Complex Engineering Problem

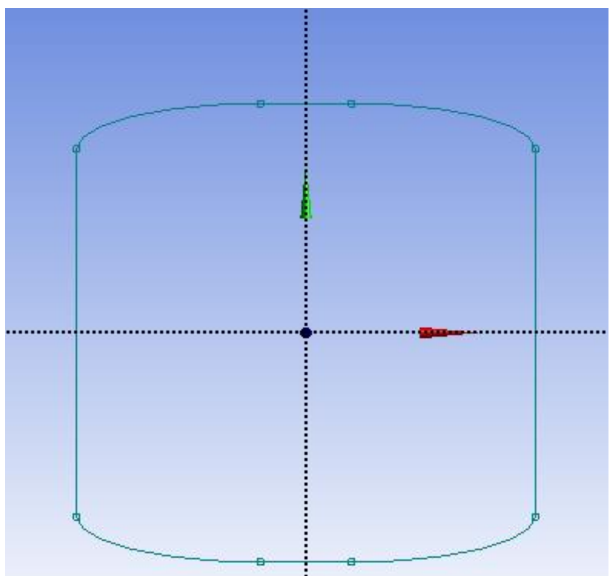
Submitted By: Ahsan Farooq
Roll NO: FA20-BME-005

SIMULATION AND ANALYSIS OF FLOW PASSING AN ELLIPTICAL BLUNT BODY ON ANCYS

CONTENTS

1-Introduction:

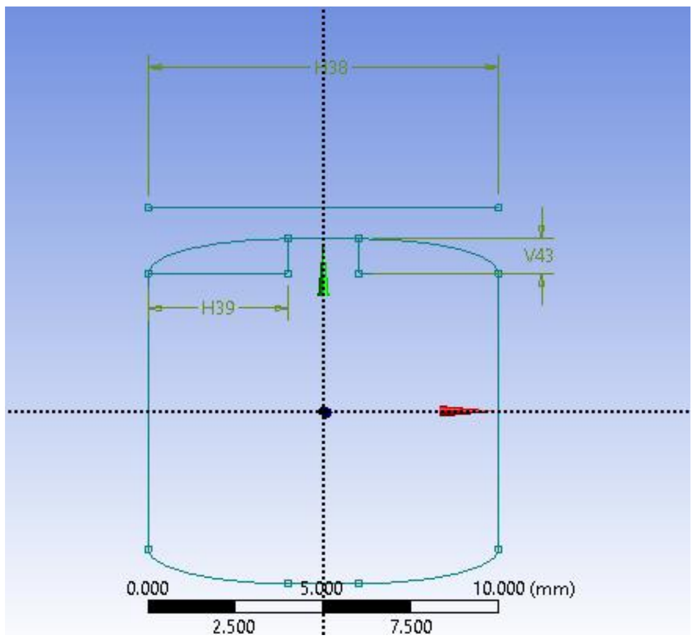
This report focuses on the analysis of flow patterns around elliptical blunt body having Reynolds Numbers (Re) of 150, using the CFD software ANSYS Fluent. By using this programme, we will examine different parameters for flow around a blunt body which includes velocity and pressure distributions along with lift and drag coefficients for body and the effects of different flow parameters on the characteristics of the flow field. We also provide insights into the characteristics of the flow field, including the regions of high and low pressure, the wake structure, and the vortical patterns that form.



2-Geometry:

We have created 2D (10 x 10 mm) square with four equal elliptical corners. It has two vertical sides of same length and two horizontal sides of same length.

The 2D geometry of the body was developed in the design modeler in the Ancys workbench. A square model with an area of 100 mm^2 was converted to an ellipsoid using an elliptical arc with a major axis of 4 mm and a minor axis of 1 mm. Also flow field is considered around the body by drawing the circle of radius 100mm.



Details View

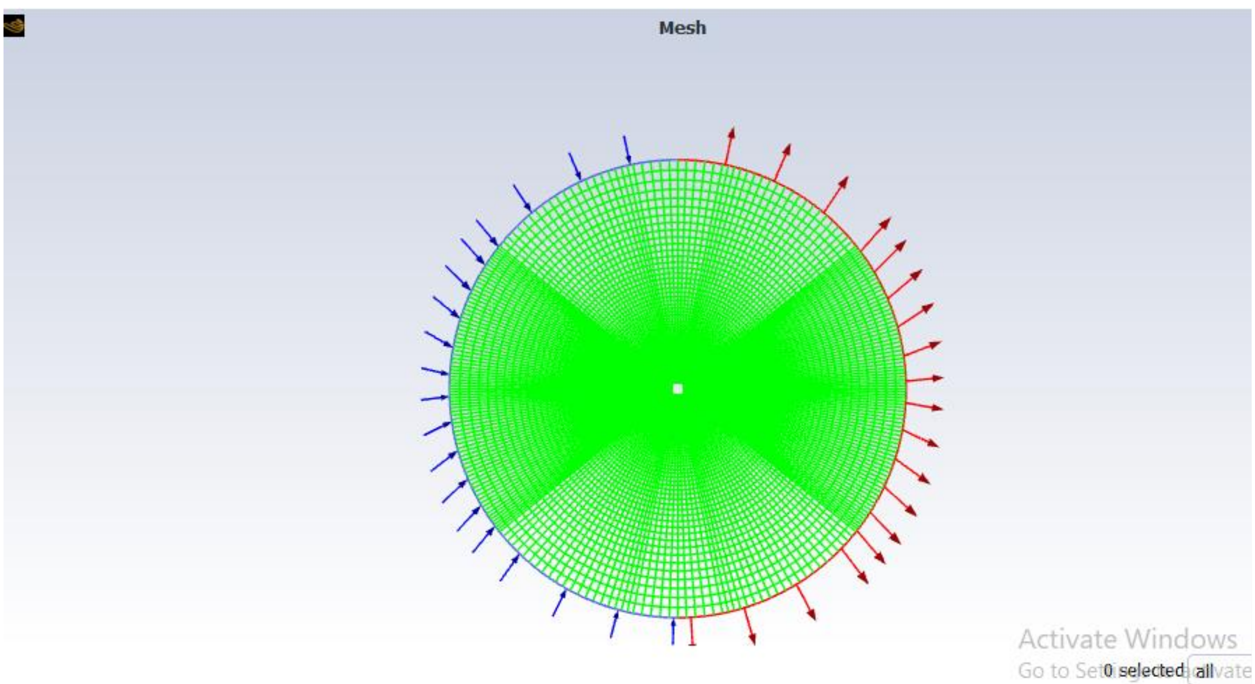
Details of Sketch2	
Sketch	Sketch2
Sketch Visibility	Show Sketch
Show Constraints?	No
Dimensions: 3	
H38	10 mm
H39	4 mm
V43	1 mm

3-Meshing:

The meshing process in ANSYS involves dividing the geometry into small, finite-sized elements. The meshing process involves several steps, including geometry preparation, mesh sizing, and mesh generation.

First step of geometry preparation is detailed in above step of “Geometry’’. The next step in the meshing process is to determine the size and distribution of the mesh elements. Face sizing is applied to different faces of geometry model i.e vertical and horizontal lines and elliptical corners. Every mm is divided into 5 divisions for face sizing of body. Small to large type biasing is selected so that meshing near the surface of the body is more dense as compare to other areas.

Meshing is shown in figure below:



The left side of the geometry is considered as flow inlet and right side of the geometry is considered as flow outlet.

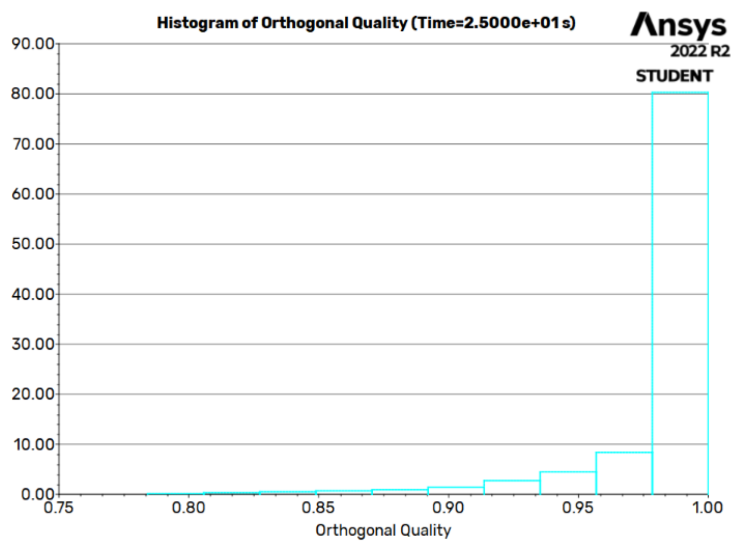
MESH SIZE

Cells	Faces	Nodes
26400	53064	26664

MESH DESIGN

Name	Type	Min Orthogonal Quality	Max Aspect Ratio
fluid-surface_body	Quad Cell	0.78418112	3.6911874

Orthogonal Quality



4-SETUP AND SOLUTION:

Upon opening ANSYS Fluent, certain parameters need to be determined based on the type of analysis being performed.

>Setup

>General>Time>Transient

>Models>Energy>Check Energy Equation

>Models>Viscous Model>Check Laminar

>Materials>Fluid>air

>Materials>Solid>aluminum

>Boundary Conditions

Inlet	
inlet	
Velocity Specification Method	Magnitude and Direction
Reference Frame	Absolute
Velocity Magnitude [m/s]	0.22
Supersonic/Initial Gauge Pressure [Pa]	0
Component of Flow Direction (x,y)	(1, 0)
Temperature [K]	298
Outlet	
outlet	
Backflow Reference Frame	Absolute
Gauge Pressure [Pa]	0
Pressure Profile Multiplier	1
Backflow Total Temperature [K]	298
Backflow Direction Specification Method	Direction Vector
Component of Flow Direction (x,y)	(1, 0)
Backflow Pressure Specification	Total Pressure

Build artificial walls to prevent reverse flow?	no
Average Pressure Specification?	no
Specify targeted mass flow rate	no
Wall	
cylinder	
Wall Thickness [m]	0
Heat Generation Rate [W/m^3]	0
Material Name	aluminum
Thermal BC Type	Temperature
Temperature [K]	300
Wall Motion	Stationary Wall
Shear Boundary Condition	No Slip
Convective Augmentation Factor	1

>Reference Values

Area	0.01 m^2
Density	1.225 kg/m^3
Depth	1 m
Enthalpy	0 J/kg
Length	0.01 m
Pressure	0 Pa

Temperature	298 K
Velocity	0.22 m/s
Viscosity	1.7894e-05 kg/(m s)
Ratio of Specific Heats	1.4
Yplus for Heat Tran. Coef.	300
Reference Zone	fluid-surface body

>Solutions

>Methods

>Solution Methods>Pressure-Velocity Coupling>Scheme>Fractional Step

>Transient Formulation>Second Order Implicit

>Check Non-Iterative Time Advancement

>Controls> Non-Iterative Solver Relaxation Factors

>Pressure=0.9

>Momentum=0.9

>Energy=0.9

>Report Definitions>New

>Surface Report>Area-Weighted Average>Field Variable>Wall Fluxes>Surface Nusselt Number>Surface>Select cylinder

>Force Report>Drag> Select cylinder

>Force Report>Lift> Select cylinder

>Initializing

>Initializing Methods>Check Standard Initializing

>Compute from>inlet

>Initialize

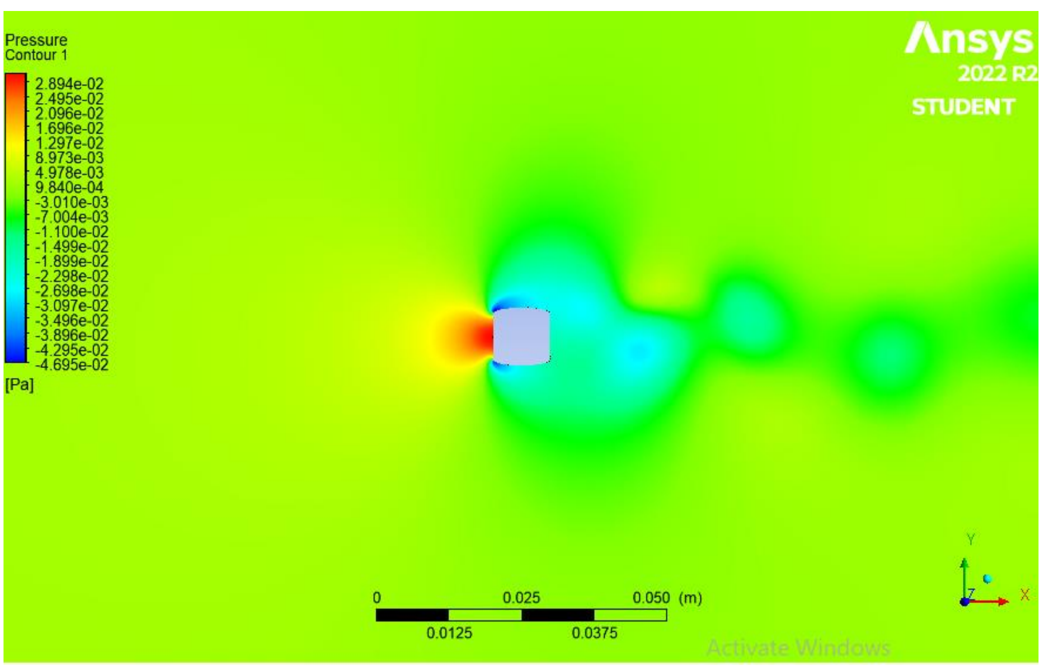
>Run Calculation

Unsteady Calculation Parameters	
Number of Time Steps	100000
Time Step Size [s]	0.0005

5-RESULTS AND ANALYSIS:

For results, we have plotted pressure and velocity contours as shown below:

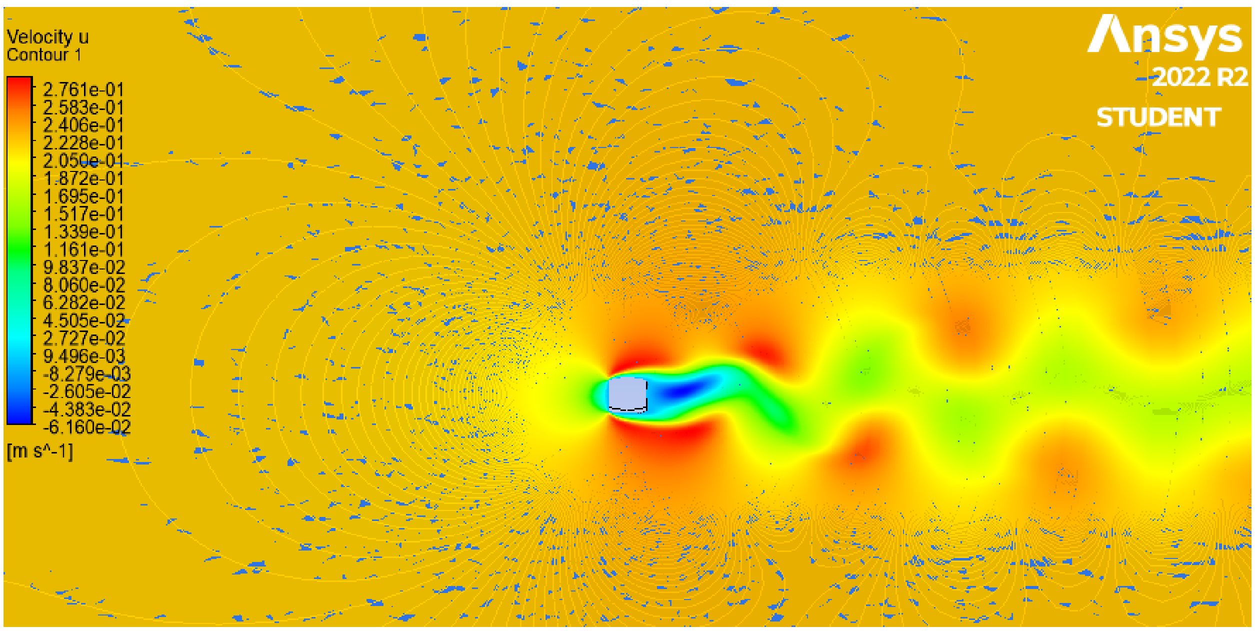
At left side, red color shows the high pressure which is reduced due to the Bernoulli's principle and the effects of flow separation. The Bernoulli's principle states that as the velocity of a fluid increases, the pressure decreases. As the fluid flows around an elliptical blunt body, the flow accelerates in certain regions, particularly around the leading edges and upper surface of the body. This acceleration results in a decrease in pressure in those regions, which is reflected in the pressure contours. Additionally, as the flow passes over the blunt trailing edge of the body, it separates from the body and creates a region of low-pressure wake behind it. This wake can further decrease the pressure.



Following figure shows velocity contour at Reynold number of 150.We can see in the figure below that on the very left side of our surface body stagnation point exists where maximum kinetic energy of the fluid has converted into pressure energy and velocity became zero and after that decreasing pressure gradient provides favourible conditions to the flow and velocity enhances.

Due to formation of the boundary layer, just along the surface of the body the velocity is small as compared to the incoming velocity.

Flow separation causes the formation of fluctuating eddies on one side of the surface body which results in vibrations in the body.These fluctuating eddies produces **VON KARMAN effect.**



LIFT AND DRAG PLOTS:

We have explained in the pressure contours that flow separation and Bernoulli effect causes pressure variations. These pressure variations causes pressure drag.

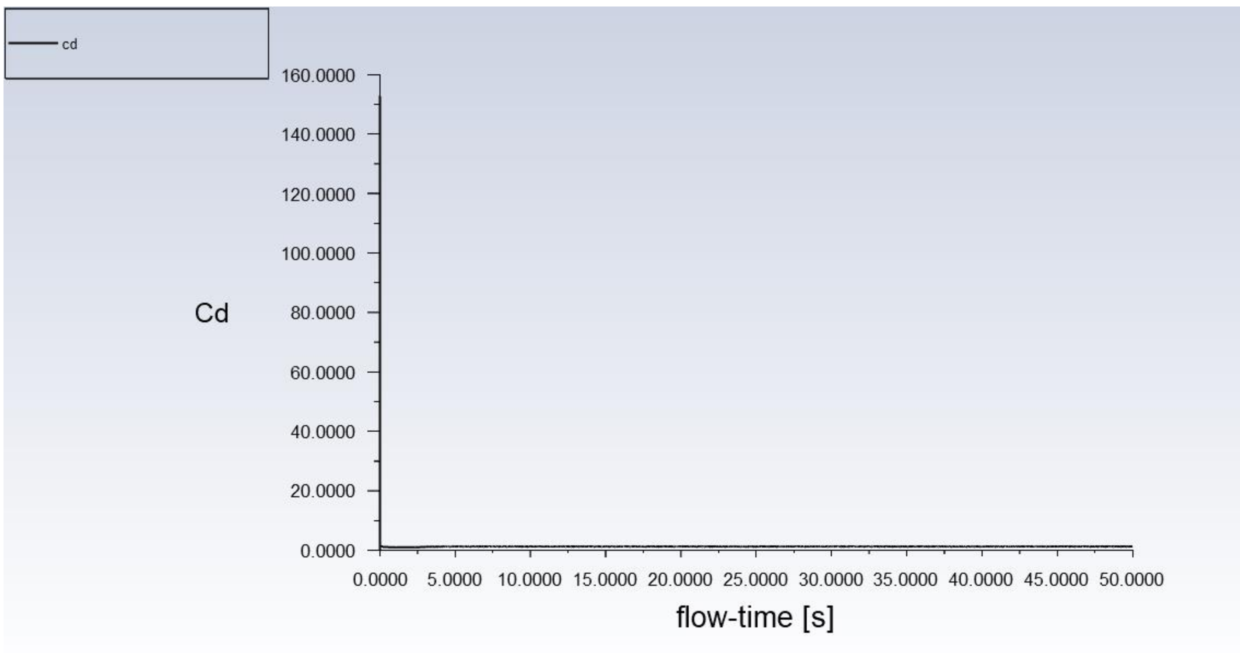
nu-no	5.616662
cd	1.240352
cl	0.2368925

Drag force can be calculated using formula

$$F(d) = Cd \times \frac{1}{2}\rho U^2 bl$$

$$F(d) = [1.240352 \times 0.5 \times 1.22 \times 0.22^2 \ \times 0.01 \times 0.01]$$

$\mathrm{F(d)} = 3.66 \times 10^{-6}$

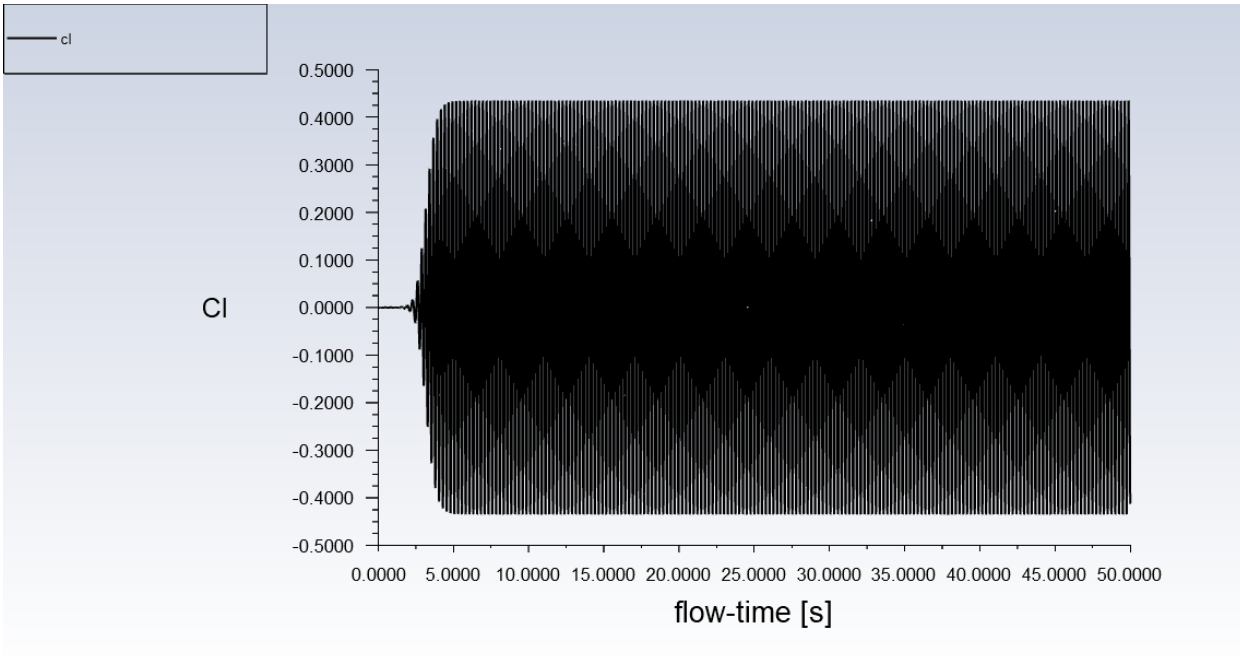


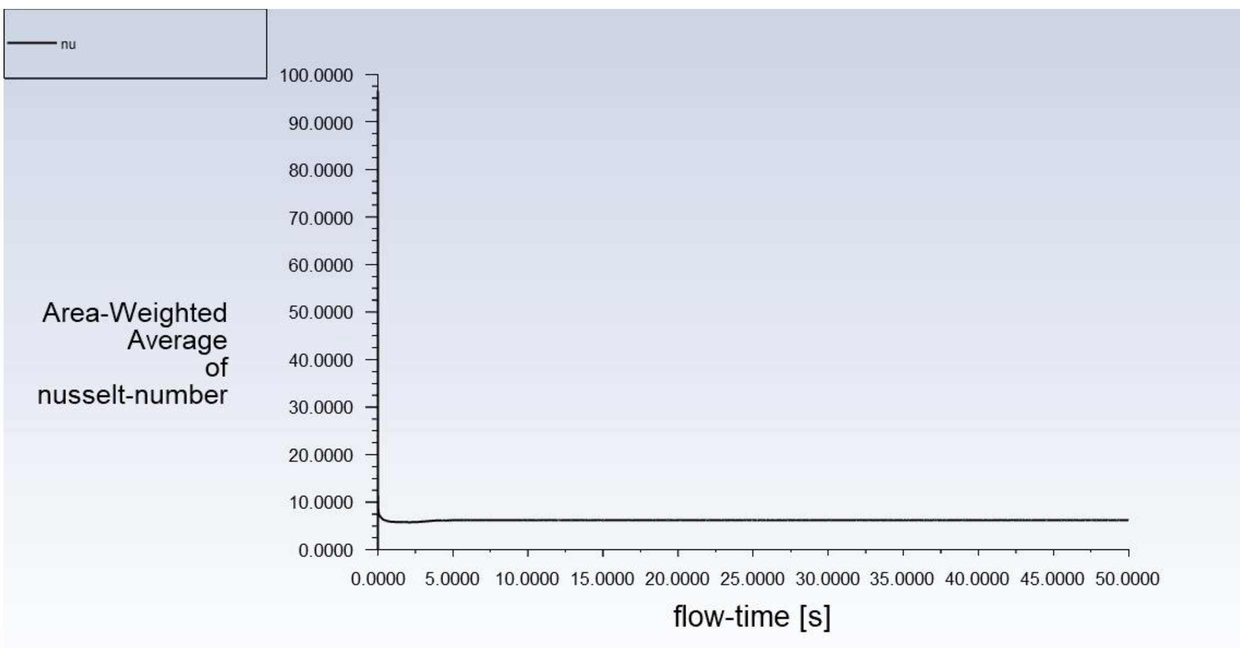
Lift forces are also generated due to the fluctuations of flow but the sum of these forces in negligible.

$$Fl = Cl\frac{1}{2}\rho U^2 bl$$

$Fl = 0.2368925 \times 0.5 \times 1.22 \times 0.22^2 \times 0.01 \times 0.01$

$Fl = 6.99 \times$ 10^-7


cl
Cl
0.5000
0.4000
0.3000
0.2000
0.1000
0.0000
-0.1000
-0.2000
-0.3000
-0.4000
-0.5000
0.0000 5.0000 10.0000 15.0000 20.0000 25.0000 30.0000 35.0000 40.0000 45.0000 50.0000
flow-time [s]


nu
Area-Weighted Average of nusselt-number
100.0000
90.0000
80.0000
70.0000
60.0000
50.0000
40.0000
30.0000
20.0000
10.0000
0.0000
0.0000 5.0000 10.0000 15.0000 20.0000 25.0000 30.0000 35.0000 40.0000 45.0000 50.0000
flow-time [s]