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/ijatcse631022021.pdf
https://doi.org/10.30534/ijatcse/2021/631022021

Numerical Investigation of Newtonian Fluid Flow around Sharp & Rounded Corner Rectangular Obstruction

Farhan Ahmed Shaikh[1*], Shaharyar Khan[2], Ahsanullah Baloch[3], Shafquat Ali Lashari[4]
[1]Lecturer, Department of Mathematics, Federal Government Degree College Hyderabad – Sindh, Pakistan.
Email: farhankhalil.shaikh@gmail.com
[2]P.G Student, Department of Computer Science, ISRA University Hyderabad – Sindh, Pakistan.
Email: sunnnykhan1@gmail.com
[3]Professor, Department of Computer Science, ISRA University Hyderabad – Sindh, Pakistan.
Email: ahsanullah.baloch@isra.edu.pk
[4]Lecturer, Department of Mathematics, DUET University Karachi – Sindh, Pakistan.
Email: shafquat.lashari@gmail.com

ABSTRACT

In the present study, very stable and converging Least Square finite element method (LSFEM) is employed to calculate the approximate solution of steady state Navier – Stokes equation, consisting continuity and momentum equations in two dimensions. The 2D rectangular domain is considered to study the behavior of linear fluid passing two parallel rectangular obstructions in an open channel. The current numerical analysis is based on two instances, in the first one two rectangular obstructions possesses sharp corners set to face the entering flow while in the other case, obstructions having round corners are analyzed concerning to examine the enhancement in the size of vortex formed due to the flow blockage, intensity of vortex and the recirculation flow rate in the dead zone for Reynolds number from 250 to 2000. The stream line patterns are also presented to monitor the changes appears in the shape of vortex for different Re. The information gather through the analysis suggests the cutoff corner obstruction is congenial in restricting the vortex length and favorable for slow down the flow circulation rate in the lower stream of the channel. The acquired results are compared with established data in past literature which turn out to be in good agreement.

Key words: Least – square FEM, Vortex length, rotational flow rate, rectangular obstruction, Reynolds number.

1. INTRODUCTION

The purpose of conducting a present numerical investigation over a rectangular obstruction having sharp and smooth corners surrounded by fluid in a rectangular domain is to explore the attributes of laminar flow within the range of Reynolds number 250 to 2000. Obstructions encircling by incompressible fluid have been a great topic of interest over years for their practical importance in many engineering and industrial applications, such as offshore structures, heat exchangers, power line bridge support, condensers, boilers, economizers, nuclear reactors and air condition coils etc. The study of past literature shows the researchers are enthusiastic over conducting various researches on the bluff body flow problems either by altering the shape of conventional or unconventional obstructions [1], [2] and sometimes introducing a modified numerical technique to produce novelty and to express several flow behaviors based on these alterations and modifications, this is because the flow characteristics, for instance; flow separation around the edges of obstruction, generation wake region, recirculation flow rate, enhancement in the vortex are somehow depends on the shape of bluff body [3]. In [4], Nasarudin et al. investigates the fluid flow around triangular and square cylinder by using finite volume method. The numerical simulation results showed the different flow patterns at different aspect ratios and the distance between cylinders creates vortex between cylinders which produced thicker boundary layer that results in increasing the drag coefficient. Kmiotek & Pietal in [5] investigates the flow around rectangular and triangular obstacles by using the ADINA package. They stated that due to imposed the rectangular and triangular obstacles the various bigger changes occurs in flow phenomena of the velocity field and also found some changes after comparison with the experimental results.

It is learned from literature review that obstruction possesses sharp edges builds a relatively larger vortex in the downstream than an obstacle owns smooth edges. In [6], researchers analyzed the effects of cutoff corners shaped body passing fluid in a channel, it is noticed during the simulation that corners cutoff shape produces considerable changes in the hydrodynamics features, it creates lesser drag force, generates smaller recirculation region and shows less enhancement in the size of vortex than a shape with right angle corners. Another research work [7] was carried out to explore the flow behavior of fluid passing two inline rectangular cylinders placed at different aspect ratios, the main purpose was to bring down the hydrodynamic forces on the obstacles and minimizes the ramifications of fluid inertia over the other facets of flow. It is noticed that the two inline cylinders are come across with reduction in the size of wake region in their downstream, slow down the flow circulation rate and efficiently turn down the drag coefficient to reasonable extent.

The current research is based on the comparison of laminar flow features around two parallel positioned obstructions in a rectangular flow field; first one is the pair of regular rectangular obstructions while the other pair of rectangular obstructions owns round corner. The schematic diagram of 2D geometry of the present work can be seen in fig.1, whose dimensions are defined on the height of rectangular obstruction, which is taken as ‘L’ m. Considering that the length of obstruction is 1.5L, distance between two parallel positioned obstructions is 2L, width of the flow channel is 8L while length of the channel is 20L which is divided in up and downstream as 4L and 16L respectively. Geometry design, triangular mesh and the numerical solution of governing equations of the flow field has been done using computational fluid dynamics (CFD) software COMSOL 4.0. The obtained results regarding to the intensity of construction of vortex, enhancement in the size of vortex and recirculation flow rate are presented as a function of Reynolds number from 250 to 2000. The streamline patterns are also available to closely visualize the gradual development in the flow features with increasing fluid inertia.

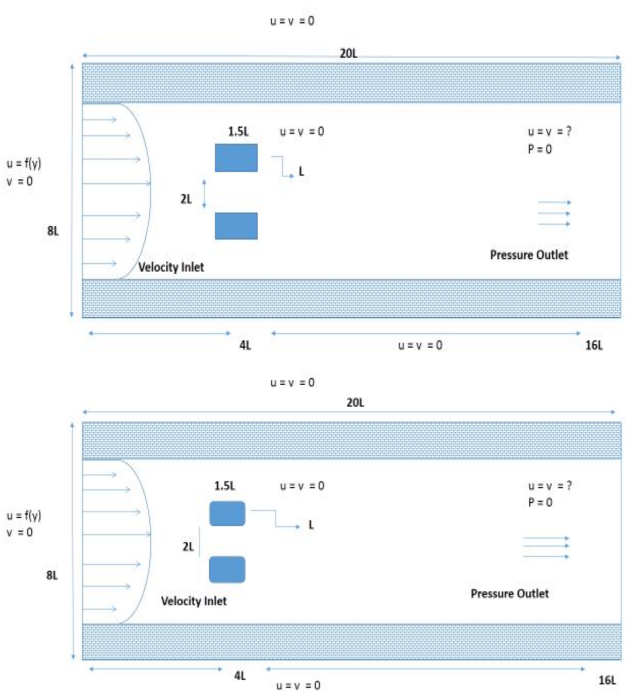


Figure – 1: Schematic ketch of the computational field around rectangular obstruction with sharp & round corners

2. BASIC GOVERNING EQUATIONS

The system of equations (1 - 2) that governs the under consideration problem of steady incompressible and Newtonian fluid with negligible forces are the fundamental equations of CFD known as continuity and Navier – Stokes equations in two dimensions with Cartesian coordinates.

$$\nabla . u = 0 \tag{01}$$

Where $\nabla = \frac{\partial u_1}{\partial x} + \frac{\partial u_2}{\partial y}$

$$\rho \frac{\partial u}{\partial t} - \eta \nabla^2 u + \rho (u.\nabla) u + \nabla p = 0 \tag{02}$$

2.1 Inlet & Outlet Conditions

This inlet and outlet conditions are quite essential to impose over appropriate boundaries in order to achieve realistic flow dynamics of simulation, can be seen in Table.1. The condition at the inlet is imposed as parabolic velocity while the upper and lower wall of the channel are considered to be static with no – slip conditions and pressure at the outlet of the channel is set to 0 pa.

Table. 1: The specified initial and boundary conditions for the computational domain

Conditions	Velocity	Pressure
Initial	$u\ (x, 0) = u_0$	$P(x,0) = P_o$
Boundary	1. At upper and lower wall: no-slip u = v = 0 2. At exit: axial velocity u is free and disappearance cross flow velocity (v = 0) 3. At the inlet $u = V_{mean} \times 6 \times y \times (1 - y)$	At exit P = 0

3. NUMERIAL SCHEME

The most important and newly developed least square Galerkin finite element approach is adopted for the quick convergence results of the steady state solutions of the governing equations comprising on the 2^{nd} order partial differential equations [8]. These partial differential equations are well known and most useful equations in the field of fluid dynamics called as continuity and momentum. The momentum equation is called as higher order due to the 2^{nd} order diffusion term that is very complex to solve directly. Therefore choose the extra variable as called as vorticity measured as curl of the velocity and this variable can easily convert the momentum equation into the system of first order partial differential equations, which will be solved by employing the high convergence rate Newton’s method. The research domain of present work is divided in to triangular elements to form an extremely fine mesh which can be seen in fig. (2 – 3) and the grid generation statistics are presented in the table. 2.

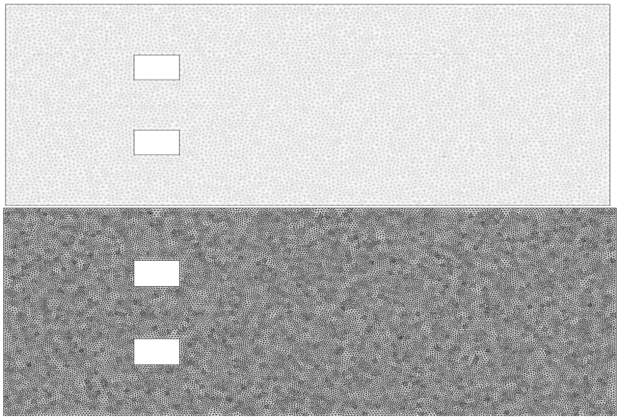


Figure – 2: Finite element fine and extra fine meshes of rectangular field consisting sharp corners rectangular obstruction.

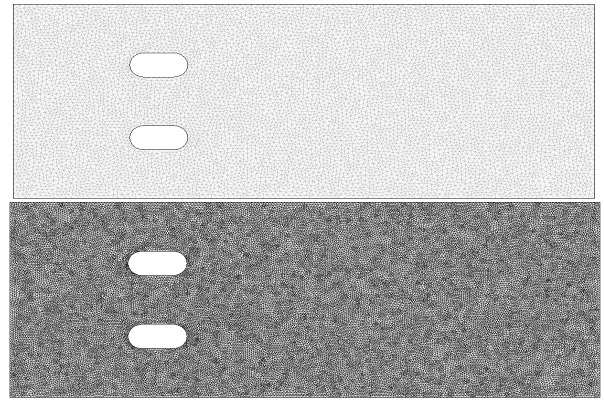


Figure – 3: Finite element fine and extra fine meshes of rectangular field consisting round corners rectangular obstruction.

Table. 2: Statistics of fine and extra fine grid of two dimensional research domain

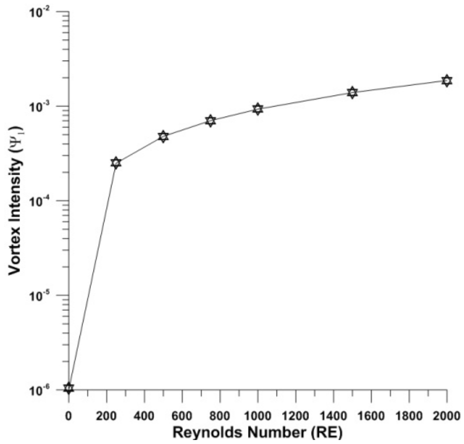
	Fine Mesh		**Extra Fine Mesh**	
Quantities	**Sharp Corner**	**Round Corner**	**Sharp Corner**	**Round Corner**
No of degree of freedom	76103	76459	302761	304085
No of mesh points	8529	8573	33787	33943
No of elements	16728	16796	66912	67184
No of boundary elements	332	352	664	704
No of vertex element	12	16	12	16
Min element quality	0.7026	0.6896	0.7026	0.6515
Element area ratio	0.1397	0.1255	0.1397	0.1255

4. EFFECT OF FLUID INERTIA ON THE RECTANGULAR OBSTACLE WITH SHARP AND ROUNDED CORNERS

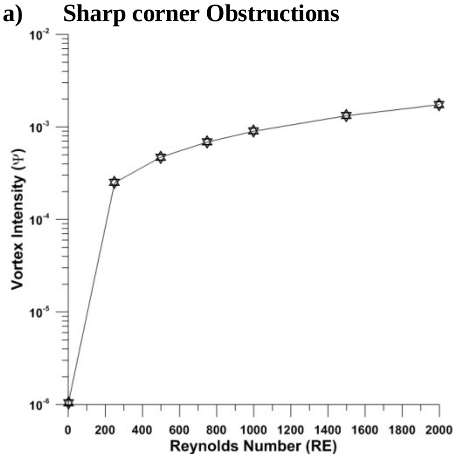
The flow characteristics are defined as a function of Reynolds number to analyze the effects of increasing fluid inertia over the obstructions in flow domain and their findings are presented to contribute in the research field of hydrodynamics. Since the study is comparison based therefore results of simulation are presented mutually of both kinds of obstructions. It is noticed that the flow around the obstructions behaves steadily in the downstream at initial value of Reynolds number at Re = 250. As the Reynolds number reaches to 750 the streamlines are started to revert in the downstream and a small region of wake is generated, this is due to the corners, whether sharp or rounded, causes flow to separate and formed vortex behind the obstruction but the vortex intensity and enhancement in the shape of vortex is clearly visible to observe the difference between two cases, [7]. The rounded corner body produces lesser vortex in the size and showed low intensity as compare to the case of obstruction with sharp edges when fluid inertia achieves its defined peak. The table.3 and fig.4(a – b) are presented to observe these flow behaviors statistically.

Table.3. Length of the vortex generated behind obstructions is presented in meters

Vortex Lengths of Rectangular Obstructions		
Reynolds Number	**Sharp corners**	**Rounded Corners**
250	0.068	0.020
500	0.128	0.068
750	0.182	0.119
1000	0.256	0.156
1500	0.385	0.233
2000	0.499	0.314



a) Sharp corner Obstructions



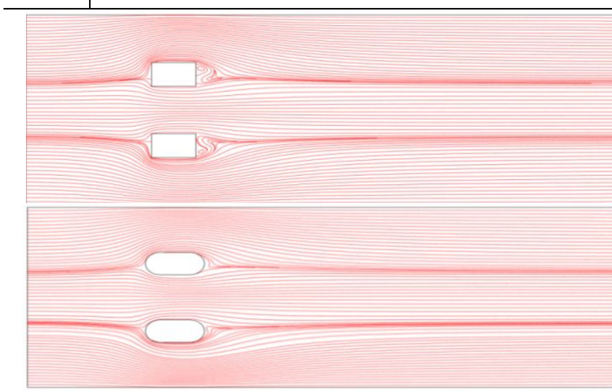
b) Round Corner Obstructions

Figure – 4(a – b): Graph of vortex intensity (Ψ_2) is a function of Reynolds number (Re) for Rectangular obstructions.

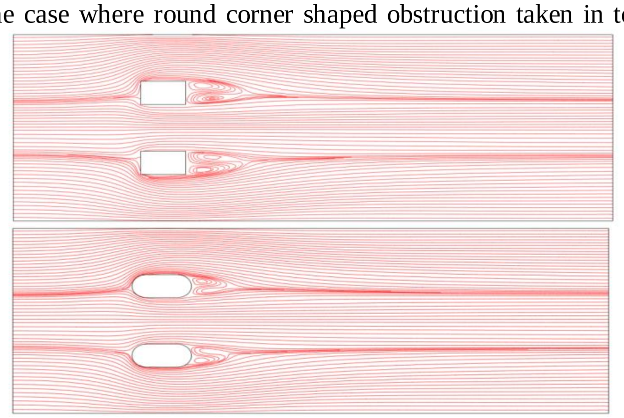
The streamline patterns colored by velocity vectors are presented in fig.5 in order to demonstrate the fluid flow phenomena in the computational domain to closely visualize impacts of inertia on the obstructions. The high inertia leads to construction of massive zone in the downstream of obstacle due to sharpness of its corners and ultimately gives higher rate value of recirculation flow while in contrast to the case where round corner shaped obstruction taken in to investigation, observed no activity of flow separation at Re = 250 and the flow is found to be laminar with the edges of obstruction, fair accordance is perceived from literature [6], the development in the vortex generation is observed at moderate inertia and produces smaller dead zone and low flow rate even at defined inertial peak, that is Re = 2000, this can be well understood from the table.4 that describes comparative values of rotational rate of flow over the range of fluid inertia.

Table.4. Flow rate of recirculation at different fluid inertia

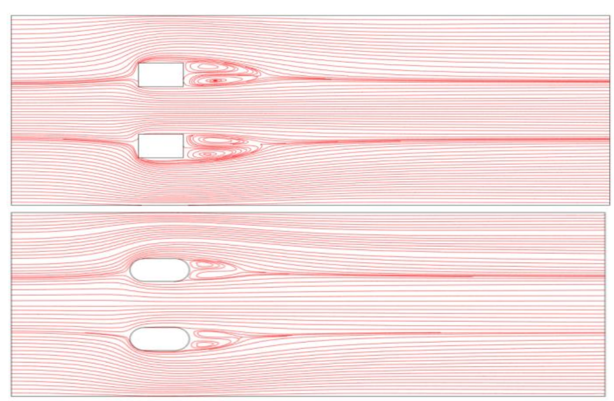
	Recirculation Flow Rate			
Re	**Sharp Corners**		**Rounded Obstacles**	
	Min	**Max**	**Min**	**Max**
250	6.25e-6	2.44e-4	6.25e-6	2.43e-4
500	1.21 e-5	4.69e-4	1.18e-5	4.61e-4
750	1.76 e-5	6.87e-4	1.72 e-5	6.71e-4
1000	2.32 e-5	9.08e-4	2.25 e-5	8.77e-4
1500	3.48e-5	1.36e-3	3.31 e-5	1.28e-3
2000	4.67 e-5	1.82e-3	4.35 e-5	1.72e-3



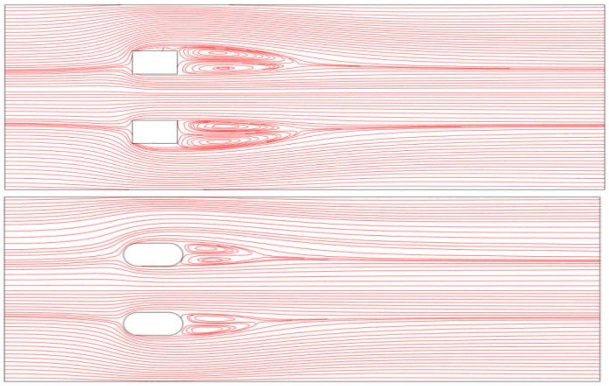
Re = 250



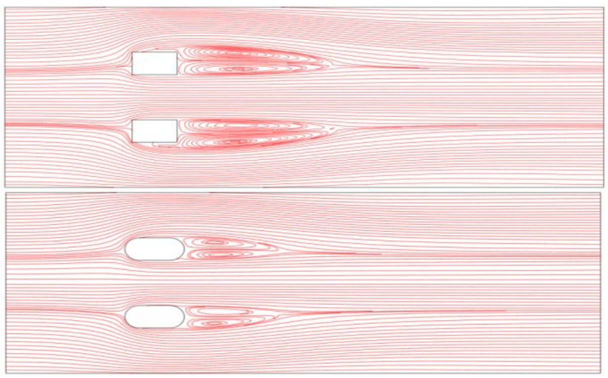
Re = 750



Re = 1000



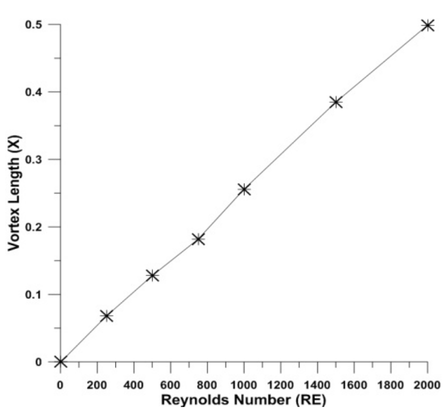
Re = 1500



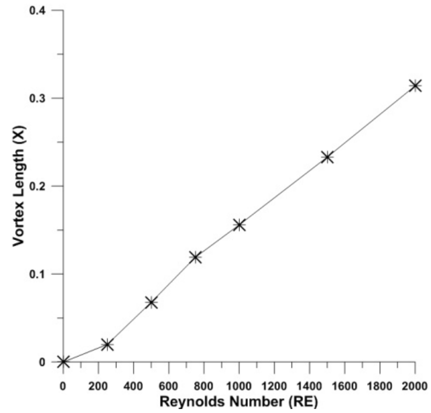
Re = 2000

Figure – 5: Streamline patterns for two–dimensional parallel positioned rectnangular obstruction in a channel from Re = 250 and 2000.

The enhancement in the wake area in two cases is the most considerable feature of the flow since the gradual increase in length of vortex is worth observing in comparison of both the studies. The graphs of vortex length depending on Reynolds number in fig.6(a – b) showed sudden jump at initial stage for the sharp corner oriented obstruction then gradually move up to 0.5m, whereas; round corner obstruction expresses thoroughly small amount of increment in vortex length. Though the rate of rotational flow is not significantly different in both studies but the overall results provided enough evidences to support our objective based on established data related to the current study.



a) Sharp Corner Obstruction



b) Round Corner Obstruction

Figure – 6 (a – b): Vortex Length (X_2) is a function of Reynolds number (RE) for rectangular obstructions**.**

CONCLUSION

The numerical model of steady incompressible and Newtonian fluid transportation through rectangular channel imposed fixed pair of rectangular obstacles with sharp and rounded corners planted at certain gap is develop and chosen to investigate in two dimensions. The governing equations of this problem, continuity and momentum equations written velocity – vorticity formulation are solved numerically by LESFEM to predict the flow behavior around the obstructions concerning to contribute in this problem which has potential relevance in many engineering fields. The results obtained through the recent study are presented as follow:

- Continuous growth in vortex length is detected in both the cases with increasing Reynolds number. Round corner obstruction displays 37% smaller vortex length as contrasted with sharp corner obstruction for Re = 2000.
- The sharp corner oriented pair of obstructions builds a massive wake region in the downstream at higher Reynolds number as compare to the pair of obstruction oriented by smooth corners, which is in good agreement with existing results.
- The flow rate of circulated fluid behind the rounded corner obstruction is slightly low when compared with sharp – edged obstruction.
- The current study is confined to laminar flow of Newtonian fluid only. One can conduct a numerical research on turbulent flow of Non – Newtonian fluid in order to express turbulent effects on pair of obstructions.
- The obstructions in this work are in parallel position. In future, the study can be revisit by placing the pair of obstruction at different angles of attack to explore unrevealed features of flow.

REFERENCES

1. N. SALAM, I. WARDANA, S. WAHYUDI & D. WIDHIYANURIYAWAN (2014) “Fluid Flow through Triangular and Square Cylinders” Australian Journal of Basic and Applied sciences, 8(2) February 2014 Pages:193-200. https://core.ac.uk/download/pdf/25495743.pdf
2. FORBES & SCHWARTZ, (1982), “Free-surface flow over semi-circular obstruction”. Journal of fluid mechanics, volume 114. January 1982. Pp. 299 – 314 DOI: https://doi.org/10.1017/S0022112082000160
3. K. MUHAMMAD AND W. A AISSA (2016) “Effect of Corner Modification on Two-Dimensional Turbulent Flow Around a Square Cylinder With Incidence” .Mech. Power Dept., Faculty of Energy Engineering, Aswan University, Aswan, Egypt Journal of Engineering Sciences Assiut University Faculty of Engineering. Vol. 44 No. 1 January 2016 PP. 91 – 102 DOI: 10.21608/jesaun.2016.117589
4. NASARUDDIN SALAM, I.N.G.WARDANA, SLAMET WAHYUDI, DENNY WIDHIYANURIYAWAN (2014) “Flluid flow through triangular and square cylinders” Australian journal of basic and applied sciences, 8(2) February 2014, ISSN 1991 – 8178 http://www.ajbasweb.com/old/ajbas/2014/February/193-200.pdf
5. MAŁGORZATA KMIOTEK & ANNA KUCABA-PIĘTAL (2010), “Preliminary Calculations of Flow in Channel with Triangular and Rectangular Obstacle”. Task Quarterly 14 No 4, 329–337 https://task.gda.pl/files/quart/TQ2010/04/tq414c-e.pdf
6. 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
7. PRATISH P. PATIL & 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, I ssue 3, 2009. https://www.researchgate.net/publication/236009516_Numerical_Investigation_of_Laminar_Unsteady_Wakes_Behind_Two_Inline_Square_Cylinders_Confined_in_a_Channel
8. CHING L. CHANG (2002), “Analysis of the L^2 least-squares finite element method for the velocity vorticity pressure Stokes equations with velocity boundary conditions” Applied Mathematics and Computation130, 121–144**.** DEVISA.J (2011) “The Finite Element Method Second Edition.” DOI: 10.1016/S0096-3003(01)00086-8
9. H. MEHDI, V. NAMDEV, P. KUMAR & A. TYAGI, (2016) “Numerical analysis of Fluid flow around a circular cylinder at low Reynolds number” IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) e-ISSN: 2278-1684,p-ISSN: 2320-334X DOI: 10.9790/1684-13030294101
10. A. A MAHESSAR, Z. A. K BALOCH, A. L. QURESHI, A. BALOCH (2015), “Transient Analysis of Two–Dimensional Free Surface Flow through Hydraulic Structure”. Sindh University Research Journal, Vol. 47 (4), 713716 https://www.academia.edu/44098070/Transient_Analysis_of_Two_Dimensional_Free_Surface_Flow_Through_Hydraul ic_Structure
11. M. SALINAS, W. VICENTE, E. BARRERA, & E. MARTINEZ, (2014) “Numerical analysis of the drag force of the flow in a square cylinder with a flat plate in front” Instituto de Ingenieria, Universidad Nacional Autonoma de Mexico, Ciudad Universitaria, 04510 Mexico D.F., Mexico. REVISTA MEXICANA DE F´ISICA 60 (2014) 102–108 https://www.researchgate.net/publication/287290543_Numerical_analysis_of_the_drag_force_of_the_flow_in_a_square_cylinder_with_a_flat_plate_in_front
12. NIDHUL. K (2014) “The Influence of Corner Geometry on The Flow Structure and Flow Characteristics for Flow Past a Square Cylinder at Re = 150”. International Journal of Research in Aeronautical And Mechanical Engineering, Volume 2 issue 12 pgs 32 – 41Issn (Online): 2321-3051 **https://www.academia.edu/10099923/INFLUENCE_OF_CORNER_GEOMETRY_ON_THE_FLOW_STRUCTURE_AND_FLOW_CHARACTERISTICS_FOR_FLOW_PAST_A_SQUARE_CYLINDER_AT_Re_150**