ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Flow Past Different Geometries Arranged in Triangular Pattern

Harshit Srivastava, Vivek Kumar

Department of Chemical Engineering, IET Lucknow, Lucknow, India
Department of Chemical Engineering, NIT Jalandhar, India

ABSTRACT: Computational Fluid Dynamic Simulations of a two-dimensional steady state flow past objects arranged in a pattern in a cylinder are analyzed. This paper aims to study the effect of objects of different shapes and sizes arranged in a triangular pattern formed by joining the center of each figure on the flow characteristics such as vortex shedding, flow stability, drag and lift experienced by the objects. In this study we observe how the different shapes of these objects at various Reynold's number based on the characteristic length of the pattern and free stream speed of the flow affect the velocity, pressure profile and the shear stress on wall. Incompressible Navier Stokes equation in Ansys Fluent 14.0 which works on a finite volume based numerical method to solve the equations of continuity and momentum is used for the computational purpose. The study done finds application in industrial shell and tube heat exchangers for comparing heat transfer with different geometries and in bio medical field for understanding the blood flow with patterned blockages in vessel to avoid various diseases. This has been accomplished by making the walls moving and selecting the liquid that suits the property of blood. In this work, flow parameters including drag and lift coefficients, velocity contours, pressure contours, temperature contours, streamlines and vectors are calculated.

Keywords: Triangular pattern, Reynolds number, Transient, Flow stability, Vortex shedding

I. INTRODUCTION

What is Fluid? A Fluid is anything that flows, usually a liquid or gas, the latter being distinguished by its great relative compressibility.

In recent years, many new problems and phenomenon have been discovered with the flow of liquid across some obstacle placed in its path. The main aim of our study is how flow changes with coming up of an obstacle and observing how it is affecting the obstacle and stress on the walls of cylindrical pipe. The property of the fluid that causes the stress is viscosity.

A. LOW REYNOLDS NUMBER FLOW

When Re<<1, viscous force dominates the flow and inertia is negligible.

B. DRAG

The force on the body is linearly related to the velocity and viscosity, DRAG coefficient is inversely proportional to Reynolds's number.
It is interesting to note that pressure and viscous shear stress on the body surface contribute comparable amount to drag.

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

C. QUASI-STEADINESS

d/dt Term vanishes at low Reynolds's number, it is immaterial that the relative velocity of the body or fluid is steady or not. The flow at any instant is same as if the boundary motion at that instant.

At low values of Reynolds's number, when viscosity is dominant, the diffusion is rapid and vorticity spreads out a way in all directions. Vorticity is local acceleration of fluid elements; the only factor that can generate vorticity is viscosity.

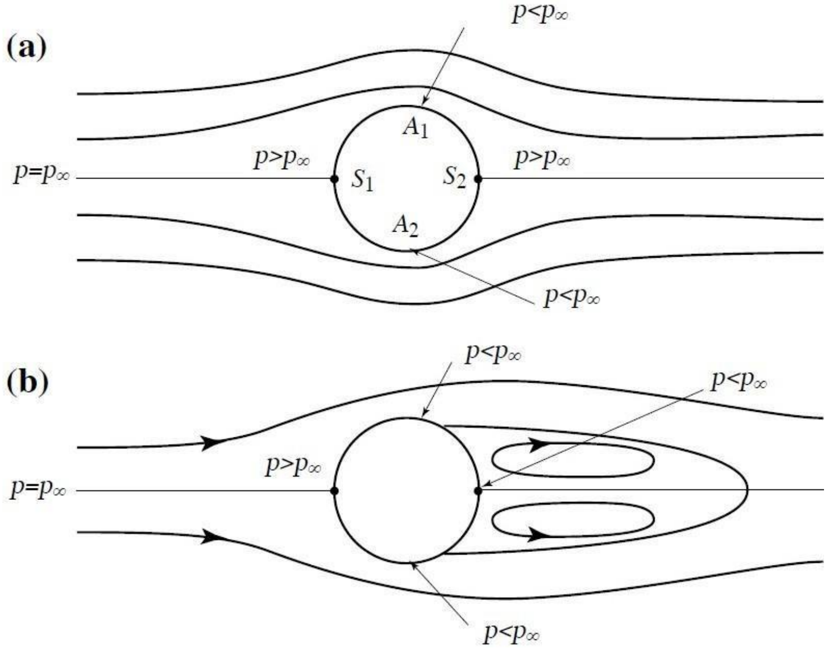


Fig 01: Sketch of streamlines and pressure for flow past a circular cylinder (a) Idealized flow of fluid with no viscosity; (b) separated flow at fairly high Reynolds number in a viscous fluid

D. HIGHER REYNOLDS NUMBER

At higher Re the wake becomes turbulent, in order to estimate the force on a body, it is necessary to work out the distribution of pressure.

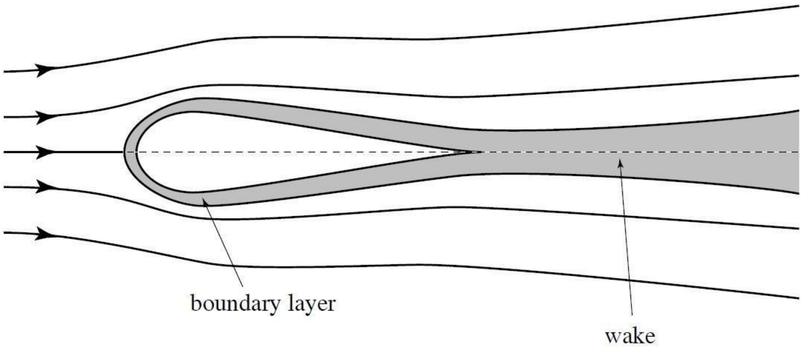


FIG 02: Sketch for boundary layer and wake for steady flow at high values of N_{re} past a symmetric streamline body

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

E. LIFT

For a symmetric streamlined body flow separation occurs only very near to the trailing edge, and direct viscous drag is more important. However, if such a streamlined body (or wing) is tilted so that the oncoming flow makes an angle of incidence with its center plane, viscosity again has an important effect. In general, a non-viscous flow past a wing at incidence would turn sharply round the trailing edge, where the velocity would be extremely high and the pressure extremely low. As the flow starts up from rest, viscosity causes separation at the corner, a concentrated vortex is shed and left behind, and thereafter the flow is forced to come tangentially off the trailing edge. In order to achieve this tangential flow, the velocity on top of the wing must increase and the velocity below must decrease. It follows from Bernoulli's equation that the pressure above the wing must fall, and that below rise, so a transverse force is generated.

F. VORTEX SHEDDING

In fluid dynamics, Vortex shedding is an oscillating flow that take place when a fluid such as air or water flow past a bluff body at certain velocities , depending on shape and size of the body . In this flow, Vortices are created at the back of the body and detach periodically from either side of the body.

Vortices are often thought of as regions of high vorticity, but there is no such universal threshold over which vorticity is considered high. More alarmingly vorticity may also be high in parallel shear flows where no vortices arepresent.

(a)

(b)

FIG 03: (a), (b) Some depictions on vortex shedding:

G. STROUHAL NUMBER:

In dimensional analysis, Strouhal number is a dimensionless number describing oscillating flow mechanism. Strouhal number is a integral part of the fundamentals of fluid mechanics.

$$St = \frac{f\text{L}}{\text{u}}$$

Where f is frequency of vortex shedding
L is characteristic length
U is flow velocity

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

For large Strouhal number viscosity dominates fluid flow

For low Strouhal number, high speed, Quasi-Steady state portion of movement dominates the oscillation.

H. FLUTTER& BUFFETING

Flutter as a controlled aerodynamic instability phenomenon is used intentionally and positively in windmills for generating electricity and in other works like making musical tones on ground-mounted devices, as well as on musical kites. Flutter is not always a destructive force; recent progress has been made in windmills for underserved communities in developing countries, designed specifically to take advantage of this effect.
Buffeting is high-frequency instability, caused by airflow separation or shock wave oscillations from one object striking another. It is caused by a sudden impulse of load increasing. It is a random forced vibration. Generally, it affects the tail unit of the aircraft structure due to air flow downstream of the wing.

II. MATERIAL AND METHOD

We used ANSYS Fluent 14.0 to do our simulations, firstly we construct our model using design modeler and the specification of figure is given below.

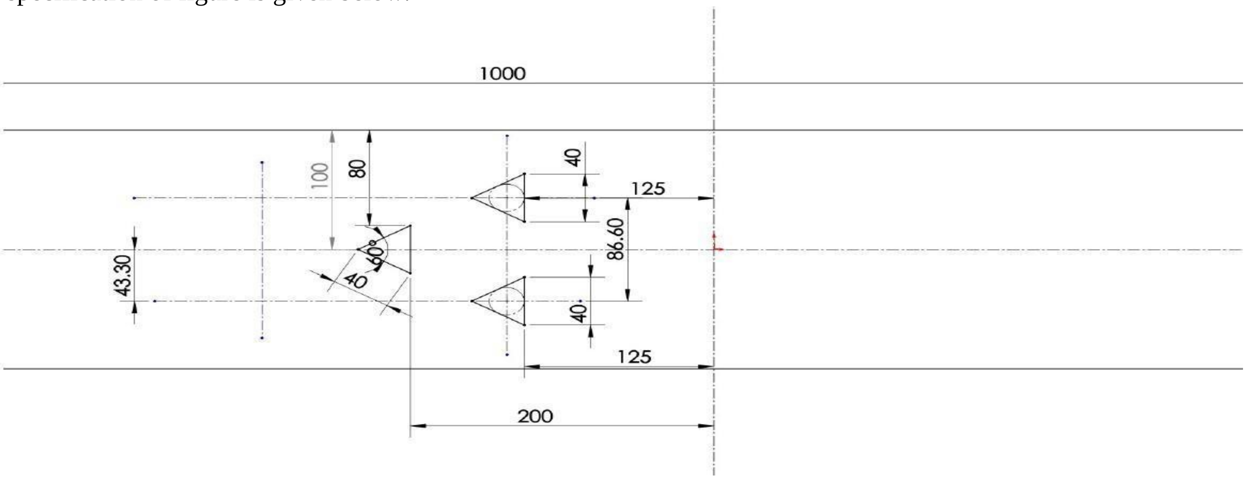


Fig 03: Specification of our drawing

We are performing our simulation on 2-D model in order to meet up the results with lower computation resources, here we are realizing a 3-D problem as a 2-D model as, it is nothing but a 3-D model of very small thickness.
Discretization of entire geometry into smaller finite volumes is done using ICEM CFD as our solver, i.e., Fluent, which works on Finite Volume Method algorithms, Mapped Meshing technique along with Relevance to center as ‘fine’ is employed in our simulation, meshing type falls under CFD solver category.

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

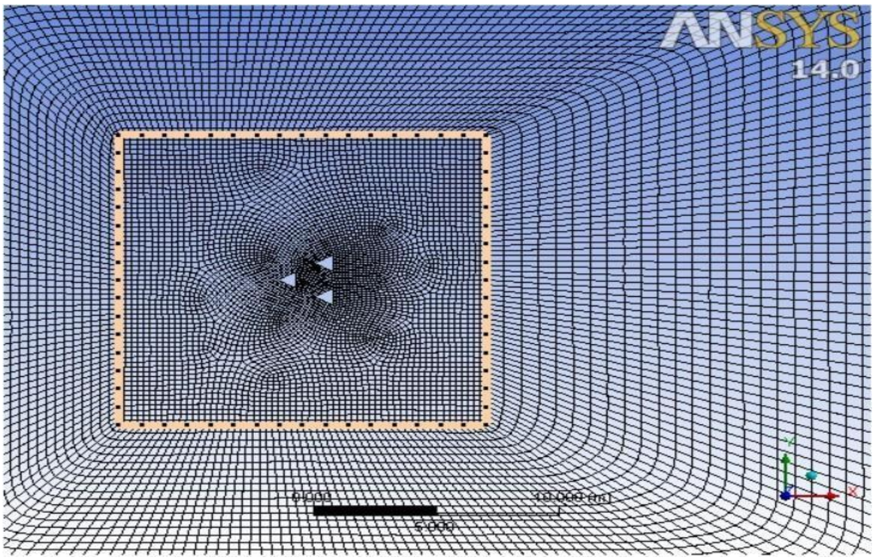


Fig 04: Meshing for vortex shedding.

The simulation was performed using commercial computational fluid dynamics package ANSYS Fluent 14.0, Incompressible Navier Stokes equation is used to study the flow of fluid past the triangular pattern of polygons. PISO scheme is used for Pressure Velocity Coupling, under Spatial Discretization for Gradient Least Square cell based is used, for pressure second order discretization scheme is used & for Momentum second order upwind scheme is used. Under Transient Formulation, bounded second order implicit with NITA (Non-Iterative Time Advancement) is used.

III. RESULT ANDDISCUSSION

As we see that the region around the first object is covered with traces of yellow region which corresponds to a different value of pressure compared to the region around it, so we have taken the value of this region for our study of pressure distribution.

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

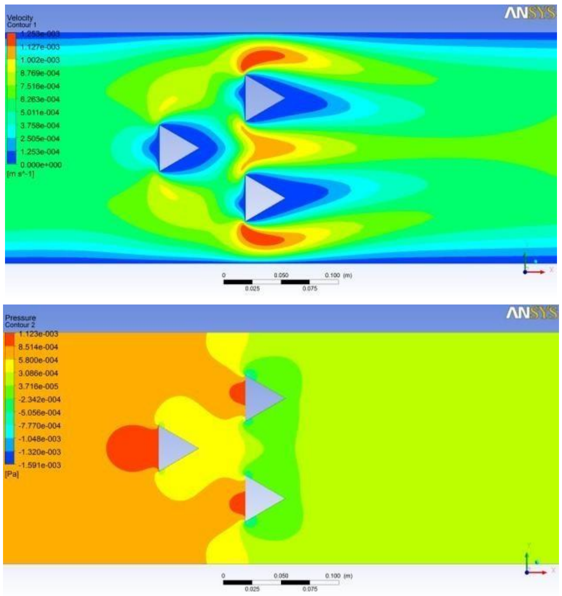
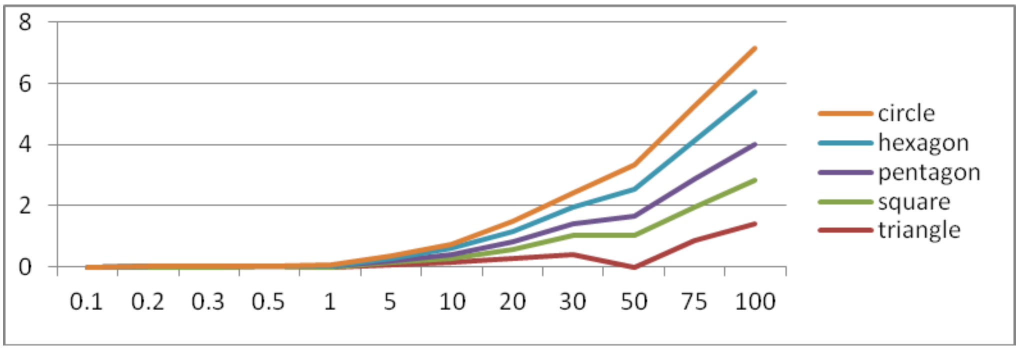


Fig 05:- Velocity & Pressure contour at N_{re} =50 Triangle Maximum drag condition.

On increasing Nre , the region around an object is more and more blue as blue region corresponds to low velocity region . At high Nre more and more streamlines are incident on an object after striking they transfer their momentum as they create a stagnation region having zero velocity maximum pressure which causes a great force along the axis which is responsible for drag.
If we notice carefully that pressure which is acting on first object is more than pressure at other two object, may be because of wall effect as these region lies near to the boundary so velocity of streamline will be slow so pressure is also less comparable to first object.

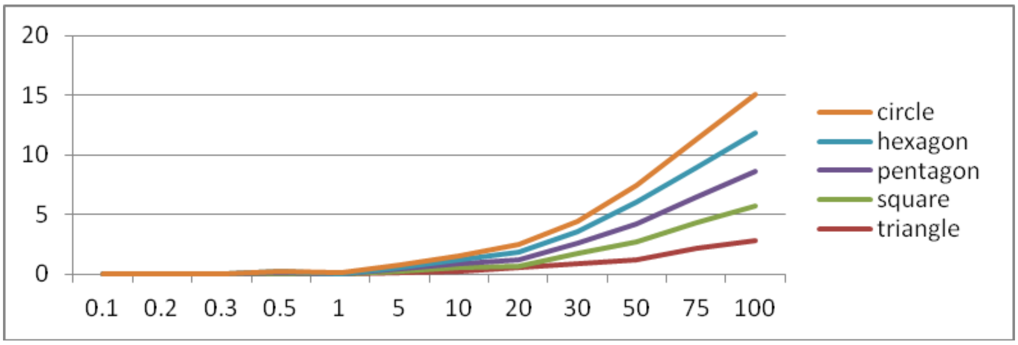
A. GRAPHS: -

Let us see the graph if we draw a line parallel to y- axis it means that we are keeping N_{re} constant then we can easily see the transition from a minimum side polygon to maximum side polygon.

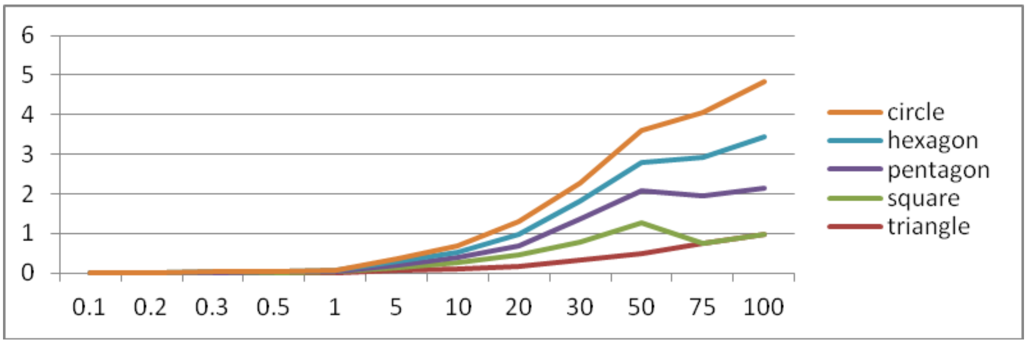


GRAPH 01: - MAX DRAG NRe V/S PRESSURE

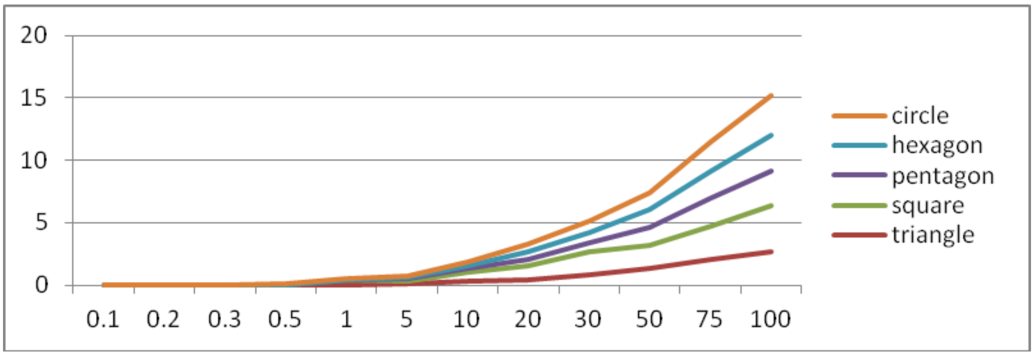
ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020



GRAPH 02: - MAX DRAG NRe V/S VELOCITY



GRAPH 03: - MINIMUM DRAG NRe V/SPRESSURE



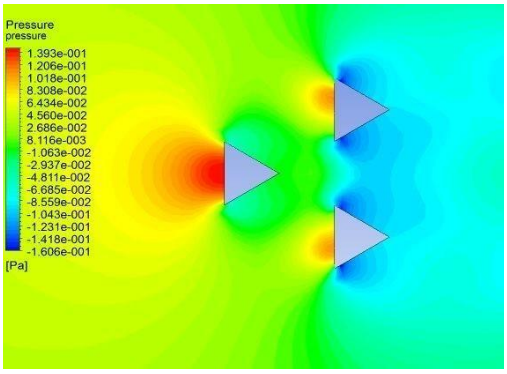
GRAPH 04: - MINIMUM DRAG NRe V/SVELOCITY

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

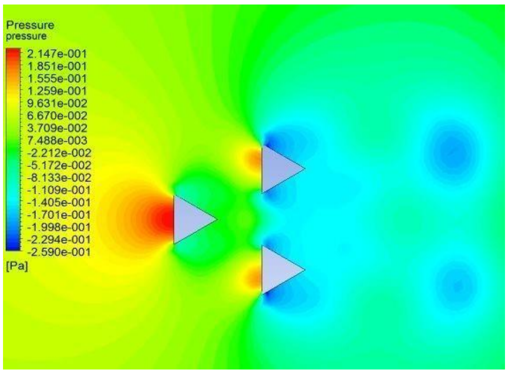
B. RESULTS FOR TRANSIENT ANALYSIS

Triangle (max drag):

a) Pressure contours

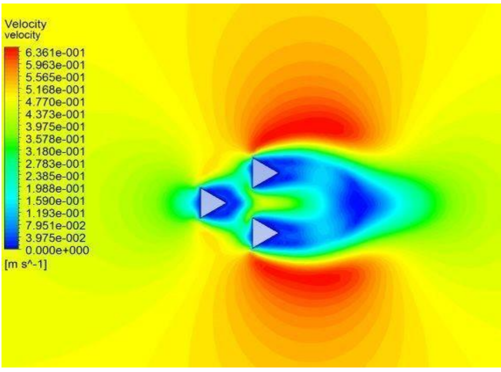


Nre=95

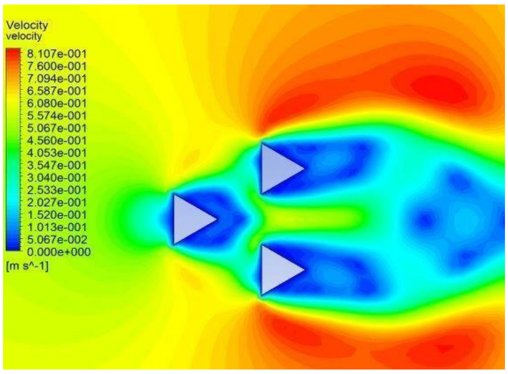


Nre=120

b) Velocitycontours

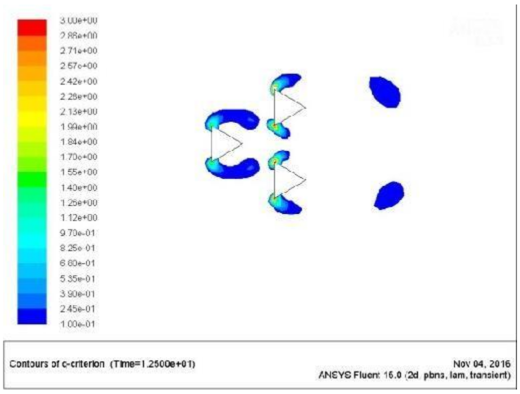


Nre=95

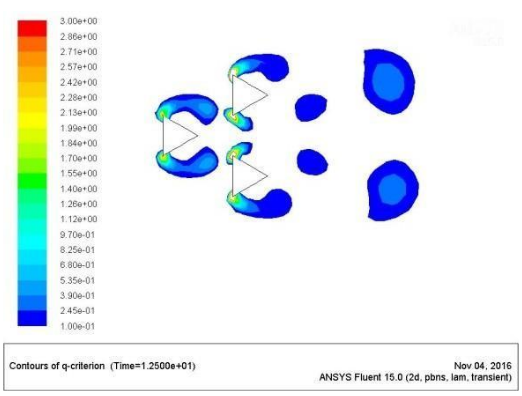


Nre=120

c) Q-criterion



Nre=95

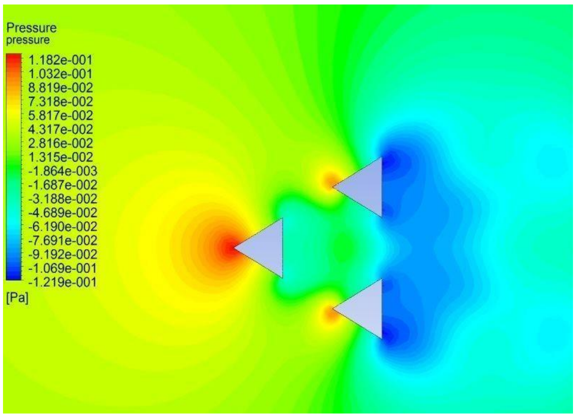


Nre=120

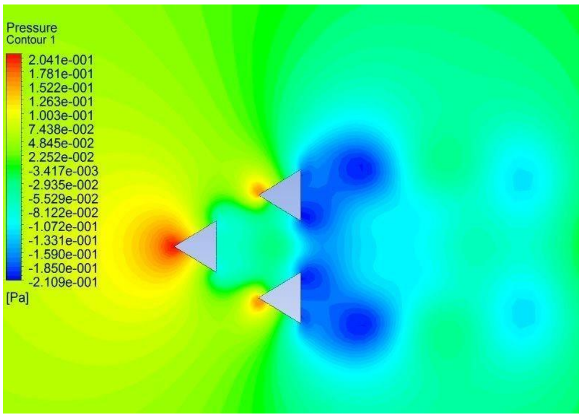
ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Triangle (Min drag):

a) PressureContour

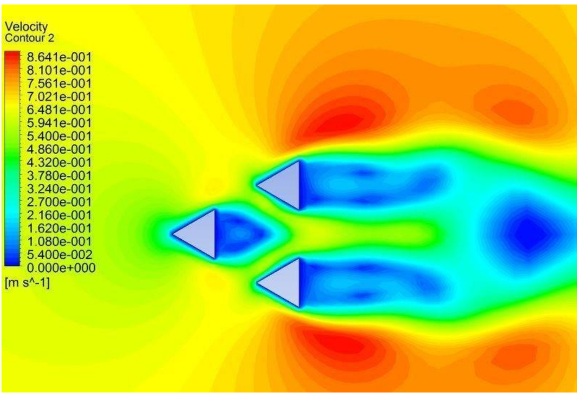


Nre=85

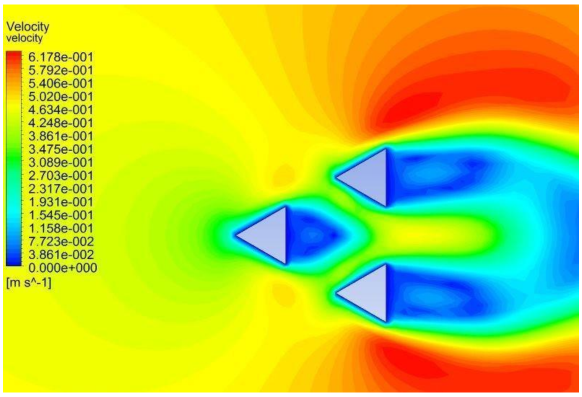


Nre=120

b) VelocityContours

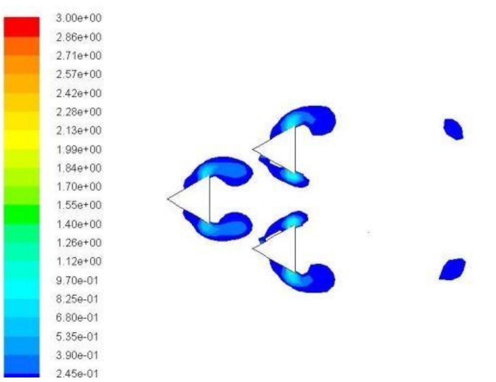


Nre=85

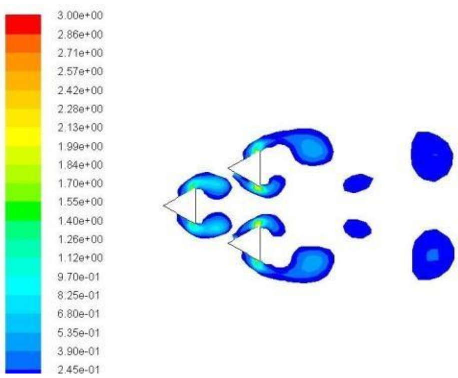


Nre=120

c) Q-criterion



Nre=85

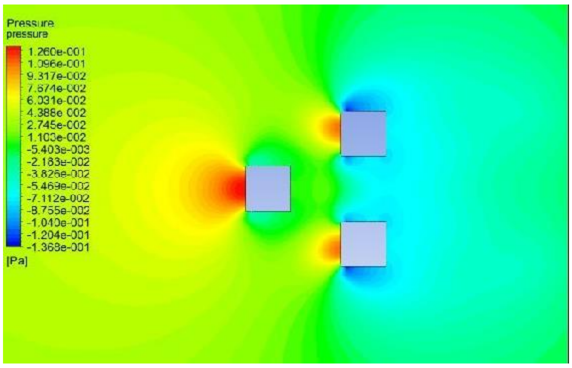


Nre=120

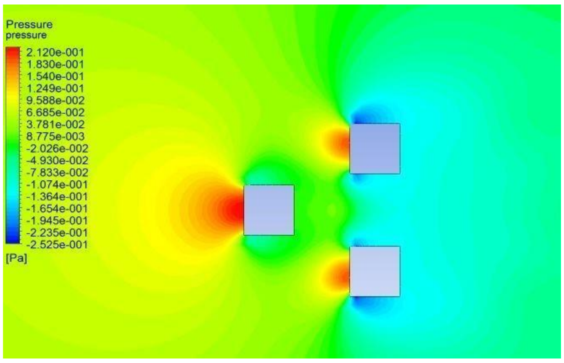
ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Square (max drag):

a) Pressurecontours

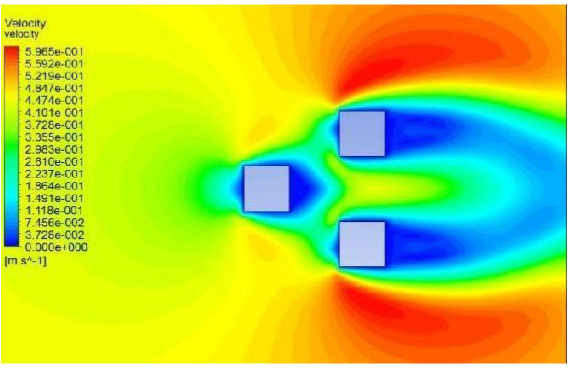


Nre=90

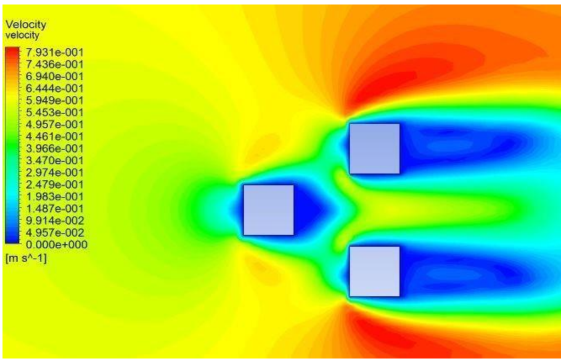


Nre=120

b) Velocitycontours

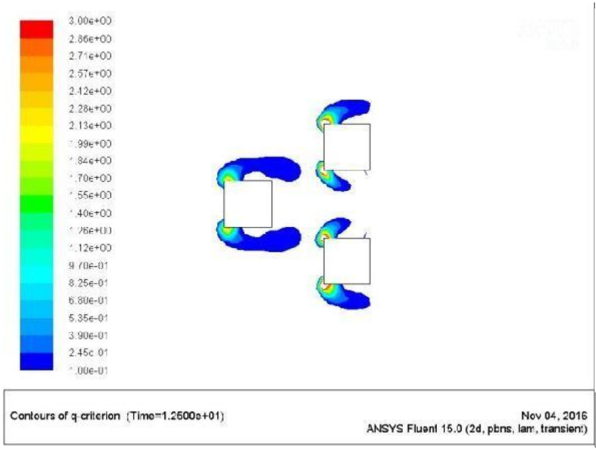


Nre=90

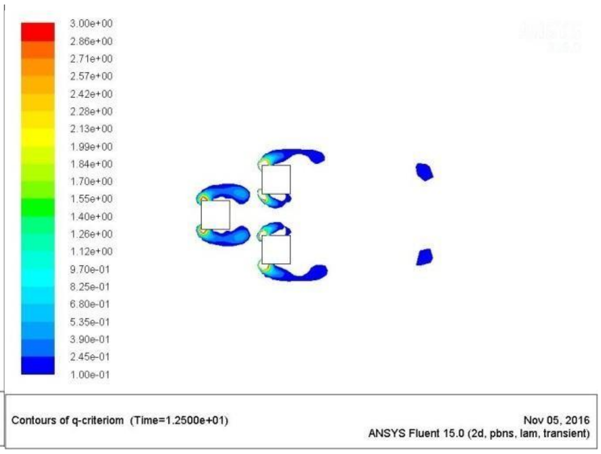


Nre=120

d) Q-criterion



Nre=90

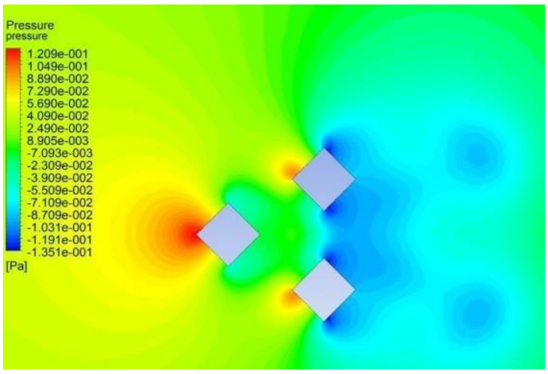


Nre=120

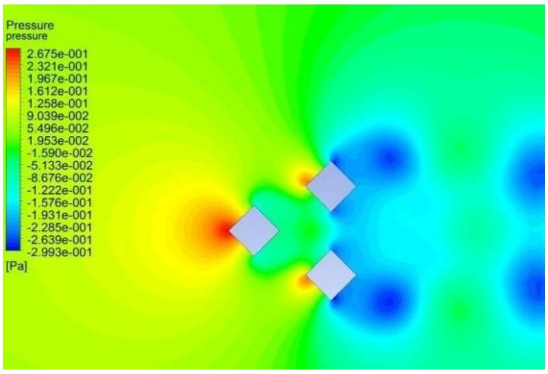
ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Square (Min Drag):

a) PressureContours

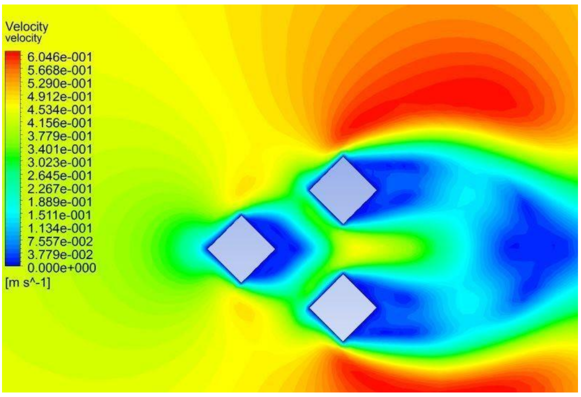


Nre=90

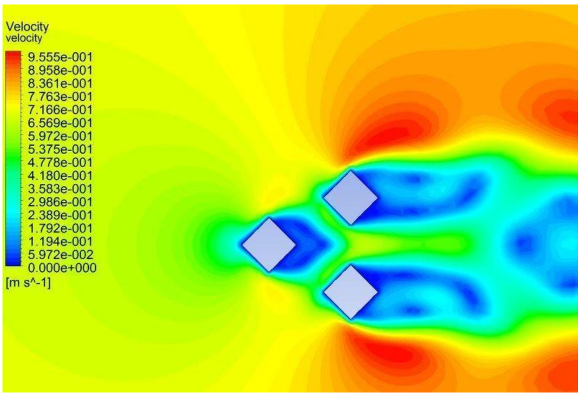


Nre=130

b) VelocityContours

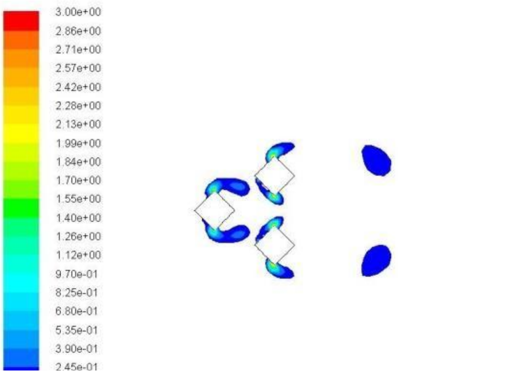


Nre=90

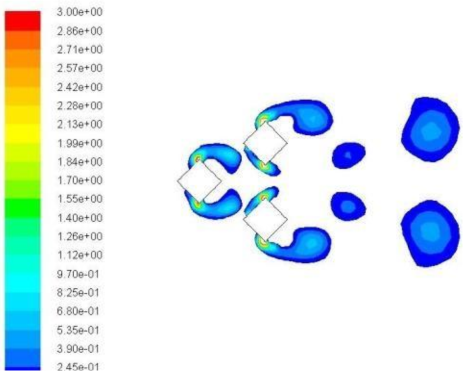


Nre=130

c) Q-criterion



Nre=90



Nre=130

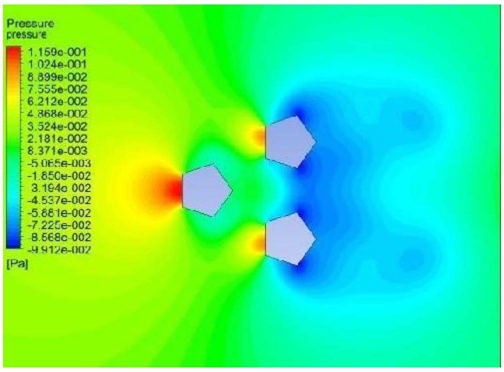
ISSN: 2350-0328

International Journal of Advanced Research in Science, Engineering and Technology

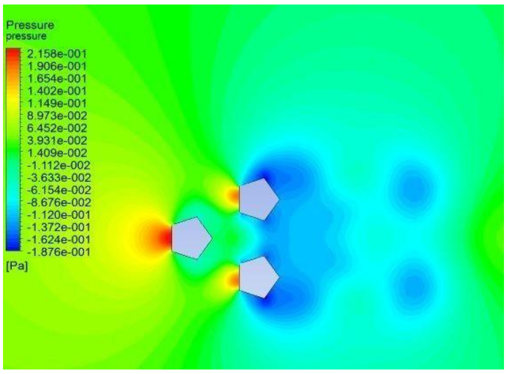
Vol. 7, Issue 12, December 2020

Pentagon (max drag):

a) Pressurecontours

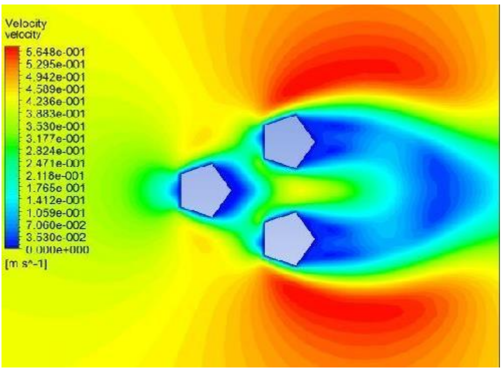


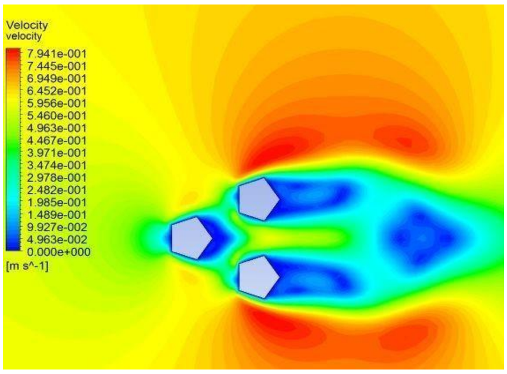
N_{re}=90

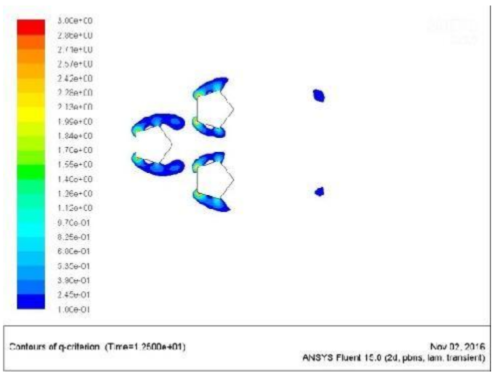


N_{re}=120

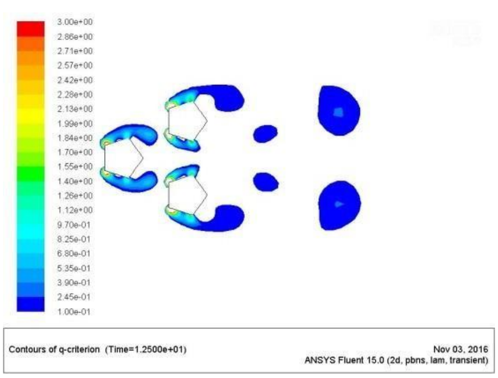
b) Velocitycontours







N_{re}=90

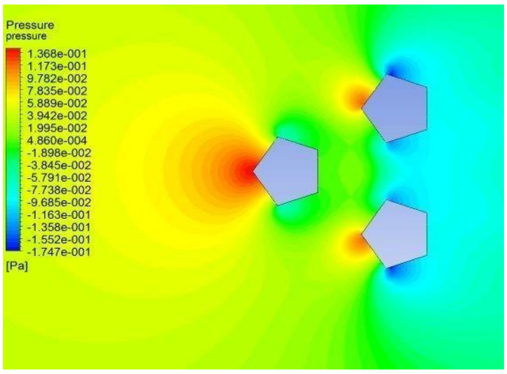


N_{re}=120

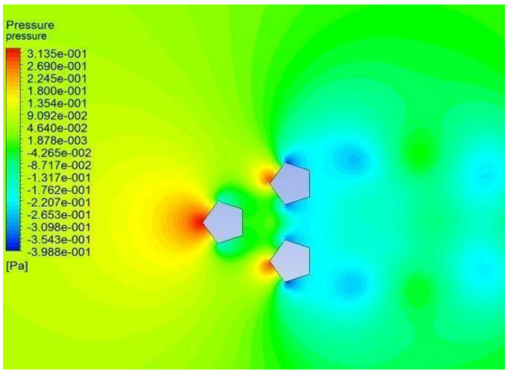
ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Pentagon (min drag):

a) PressureContour

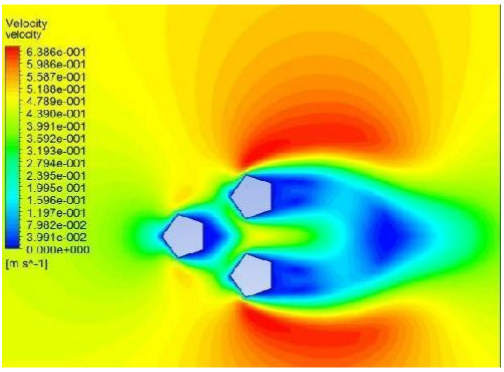


Nre=95

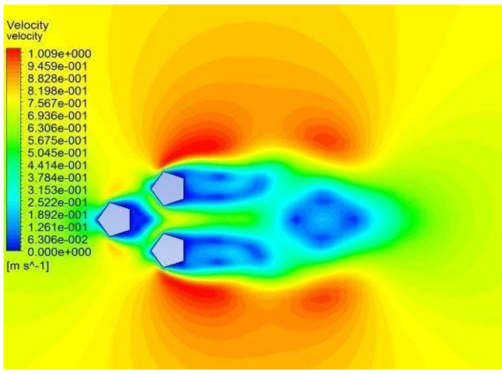


Nre=145

b) Velocitycontour

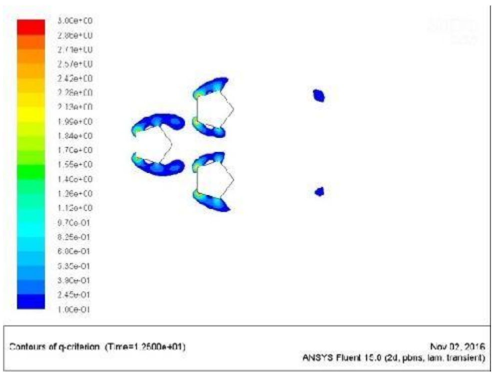


Nre=95

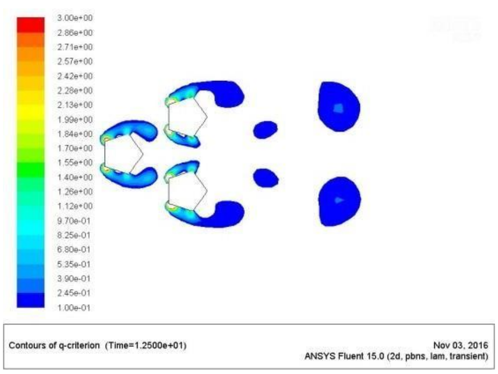


Nre=145

c) Q-criterion



Nre=90

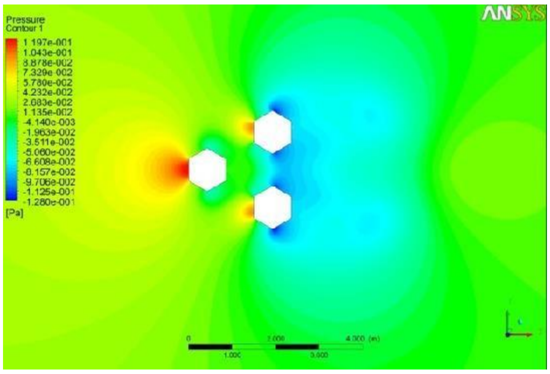


Nre=120

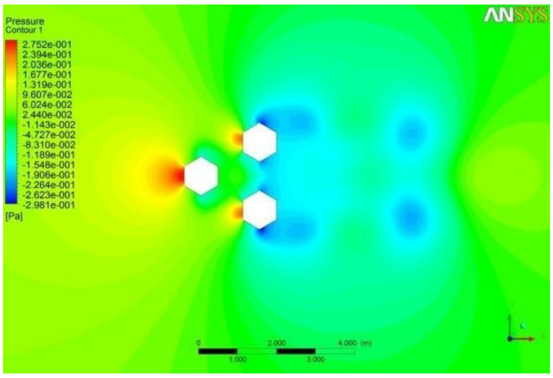
ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Hexagon (max drag):

a) PressureContour

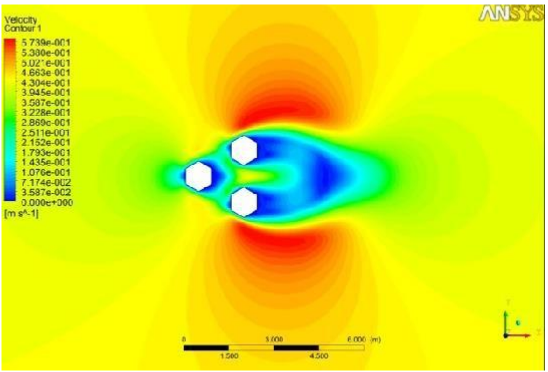


Nre=85

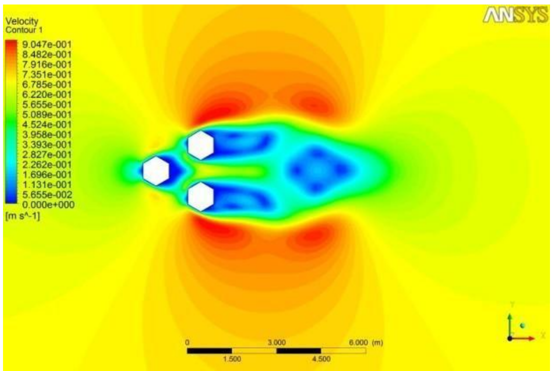


Nre=1453

b) Velocitycontour

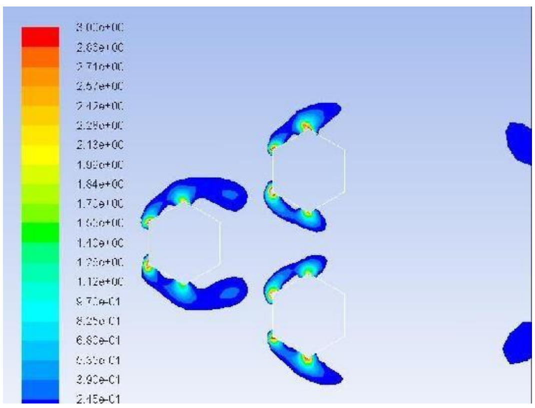


Nre=85

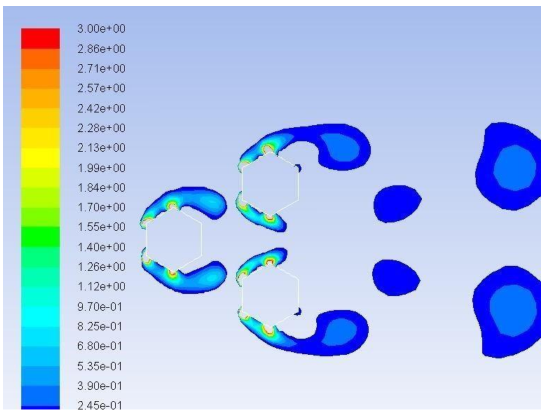


Nre=135

c) Q-criterion



Nre=85

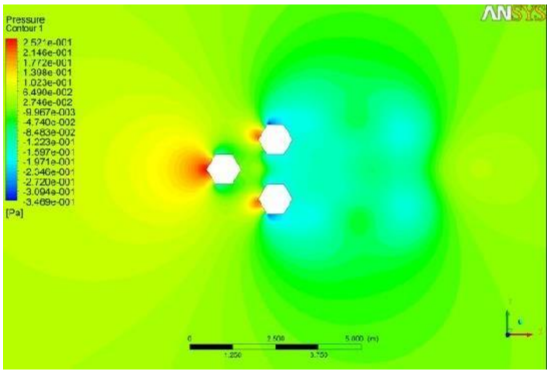


Nre=135

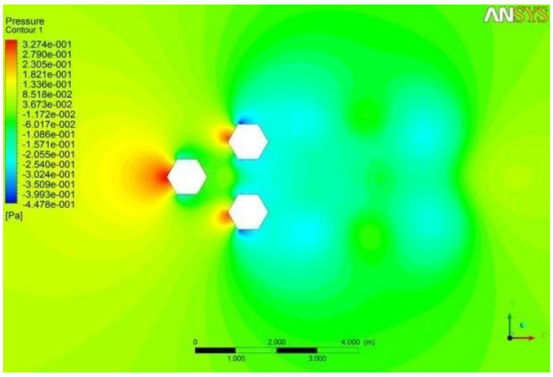
ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Hexagon (min drag):

a) PressureContour

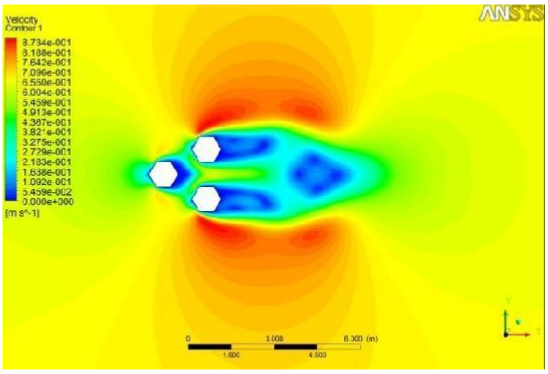


Nre=100

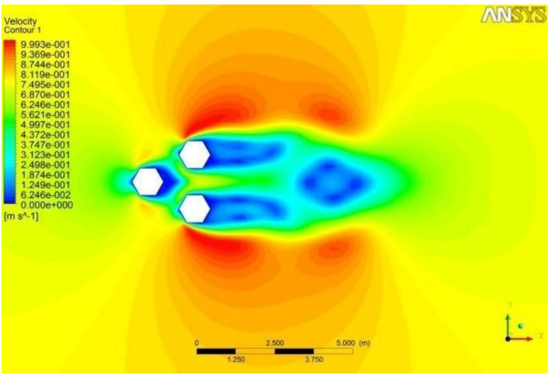


Nre=150

b) Velocitycontour

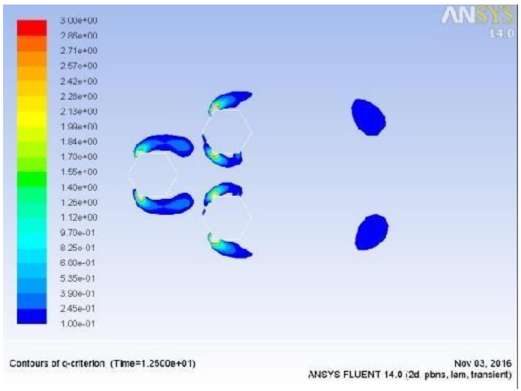


Nre=100

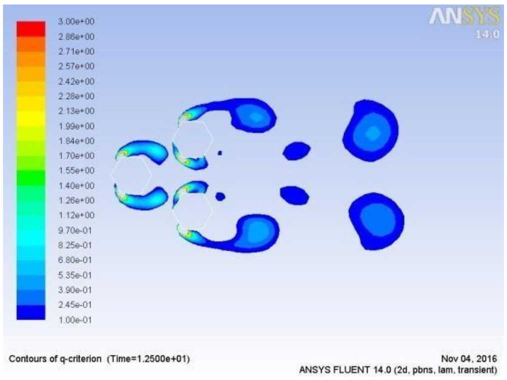


Nre=150

c) Q-criterion



Nre=100

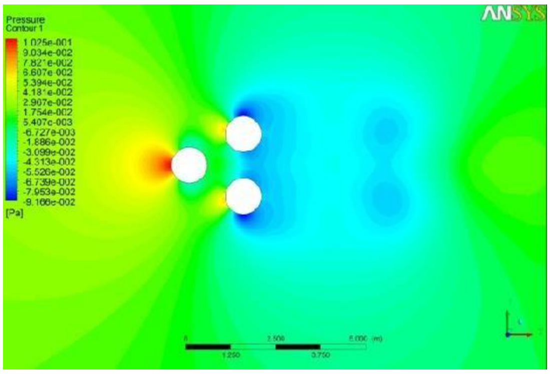


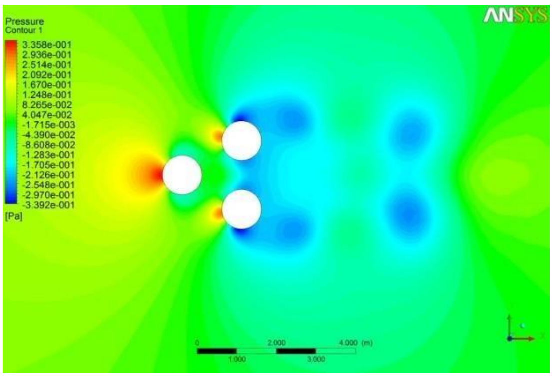
Nre=150

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

Circle:

a) Pressure Contour

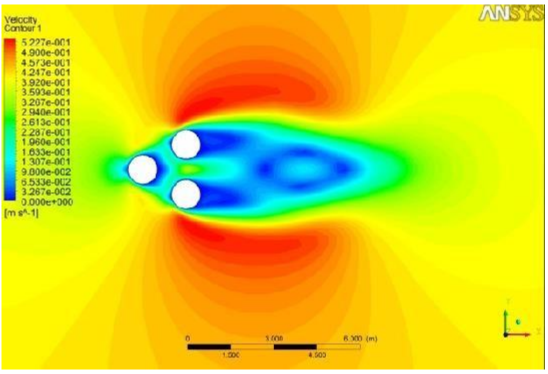


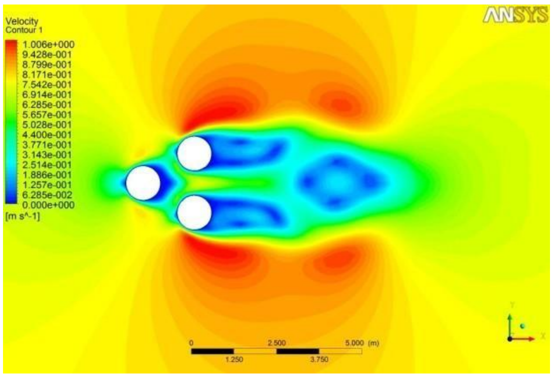


Nre=80

Nre=150

b) Velocitycontour

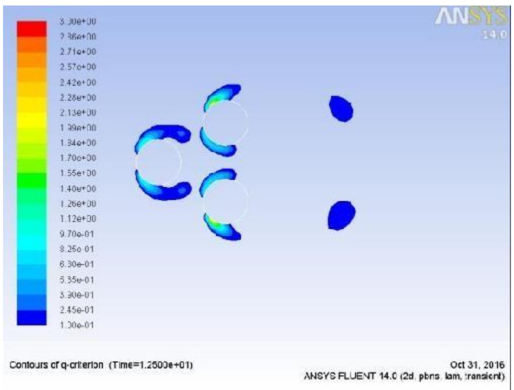


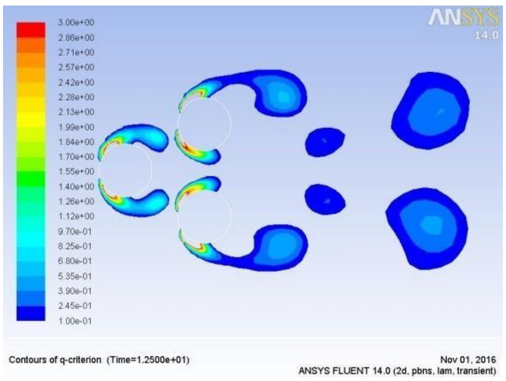


Nre=80

Nre=150

c) Q-criterion





Nre=80

Nre=150

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

IV. CONCLUSION

- On increasing the Reynolds's number, velocity is also increasing and the pressure on the first object is also increasing.

- Pressure on the first object is increasing so the force is also increasing and as we know that drag is force along the horizontal axis. Hence it also increases as we increase the N_{re}, keeping the remaining conditions same.

- Ongoing from minimum side polygon i.e., Triangle to maximum side polygon i.e. Circle we generally see an increase in velocity and transition in pressure profile also.

- As we are increasing the N_{re} the wall effect is also increasing as we can see the fluid along the wall is becoming slower and maximum velocity flow has been concentrated at the center and it is confining to a small region.

- Transient analysis is done for both minimum and maximum drag orientation for all set of polygons and we observe the following:

For minimum drag orientation as the side of the polygons vary in ascending order (triangle to Circle) Reynolds number for initialization of vortex shedding increases from 85 to 100 but for circle vortex shedding is initialized at low Reynolds number, i.e., 80. At stabilizing condition for vortex shedding, Reynolds number increases from 120 to 150 for triangle to circlerespectively.

Polygons (Min Drag)	Reynolds Number	
	Vortex Initialization	Vortex Stabilization
Triangle	85	120
Square	90	130
Pentagon	95	145
Hexagon	100	150
Circle	80	150

For maximum drag orientation as the side of the polygons vary in ascending order (triangle to Circle) Reynolds number for initialization of vortex shedding remains constant in range of 85 to 90 except for circle whereas for stabilizing condition of vortex shedding, Reynolds number increases from 120 to 150 for triangle to circle, respectively.

Polygons (Max Drag)	Reynolds Number	
	Vortex Initialization	Vortex Stabilization
Triangle	85-90	120
Square	85-90	120-125
Pentagon	85-90	125-130
Hexagon	85-90	135
Circle	80	150

ISSN: 2350-0328
International Journal of Advanced Research in Science, Engineering and Technology
Vol. 7, Issue 12, December 2020

REFERENCES

1. T.J PEDLEY "INTRODUCTION TO FLUID DYNAMICS"
2. MCCABE AND SMITH "UNIT OPERATION OF CHEMICAL ENGINEERING"
3. SCHLICHTING "INTRODUCTION TO BOUNDARY LAYER THEORY"
4. G.HALLER "AN OBJECTIVE DEFINITION OF A VORTEX"
5. A.LIFSCHITZ " ESSENTIAL SPECTRUM AND LOCAL STABILITY CONDITION IN HYDRODYNAMICS" VOL152,ISSUES 3-4,1991
6. LUGT.H.J " THE DELIMA OF DEFINING A VORTEX "
7. IMAGES FOR VORTEX SHEDDING: - https://en.wikipedia.org/wiki/Vortex_shedding