ISSN 2278-3091

Volume 10, No.2, March - April 2021

International Journal of Advanced Trends in Computer Science and Engineering

Available Online at http://www.warse.org/IJATCSE/static/pdf/file/ijatcse691022021.pdf

https://doi.org/10.30534/ijatcse/2021/691022021

Numerical Analysis of Unsteady Laminar Flow past a Pentagonal Obstacle

Farhan Ahmed Shaikh[1], Ahsanullah Baloch[2], Shafquat Ali Lashari[3], Shaharyar Khan[4] and Mir Afzal Talpur[5]

[1]Lecturer, Department of Mathematics, Federal Government Degree College Hyderabad –Pakistan
Email: farhankhalil.shaikh@gmail.com
[2]Prof. Dr., Department of Computer Science, ISRA University Hyderabad –Pakistan
Email: ahsanullah.baloch@isra.edu.pk
[3]Lecturer, Department of Mathematics, DUET University Karachi – Pakistan
Email: shafquat.lashari@gmail.com
[4]P.G Student, Department of Computer Science, ISRA University Hyderabad – Pakistan
Email: sunnnykhan1@gmail.com
[5]Asst. Prof., Department of Computer Science, ISRA University Hyderabad – Pakistan
Email: afzal.talpur@isra.edu.pk

ABSTRACT

The current article dispenses the numerical investigation of a two dimensional unsteady laminar flow of incompressible fluid passing a regular pentagonal obstacle in an open rectangular channel. The centre of attention of this work is the comparison of drag coefficients estimated for two distinct cases based on the orientation of face and corner of an obstacle against the flow direction. The numerical results shows that the corner – oriented obstacle bring about 42% larger value of drag coefficient at Re = 500 than face – oriented obstacle. The substantial growth in the expanse of vortex behind obstacle (presented as a function of fluid inertia 25 < Re < 500) is analyzed through the contours and streamline patterns of velocity field. The two eddies in the downstream become entirely unsymmetrical at Re = 500 for both the cases, whereas; the flow separation phenomena occurs a bit earlier in the face – oriented case at Re = 250. Two dimensional Pressure – Based – Segregated solver is employed to model the governing equations written in velocity and pressure fields. The numerical simulations of unsteady flow are presented for 50 seconds time frame with time step 0.01 by using one of the best available commercial based Computational Fluid Dynamics (CFD) software, ANSYS 15.0.

Key words: Pentagonal Obstacle, Drag Coefficient, Wake generation, Von Karman Effects, Reynolds Number.

1. INTRODUCTION

The present work is subjected to numerical investigation of flow past a pentagonal obstacle surrounded by an incompressible fluid in a rectangular channel. A flow around an obstacle is one of the most studied problems in CFD for investigating the phenomena of flow separation, wake generation, vortex enhancement behind the obstacle and the determination of hydrodynamic forces on the surfaces of obstacle. Many researchers have been studied the phenomena of bluff body either by adopting unconventional shaped bodies or by altering the geometry of well known shaped bodies in order to observe significant differences in the characteristics of flow [1], [2]. In [3], two inline square cylinders were investigated for examining the influence of Reynolds number over flow characteristics, it is noticed that the drag coefficient for upstream cylinder continuously increase with progressive Reynolds number whereas on other hand, drag coefficient of downstream cylinder gives negative value as fluid inertia arises. During the review of literature related to that specific problem, it has been observed that the obstacles having sharp corners (such as; square, rectangle or triangle etc.) builds a massive zone of separated flow and wake in the downstream of an obstacle which eventually generates periodic vortex shedding that can cause alternating low pressure zones on the sides of obstacle, this results in obstacle vibration normal to the flow direction and when shedding and natural frequencies of the obstacle gets close to each other resonance may appear which often leads to structure failure. This underline the urgency of preventing the resonance condition therefore designs of obstacle plays an crucial role in suppressing the drag and lift forces over the obstacle since these forces significantly depends on the structure of the wake region and size of the vortex behind obstacle [4], [5].

In [6], researchers investigated the different shaped cylinders includes; circular, square, corner-oriented hexagonal and face-oriented hexagonal. Their study primarily based on the comparison of drag coefficient determined at moderate Re = 200, it is beheld that the value of drag coefficient of face-oriented hexagonal is comparatively smaller than the drag coefficient found in the case of corner-oriented hexagonal cylinder. Such studies encourages scholars to conduct more research works on unconventional shaped cylinders in order to determine the effects of fluid inertia over the flow characteristics, specifically drag force, [7]. As discussed earlier that unconventional shaped obstacles getting too much attention of researchers since they come up with novelty in order to explore different behaviors of flowing fluid in open channel. Pentagonal, hexagonal and octagonal cylinders have been considered for experimental computation of wind load effects in the study of [8]. The reason behind using the pentagonal obstacle in the recent analysis is to observe the fluid forces, such as drag and lift, over the obstacle and also trying to control the size of the wake region, because these can be done by either altering the shape of an obstacle or its surroundings. The current CFD problem has potential relevance in many practical applications such as offshore structures, bridge piers, irrigation system, oil refinery industries, pipe lines, pharmacological and chemical reactions e.g. [9].

The present study is divided into two cases, in the first of which, one of the corners of pentagonal obstacle is placed against the flow direction while in the second case one of the walls of obstacle is placed against flow. The core of this work is to analyze the formation of vortex and wake region in the downstream based on the range of Reynolds number [25 , 500]. Furthermore; the hydrodynamic forces produced by the fluid inertia are also measured over the obstacle at the same range of Reynolds number. The mathematical equations that govern this problem are the well-known continuity and Navier – Stokes equations written in pressure-velocity formulation. The initial mechanism for the formation of vortex shedding and wake region is demonstrated and the fluid forces over the cylinder are computed over the different Reynolds number. The unsteady simulation of laminar flow is presented at time step 0.01 for 50 seconds, simulation results are only shown at full time step. The numerical results are obtained by using finite volume method based on SIMPLE algorithm in a commercial package of CFD, ANSYS 15.0

2. PROBLEM SPECIFICATIONS

The dynamic features of the pressure and velocity fields of unsteady laminar flow of linear fluid past a pentagonal obstacle are examined numerically in an open rectangular channel of stationary walls. The geometry of present problem with suitable dimensions is shown in following figures.

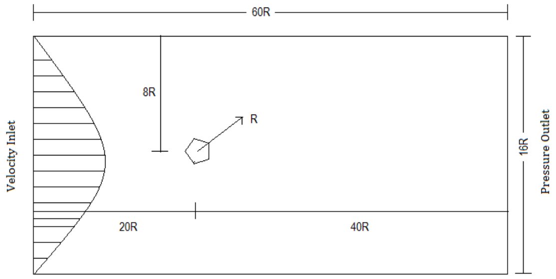


Figure – 1: Diagram of the flow field around corner – oriented pentagonal obstacle

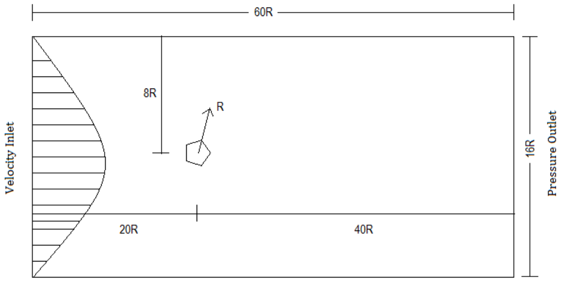


Figure – 2: Diagram of the flow field around face – oriented pentagonal obstacle

The obstacle is modelled as regular pentagon in an open rectangular channel. The flow from left to right with the obstacle of circum radius $'R'$ immersed in an incompressible fluid is considered for present study. The computational region comprises of width 16 times the circum radius while the upstream and downstream of the regions are 20 and 40 times of the circum radius of the obstacle, respectively.

3. MATHEMATICAL MODEL

The following system of partial differential equations (1 - 3) based on conservation laws are best to describe the physical problem of unsteady laminar flow passing 5 – sided regular polygon immersed in incompressible fluid in 2-dimensional rectangular coordinates written in pressure – velocity formulation with negligible body forces.

$$\frac{\partial(\rho u)}{\partial x}+\frac{\partial(\rho v)}{\partial y}=0 \qquad (1)$$

$$\left(\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right) \qquad (2)$$

$$\left(\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}\right)=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu\left(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right) \qquad (3)$$

The key step in the development of FVM is the integration of governing equations (1 – 3) over the 2D control volume concerning to discretize them at each nodal point of the computation domain. The integral form of governing equations (4 – 6) then transformed into system of linear equations which is quite easier to solve as compare to the system of nonlinear PDE's. The complete process of finite volume discretization can be seen in standard book of CFD.

$$\int_{cv}\nabla(\rho\tilde{u})\;dV=0 \qquad (4)$$

$$\begin{aligned}&\int_{\Delta t}\frac{\partial}{\partial t}\left(\int_{cv}(\rho u)\,dV\right)dt+\int_{\Delta t}\int_{A}n.(\rho\,\tilde{u}u)\,dAdt=-\int_{\Delta t}\int_{cv}\frac{\partial p}{\partial x}dVdt\\&+\int_{\Delta t}\int_{A}n.(\Gamma\,\nabla u)\,dAdt\end{aligned} \qquad (5)$$

$$\int_{\Delta t}\frac{\partial}{\partial t}\left(\int_{cv}(\rho v)\,dV\right)dt+\int_{\Delta t}\int_{A} n.(\rho\,\tilde{u}v)\,dAdt=-\int_{\Delta t}\int_{cv}\frac{\partial p}{\partial y}dVdt+\int_{\Delta t}\int_{A} n.(\Gamma\nabla v)\,dAdt \qquad (6)$$

3.1 Boundary Conditions

Following are the boundary conditions defined for the analysis of mathematical model of flow around pentagonal obstacle in an open channel.

i. Inlet – Parabolic Velocity Profile

$$V_x = U_m - U_m\left(\frac{y}{y_o}\right)^2$$

ii. Edges of pentagonal obstacle – No Slip condition $\left(u = v = 0\right)$

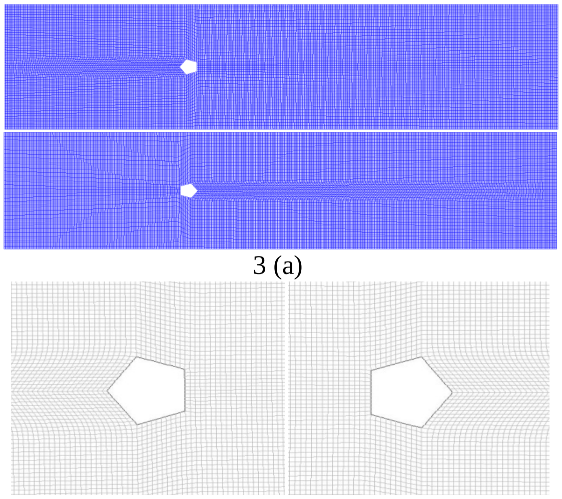
iii. Upper and Lower walls – No Slip

$$u = v = 0;\ \frac{\partial p}{\partial y} = 0$$

iv. Pressure Outlet – Condition $\left(P = 0\right)$

3.2 Numerical Scheme

In this investigation, the numerical solution of continuity and Navier – Stokes equations which governs the problem of transient laminar flow past pentagonal obstacle is obtained through FVM formulation. An approximate solution is determined by discretizing each term involving in the governing equations over the 2D control volume (CV). Second order upwind differencing scheme is adopted for the discretization of space derivatives while the coupled equation of pressure – velocity is formed by employing SIMPLE algorithm to describe pressure field. For time derivative, fully implicit technique is used due to its robustness and unconditional stability that suits to general transient flow calculations. The physical domain of this problem is meshed into quadrilateral elements to form a fine structured grid by using mapped face meshing option in ANSYS ICEM. The mesh of complete flow field with zoom in view of pentagonal obstacle can be seen in figure.3(a – b).



3 (a)

3 (b)

Figure – 3 (a – b): Extremely fine structured mesh of the flow field around corner & face – oriented pentagonal obstacle in2D rectangular channel. 3 (a) Entire domain 3 (b) Close view

Since the hydrodynamic forces such as; drag and lift are aim to be calculated on the edges, where their significance is quite essential, the element size in the mesh is considered small enough as per requirement for better approximation of results. The following equations (7 – 8) are the well known formulae to compute drag and lift coefficients.

$$C_D = \frac{F_D}{\rho V^2 R} \qquad (7)$$

$$C_L = \frac{F_L}{\rho V^2 R} \qquad (8)$$

Where F_D , F_L be the drag and lift forces, ρ be the density of a working fluid (for water, $\rho = 998.2$), V be the inlet velocity and $'R'$ is the circum radius of a two dimensional regular pentagon. The flow characteristics in this problem are presented as a function of dimensionless number $\mathrm{Re} = \frac{2R\rho V}{\mu}$ with $\mu = 1.003e^{-3}$ as viscosity of fluid.

3.3 Mesh Statistics

The computational domain of present study is divided into small quadrilateral elements to form the fine grid in the entire channel. The extremely fine mesh is obtained around the edges of pentagonal obstacle in order to observe and calculate flow characteristics very efficiently. Following is table showing the stats of our mesh.

Table. 1: Description of mesh statistics for both the cases.

Quantities	**Case – I** **Corner – Oriented**	**Case – II** **Face – Oriented**
No of Nodes	61282	62562
No of Elements	60648	61928
Element Size (m)	(Min.) $9.06e^{-6}$ (Max.) $5e^{-4}$	(Min.) $9.06e^{-6}$ (Max.) $5e^{-4}$
Element Quality (Avg.)	0.9967	0.9786
Orthogonal Quality (Avg.)	0.9941	0.9938
Skewness (Avg.)	$2.43e^{-2}$	$2.45e^{-2}$

4. EFFECTS OF FLUID INERTIA & DISCUSSION OF RESULTS

The analysis of unsteady flow past a pentagonal obstacle in a rectangular channel is conducted for two different cases; in case – I, one of the corners of pentagon is placed against the flow direction, whereas in case – II, one of the sides of regular pentagon is facing the direction of flow.

The study aims to evaluate the influence of Reynolds number on the hydrodynamic forces, specifically drag coefficient, over the surfaces of pentagonal obstacle. The numerical results shows that the face – oriented obstacle is statistically favorable since it reduced the drag coefficient value up to 42% when compare to the case of corner – oriented obstacle.

Although, in both cases, drag coefficient value gradually decreases with progression of fluid inertia in a similar manner, such development can be witnessed in fig. 04.

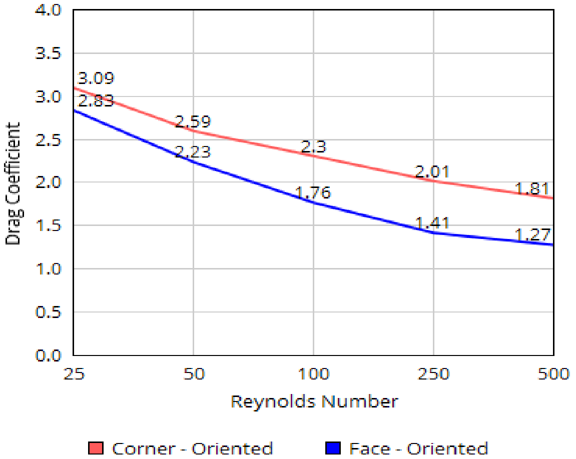
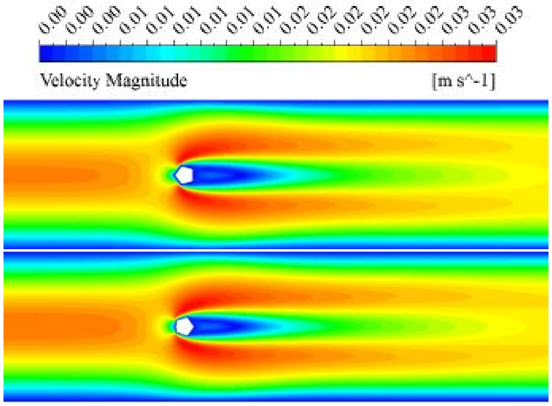
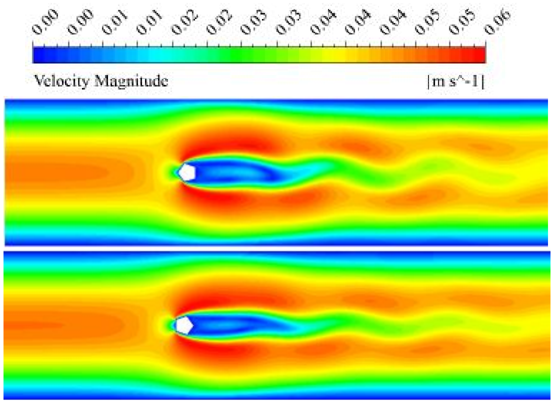


Figure – 4: Comparison of drag coefficient values for corner & face – oriented obstacle.

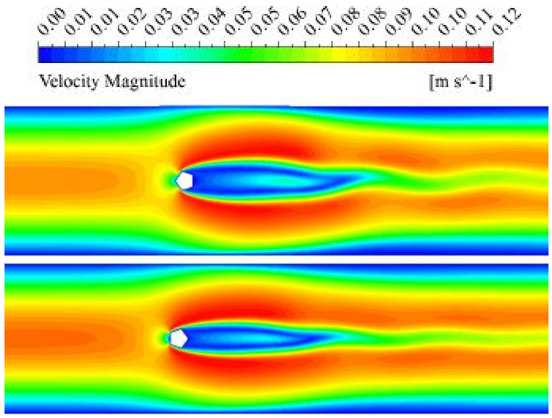
During the simulation of two cases, it is visualized and obtained through the velocity contours in figure 5(a – e) that the rate of enhancement in the size of recirculation zone behind the obstacle of case – I quicker than the case – II over the increasing inertia. This is due to the sharpness of obstacle's corner as the flow separation occurs earlier in this case which enforces the vortex to expand in the downstream faster, relatively.



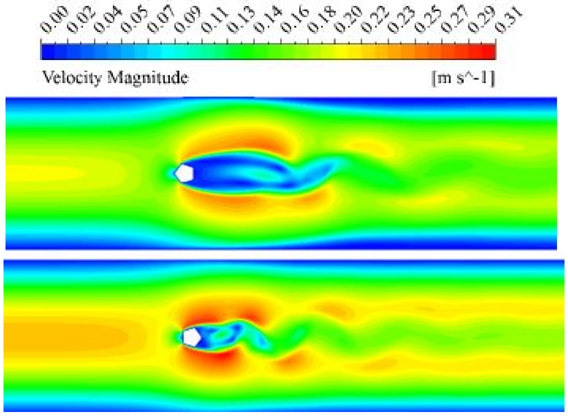
(a) Re = 25



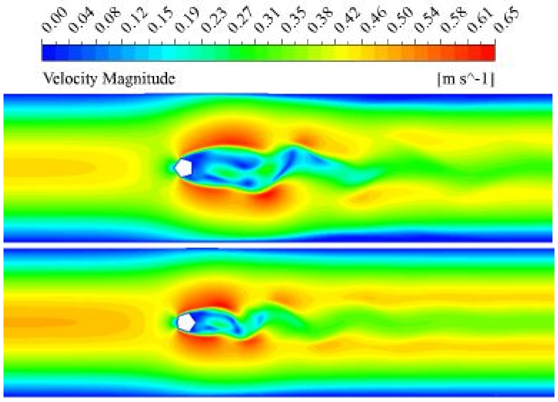
(b) Re = 50



(c) Re = 100



(d) Re = 250

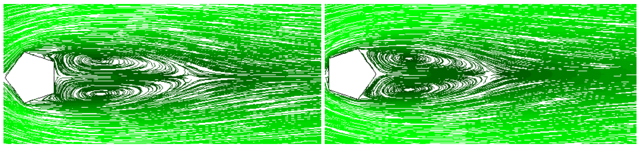


(e) Re = 500

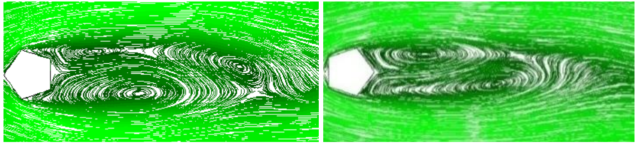
Figure – 5 (a – e): Velocity magnitude contours of case – I & case – II for different Reynolds numbers in a rectangular channel.

No separation of flow occurs at Re = 25 and drag is only because of skin friction, the behavior of flow is symmetrical in the downstream where two small eddies are appeared of the same size, angle of incident for obstacle do not have any impact on flow at this stage. At Re = 50, the two vortexes becomes slightly unsymmetrical in the downstream and boundary layer has started to separate from the edges of an obstacle but there is no significant increase in the size of wake region observed.

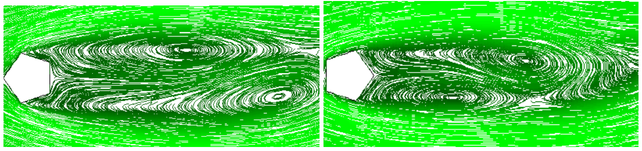
As the inertial forces become dominant the unsteady effect gets prominent, the flow started to separate from the edges of an obstacle and the two eddies behind it turns into unsymmetrical to highlights the von – karman effects at Re > 100. The streamline patterns of velocity magnitude in fig 6(a – e) are presented in this regard to visualize the overall development.



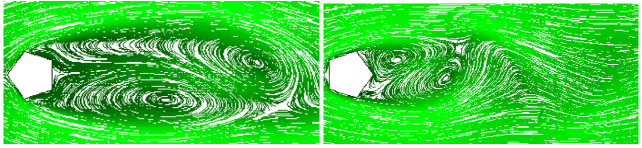
(a) Re = 25



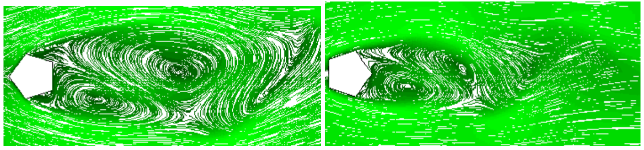
(b) Re = 50



(c) Re = 100



(d) Re = 250



(e) Re = 500

Figure – 6 (a – e): Streamlines of velocity magnitude of case – I & case – II for 25 < Re < 500 in a rectangular channel.

The vortexes behind the obstacle in case - II completely became unsymmetrical and the fluid has been separated from the edges of the cylinder as Re reaches to 250, the shedding of vortex is observed at this level which cannot be seen in case – I at Re = 250. Also, the estimated drag is found to be different in either case with negligible lift coefficient C_L. The comparison of lift and drag coefficients values is presented in table 2 based on Reynolds number.

Table. 2: The values of drag and lift coefficients at different Reynolds numbers of both the cases.

Re	Case – I Corner Oriented		Case – II Face Oriented	
	C_D	C_L	C_D	C_L
25	3.09	$1.51e^{-4}$	2.83	$3.04e^{-2}$
50	2.59	$1.71e^{-2}$	2.23	$2.03e^{-3}$
100	2.3	$1.67e^{-3}$	1.76	$1.92e^{-2}$
250	2.01	$1.32e^{-2}$	1.41	$1.66e^{-4}$
500	1.81	$1.75e^{-3}$	1.27	$1.13e^{-2}$

It is noticed that corner-oriented pentagonal obstacle having larger drag coefficient value than face-oriented pentagonal obstacle, this fact is in good agreement with existing findings in [6]. Also, it is known that the obstacle owns larger value of drag, produces relatively less pressure in the downstream and vice versa. The pressure contour is available in figure 7 for Re = 500 to closely visualize the pressure stagnation point and pressure difference in upper and lower streams of laminar flow in the computational domain.

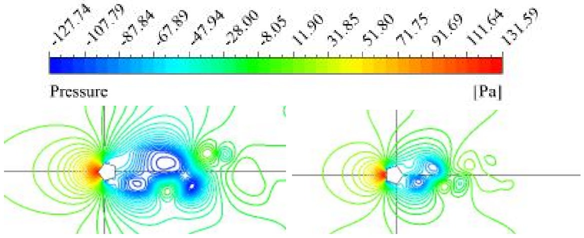


Figure – 7: Pressure contour of case – I & case – II for Re = 500 in a rectangular channel.

From the review of literature, it can be seen that the findings of this article are justifying the fact that obstacle having smooth rounded shape or flat face gives smaller value of drag coefficient when putting them against the flow direction [1]; whereas the shapes possesses sharp corners are comparatively produces larger drag value, [2], [7]. The case of Re = 500 is the case where both studies are found to have similar flow characteristics to some extent, the unsymmetrical eddies and phenomena of vortex shedding can be observed in both the cases in figure.5(e) & 6(e). During the entire simulation of both cases it is noticed that the value of drag coefficient is continuously decreasing with increasing Reynolds number, on the other side, value of lift coefficient increases as Re reaches to 500.

5. CONCLUSION

The analysis of five sided (Regular Pentagon) obstacle surrounded by incompressible laminar flow in a 2D rectangular channel is conducted to explore the effects of fluid inertia over the characteristics of flow. The first objective was to develop the numerical model of unsteady laminar flow of incompressible fluid, which has been achieved quite successfully. The FVM with SIMPLE algorithm, has been used in past by various researchers to conducting an analysis of flow around various shaped cylinders e.g, [10], [11] is found to be a good technique to validate the solution of equations which governs the fluid flow phenomena in the present study. For the numerical investigation of flow characteristics, the occurrence of flow separation from the edges of an obstacle is closely monitored over the predefined range of Reynolds number. It is noticed for Re = 250 that vortexes in the downstream became completely unsymmetrical and started to shed and ultimately form von karman street in case – II, whereas in case – I at the same Re, von karman effects are not shown properly. The generation of wake region behind the obstacle at full time step for each simulation is also observed and concluded that the shape and size of the recirculation zone is getting change with the progression of Re, which results in the pressure drop, the pressure contours are pretty much helpful to visualize these effects. The drag coefficient in the first case where one of the corners of obstacle placed against the flow has comparatively larger value than second case. It is concluded that the drag can be controlled or reduce to some extent if one of the flat side of pentagonal obstacle is placed against flow direction. Since the present study is confined to laminar flow with maximum Re = 500, one can extend the current investigation to turbulent flow in order to explore the behavior of flow past pentagonal obstacle. Moreover, the results of this simulation only presented at full time step, that is 50 secs, one can simulate the results at different time steps to highlight the inertial effects at small increment of time.

REFERENCES

[1] YOICHI YAMAGISHI, SHIGEO KIMURA, MAKOTO OKI AND CHISA HATAYAMA (2009) "Effects of corner cutoffs on flow characteristics around a square cylinder"
10th International conference on fluid control, measurement and visualization. Flucome 2009. August 17-21, 2009.
http://www.ihed.ras.ru/flucome10/cd/papers/215.pdf

[2] DONG-HYEOG YOON, KYUNG-SOOYANG AND CHOON-BUMCHOI, (2010). "Flow past a square cylinder with an angle of incidence" 2010 American Institute of Physics. doi:10.1063/1.3388857, Physics of fluids 22, 043603 (2010)
https://aip.scitation.org/doi/10.1063/1.3388857

[3] PRATISH P. PATIL AND SHALIGRAM TIWARI (2009). "Numerical investigation of laminar unsteady wakes behind two inline square cylinders confined in a channel"
Journal of engineering applications of computational fluid mechanics. Volume 3, issue 3, 2009.
https://www.researchgate.net/publication/236009516_Numerical_Investigation_of_Laminar_Unsteady_Wakes_Behind_Two_Inline_Square_Cylinders_Confined_in_a_Channel

[4] SINGH, SATYA & MITTAL, SANJAY. (2005). "Flow past a Cylinder: Shear Layer Instability and Drag Crisis." International Journal for Numerical Methods in Fluids, 47(1), 2005, pp. 75-98..
International Journal for Numerical Methods in Fluids. 47. 75-98. 10.1002/fld.807.
https://www.researchgate.net/publication/261991939_SP_Singh_and_S_Mittal_Flow_past_a_Cylinder_Shear_Layer_Instability_and_Drag_Crisis_International_Journal_for_Numerical_Methods_in_Fluids_471_2005_pp_75-98

[5] A. F. ABDEL GAWAD , N. A. MOHAMED , S. A. MOHAMED , M. S. MATBULY (2013). "Investigation of the channel flow with internal obstacles using large eddy simulation and finite-element technique."
Applied and Computational Mathematics 2013; 2(1) : 1-13 Published online February 20, 2013 (http://www.sciencepublishinggroup.com/j/acm) doi: 10.11648/j.acm.20130201.11
https://www.researchgate.net/publication/236260879_Investigation_of_the_Channel_Flow_with_Internal_Obstacles_Using_Large_Eddy_Simulation_and_Finite-Element_Technique

[6] HASSAN KARAMPOUR, ZHUOLIN WU, JULIEN LEFEBURE, DONG-SHENG JENG, AMIR ETEMAD-SHAHIDI AND BENJAMIN SIMPSON (2018). "Modeling of flow around hexagonal and textured cylinder"
Proceedings of the Institution of Civil Engineers – Engineering and Computational Mechanics, ICE,Publishing
https://doi.org/10.1680/jencm.17.00021
https://www.researchgate.net/publication/327945704_Modelling_of_flow_around_hexagonal_and_textured_cylinders

[7] SEYED REZA DJEDDI, ALI MASOUDI, PARVIZ GHADIMI (2013). "Numerical Simulation of Flow around Diamond Shaped Obstacles at Low to Moderate Reynolds Numbers"
American Journal of Applied Mathematics and Statistics, 2013, Vol. 1, No. 1, 11-20 Available online at http://pubs.sciepub.com/ajams/1/1/3. DOI:10.12691/ajams-1-1-3
https://www.researchgate.net/publication/314543166_Numerical_Simulation_of_Flow_around_Diamond-Shaped_Obstacles_at_Low_to_Moderate_Reynolds_Numbers

[8] GHOSH K, ISLAM M. Q, ALI M. (2014). "An experimental investigation of wind effect on pentagonal and hexagonal staggered cylinders." Procedia Engineering 90 (2014) 314 – 319
https://pdfs.semanticscholar.org/9286/3796900d341851b2673742035416db6cfdda.pdf?_ga=2.239750275.1409670163.1606807107-1669315334.1606680839

[9] BIN XIA AND DA-WEN SUN (2002),"Applications of computational fluid dynamics (CFD) in the food industry: a review." Computers and Electronics in Agriculture 34 (2002) 5–24. www.elsevier.com/locate/compag
https://www.researchgate.net/publication/222511397_Applications_of_computational_fluid_dynamics_CFD_in_the_food_industry_A_review

[10] MD. MAHBUBAR RAHMAN, MD. MASHUD KARIM AND MD. ABDUL ALIM (2007). "Numerical Investigation of Unsteady Flow Past A Circular Cylinder Using 2-D Finite Volume Method"
Journal of Naval Architecture and Marine Engineering, June, 2007. http://jname.8m.net
https://www.researchgate.net/publication/291036436_Numerical_investigation_of_unsteady_flow_past_a_circular_cylinder_using_2-D_finite_volume_method

[11] GERA, B. & SHARMA, PAVAN & SINGH, RAM. (2010). "CFD Analysis of 2D unsteady flow around a square cylinder"
International Journal of Applied Engineering Research, Dindigul. Volume 1, No3, 2010 (602-610)
https://www.researchgate.net/publication/275644199_CFD_Analysis_of_2D_Unsteady_Flow_around_a_Square_Cylinder

[12] VIKRAM C. K, DR. Y. T. KRISHNE GOWDA, DR. H. V. RAVINDRA, DR. C. J. GANGADARA GOWDA AND MANU (2011). “Numerical simulation of two dimensional unsteady flow past two square cylinders” International Journal of Technology And Engineering System (IJTES).
Jan –March 2011- Vol.2.No.3.
http://ijcns.com/pdf/374.pdf

[13] R. KERAMATI FARHOUD, SABER AMIRALAIE, GHADER JABBARI, SOLMAZ AMIRALAIE (2012) “ Numerical study of unsteady laminar flow around a circular cylinder”
Journal of civil engineering and urbanism. volume 2 issue 2: 63-67
https://www.researchgate.net/publication/268409852_Numerical_Study_of_Unsteady_Laminar_Flow_around_a_Circular_Cylinder

[14] ISLAM, TOUKIR & HASSAN, RAKIBUL. (2013). “Experimental and Numerical investigation of flow over a Cylinder at Reynolds Number 10^5.” Journal of Modern Science and Technology Vol.1.No.1. May 2013 Issue. Pp.52-60
https://www.researchgate.net/publication/305387181_Experimental_and_numerical_investigation_of_flow_over_a_cylinder_at_reynolds_number_10

[15] TOUKIR ISLAM, S.M. RAKIBUL HASSAN AND MOHAMMAD ALI (2013). “Flow separation phenomena for steady flow over a circular cylinder at low Reynolds number”
International Journal of Automotive and Mechanical Engineering (IJAME)
ISSN: 2229-8649 (Print); ISSN: 2180-1606 (Online); Volume 8, pp. 1406-1415, July-December 2013. DOI: http://dx.doi.org/10.15282/ijame.8.2013.28.0116
http://ijame.ump.edu.my/images/Volume_8/28_Islam%20et%20al.pdf