stokes_equations)

## System and Software

This study was conducted on the system whose hardware specification is listed as follows:

Device	Acer Nitro 5
Processor	Intel i5-7300HQ
GPU	HD Graphics 630 GTX 1050
RAM	8 GB
Drive Type	HDD
OS	CAELINUX (Case Study) Windows 10 (Documentation)

Software used for Case Study:

- Geometry and Mesh- Gmsh v3.0.6
- Simulation- OpenFOAM v4.1
- Post-processing- Paraview v5.0.1 & Gnuplot v4.6