STUDY OF NUMERICAL SIMULATION OF ARRAY OF STAGGERED CYLINDER IN DIFFERENT REYNOLDS NO. REGIME

Capstone Project-II Report submitted in

Completed successfully requirements for the award of the

Degree of

BACHELOR OF TECHNOLOGY

Submitted by

ABHISHEK KUMAR

(18SCME1010030)

ADIL HUSAIN

(18SCME1010010)

IN

MECHANICAL ENGINEERING

DEPARTMENT OF MECHANICAL ENGINEERING

Under the Supervision of

MR. FAISAL SHAMEEM

MAY 2022

GALGOTIAS UNIVERSITY

(Established under Galgotias University Uttar Pradesh Act No. 14 of 2011)

GALGOTIAS UNIVERSITY

DEPARTMENT OF MECHANICAL ENGINEERING

BONAFIDE CERTIFICATE

Certified that this project report **"STUDY OF NUMERICAL SIMULATION OF ARRAY OF STAGGERED CYLINDER IN DIFFERENT REYNOLDS NO. REGIME"**is the bonafide work of "**ABHISHEK KUMAR (18SCME1010030),ADIL HUSAIN(18SCME1010010)"** who carried out the project work under my supervision.

SIGNATURE OF DEAN

SIGNATURE OF SUPERVISOR

APPROVAL SHEET

This thesis/dissertation/project report entitled titled **STUDY OF NUMERICAL SIMULATION OF ARRAY OF STAGGERED CYLINDER IN DIFFERENT REYNOLDS NO. REGIME** by **Abhishek Kumar-(18SCME1010030), Adil Husain-(18SCME1010010),** is acceptable for the bachelor of technology in mechanical engineering degree.

Examiners

____

____

____

Supervisor

____

Dean

____

Date: _____________

Place: _____________

Statement of Project Report Preparation

1. Project report title:**"**STUDY OF NUMERICAL SIMULATION OF ARRAY OF STAGGERED CYLINDER IN DIFFERENT REYNOLDS NO. REGIME"**.**

2. The degree at which proposal is delivered:BACHELOR DEGREE OF TECHNOLOGY.

3. The report was determined by the Project Supervisor.

4.The thesis format specifications were strictly adhered.

5.The contents of the thesis have been prepared in accordance with provision

6.The report was written without the use of plagiarism.

7.All sources utilised have been carefully attributed.

8.The report has not been accepted for a degree elsewhere.

(Signature of the student)

Name:(ABHISHEK KUMAR)

Add. No: (18SCME1010030)

Statement of Project Report Preparation

1. Project report title: **“**STUDY OF NUMERICAL SIMULATION OF ARRAY OF STAGGERED CYLINDER IN DIFFRENT REYNOLD NO. REGIME”**.**
2. The degree at which proposal is delivered: BACHELOR DEGREE OF TECHNOLOGY.
3. The report was determined by the Project Supervisor.
4. The thesis format specifications were strictly adhered.
5. The contents of the thesis have been prepared in accordance with the provisions.
6. The report was written without the use of plagiarism.
7. All sources utilised have been carefully attributed.
8. The report has not been accepted for a degree elsewhere.

(Signature of the student)

Name: (ADIL HUSAIN)

Add.No.(18SCME1010010)

ACKNOWLEDGEMENT

I have worked hard on this project. However, it would not have been possible without the kind help and support of many individuals and organizations. I'd want to express my heartfelt gratitude to each and every one of them. I am really grateful to **Dr. P.K.S. NAIN (DEAN S.O.M.E., G.U.)** & **MR. FAISAL SHAMEEM (S.O.M.E., G.U.)** for their direction and ongoing monitoring, as well as for supplying vital project help and advice in accomplishing the work I'd want to show my appreciation to a member of **GALGOTIAS UNIVERSITY** for their kind guidance and support in completing this project.

My grateful thanks also go to my colleague who supported me in designing the project and to those who agreed to support me with their skills.

Abhishek Kumar (18SCME1010030)

Adil Hussain(18SCME1010010)

(School of Mechanical engineering)

ABSTRACT

The fluid flow past cylinder is still investigating understand the complex phenomenon involved .There are several application in the area of structural and mechanical design where cylinder in the The configuration is side-by-side. The flow across staggered cylinders is numerically simulated in this research at various variables of placement and Re No.s. The impacts of On the motion, the gap proportion and the Re No.s parameters will be explored. Examples include immediate Vortices speeds, velocity factors, or vortex shedding shapes A combined influence of Reynolds number and gap spacing(0.5 thebehind the cylinders, the length of the wake changes in this computational investigation. Some physical factors that are important in practise are also examined.The fluid flow past cylinder is still being investigated to understand the complex phenomenon involved. There are several Application in the area of structural and mechanical design where cylinder in the side -by -side ordering are being used. this paper, we need to simulate flow over cylinder are performed at different positioning ratio and Reynolds number. The impacts of the positioning ratio and Reynolds number on the pass feature, such as immediate vorticity contours, force coefficients, and regularity throw out of vortex, have been investigated. Using computational flow dynamics, the flow of fluid near two circular staggered cylinders of same size was examined (CFD). The incidence angle changed from 0° to 90° but the fraction of centre pitch remained fixed. During the simulation, the Re No.s was employed.

TABLE OF CONTENT

Chapter No.	Title	Page No.
	Certificate	
	Approval sheet	**3**
	Student declaration (Student 1)	**4**
	Student declaration (Student 2)	**5**
	ACKNOLEDGEMENT	**6**
	Abstract	**7**
	Table of content	**8**
	List of abbreviation	**9**
	List of figures	**10**
	List of tables	**10**
1.	**Introduction**	**11**
2.	**Literature Review**	**12**
	2.1 MATERIALS AND METHODDS	
3.	**Methodology and Formulation**	**18**
	3.1 SCHEMATIC OF STAGGERED CYLINDERS	19
	3.2 GAP SPACINGS AND Re No.s AS A FACTOR IN FLOW REGIMES	
	3.3 EXPLAINATION OF THE PROBLEM AND NUMERICAL INFORMATION	
	3.4 ANALYSIS OF GRID SENSITIVITY, DOMAIN SENSITIVITY, AND VALIDATION	
	3.5 STEADY TO UNSTABLE FLOW TRANSITION	
4.	**Analysis of model at different gap ratios**	**22**
	4.1 Model at Gap Ratios =0.5	
	4.2 Model at Gap Ratios= 1	
	4.3Model at Gap Ratios=3	
5.	**Result and Conclusion**	**28**
6.	Conclusion	29
7.	**References**	**30**

List of abbreviations

Serial no.	symbol	term
1.	Ly	Ly is the computation property's length, which was modified by varying the gap between the cylinders.
2.	Re	Reynold number
3.	g	Gap between cylinder
4.	U	Inlet velocity
5.	x	x indicates for flow speed dimensions.
6.	Y	Y indicates for transversal axis
7.	μ	Viscosity
8.	ϱ	Density
9.	α	Angle of incidence
10.	Ćd	Coefficient of drag
11.	Ćl	Coefficient of lift

List of figures

Figure	Title	Page No.
1.	Schematic of staggered row of cylinders	4
2.	Model at Gap Ratios 0.5	14
3.	Model at Gap Ratios 1	15
4.	Model at Gap Ratios 3	15

List of tables

Table No.	Title	Page No.
1.	Boundary conditions used in fluent 19.2	9

CHAPTER 1

INTRODUCTION

Several marine and ocean engineering applications need the analysis of unstable viscous flow across an array of circular cylinders. It's harder to forecast flow around a multicylinder layout using geometric factors like the cylinder spacing ratio than it is for a single cylinder. The huge dynamic interplay In the wake flow, shearing planes shed vortex and Karman vortex.makes flow behaviour behind multicylinders substantially more difficult. One assembly's wake interacts with the wake of another, forming streams with high Reynolds numbers.Surprisingly, this natural stream is pulsatile and consistently has a low Re number. In bio liquid mechanics, a low Reynolds number stream in a pulsatile channel climate is critical. The absolute artificial lung (TAL), a device that operates in the Re range and is made up of a series of micro-sized cylinders through which air and blood are combined, has recently sparked a lot of interest.In order In order to have a better understanding of the flow patterns in the area complicated as well as enormous structures, the flow across many airfoil and its complex dynamics must be explored. Interesting flow patterns for passing through staggered rows have been found in this setting.These flow patterns are determined by the entering flow and the cylinder gaps spacings (g) but they also ensure that the wake contact process occurs air foils with many slots, and flow through large construction blocks are among the engineering applications investigated in this work. The fundamental goal of this research is to better understand how complicated multiple structures generate and interact with wakes.Non-dimensional factors of practical relevance include Re No.s (Re = Ud/), as well as gap spacings (g = s/d). The distance between the cylinders' surfaces.is denoted by s, the cylinder size is denoted by dis, the uniform input velocity is denoted by U, and the fluid's kinematic viscosity For simulations, square shape was chosen because it can reduce the amount of errors associated with cylinder wake. Analysis of unsteady streamline flow past annular cylinder arrangement is a significant field for various Oceans and offshore engineering field applications measured to a single solitary annular. This is also attempting to predict flow through many cylinder configurations based on geometrical stipulations such as spacing ratio between the annular. The intricate powerful relationship between the shearing layers shed eddy and eddy street in the flow of water makes the flow behaviour behind several cylinders considerably more difficult. These streams have a higher Reynolds number, and the wake of one chamber interacts with the wake of another. This natural stream is pulsatile and low in Re number In bio liquid mechanics, a low Reynolds number stream in a pulsatile channel climate is critical. An absolute artificial lung (TAL), which is a device that works in Re 1-20 and consists of a series of micro-sized cylinders across which air and blood are mixed, has recently gained attention. Two fluid-dynamically interacting cylinders may be thought of as the basic constituent of various constructions, and knowing this flow can help you grasp what's going on around them. As a result, the flow between two cylinders has gotten a lot of attention in the literature. For almost a century, engineers and fundamental-fluid-dynamics researchers have been studying cross-flow through circular cylinders in various designs. This is observed that this type of flow can be found in a variety of engineering fields, including heat exchangers, offshore constructions, and transmission lines. When fluid flows through circular cylinders, boundary-layer separation, shear-layer development, and vortex mechanics are all examples of fundamental fluid-dynamic processes. A asymmetrical configuration is the most basic form of a pair of cylinders in a fluid flow. Despite this, the overall staggered design has received less attention than the simpler parallel and side-by-side arrangements.

CHAPTER 2

LITERATURE REVIEW

Oyewole et al. [1] performed a numerical simulation of time-dependent 2-D forced convection flow through the pair of parallel circular cylinders in a rectangle channel. The author of this study employed cosmol Multiphysics, a finite-element based software. The spacing ratios, according to the author, change the dynamics of the flow. Furthermore, as S/D grows, the temperature difference between the cylinders decreases dramatically.

The FEA method based ANSYS fluent software was used by Dwivedi et al. [2] to Examine the fluid flow and combined convective heat transfer of an incompressible fluid throughout identical cylinders organised in a restricted tandem arrangement. The researcher discovered that there is a huge shift in physical properties after a particular gap ratio, and that the transformation is continuous beyond that gap ratio..

Necati et al. [3] investigate the precarious laminar convective heat transfer from two constant temperature cylinders arranged in tandem. The mathematical simulations have been carried out using FLUENT. The study is conducted for the Reynolds numbers of 100 and 200, as well as the central range ratios of 2, 3, 4, 5, 7, and 10. For $L/D > 4$, the upstream cylinder's average Nusselt reaches that of an only one constant temperature cylinder, as well as the downstream cylinder's average Nusselt number is somewhere around 80% of the upstream cylinder.

Sun et al.[4] investigatedVibrations caused by flow and heat generated by forced convection transmit of 2 parallel circular cylinders in laminar flow with $Re=150$ and $Pr=0.7$ using a fluid interaction simulation. In the calculated range of parameters, the time average of a separated cylinder subjected to circulation vibration is reached up to 15%. The upstream cylinder in the two-cylinder model needs to perform correspondingly to a separated cylinder in aspects of reaction and heat transmitting, but the downstream cylinder starts behaving particularly different.

Neslihan et al. [5] calculated the heat and flow conditions of a laminar flow passing through two tandem circular cylinders in a canal. The effect of the friction factor and the space between both the cylinders on heat and flow features was studied. The findings demonstrate that as the friction factor and gap size rise, the detachment angular position on the downstream cylinder declines. The detachment angle on its front cylinder is independent of changes in the distance between two cylinders. When the barrier ratio increases, the drag coefficients for both cylinders increase significantly, as long as front cylinder's drag coefficient is lower than the back cylinder.

Moshkin et al. [6] used computational models to explore laminar 2-D heat transfer from two rotating circular cylinders in flow path of incompressible viscous fluid under constant temperature boundary conditions. The analysis is focusing on numerically solving the entire mass, momentum, and energy conservation equations. They found that when the distance between both the cylinders is greater beyond one diameter, the rate of energy transfer decreases as the cylinders' rotational speed is increased. When Pr increased, the ordinary Nu increased significantly. The thermal contour behaviour and the streamline contour structure are very comparable.

Sohankar et al. [7] conduct a numerical investigation of laminar heat transfer from two inline square cylinders with active flow conditions. A finite-volume algorithm which is based on a coordinated grid system is being used in the 2-D numerical simulations. Such flow control systems are being used to reduce moment and fluctuations forces while also preventing vortices. To regulate the flow, the cylinder's and front return surfaces are subjected to continuous blowing, while an end sides are subjected to constant suction. To regulate the flow, the cylinder's front and back areas are subjected to considerable blowing, while the shoulder sides are exposed to constant suction. RC, UFC, and RC flow control specifications are evaluated.

Singh et al.[8] analyzed numerically the several configurations of a cylinder wherein the upstream cylinder shifts at various angles with regard to the down - stream cylinder in a fixed location at a lag time ratio. The vortex removing forming is researched when the vortices from the wind speed cylinder communicate with the vortices from the downwind cylinder and generate remarkable flow patterns. The drag and lift coefficients for both cylinders increase as the angle location of the cylinders changes. The impact of air flow canister on downwind cylinder is powerful for =30, 45, and 60 direction circumstances, and vortex - induced based on participants. Excluding the downwind cylinder, the local Nu varies little between both the cylinders.

Chatterjee et al.[9] studied the impacts of Reynold number and P numbers on fluid flow properties in a tandem configuration of two equivalent constant temperature square cylinders in flow path. The length between the cylinders is ascertained by the widths of the cylinders. Numerical simulations are performed in a clustered power grid using a finite volume program based on the PISO method. The drag coefficient was found to be highly sensitive to Pr but increases with Re. The Nu is raised by both the Pr as well as the Re. The drag and thermal transfer increase as the restriction grows.

Chatterjeeet et al. [10] conducted experimental research. A 2-D numerical analysis of flowing fluid and forced convective heat transfer coefficient all over two equal constant emperature square cylinders placed through tandem as well as subjected to a Newtonian fluid with a reynolds Numbers flow path. The mathematical results are carried out using a PISO automated process finite volume solver in a clustered power grid. And they found that as the Reynolds number inside the selected limits rises, so does crucial separation distance. A local minima for the Nusselt number is seen near critical separation distance ratio. The Nu tends to increase as that of the Re rises both for cylinders and separation distance ratios.

Prasad et al.[11] did a computational analysis to investigate the steady and dynamic laminar heat transfer characteristics in a stream with two square bars arranged beside each other. The investigation is carried out for transversal gap distance between both the bars at distinct obstruction ratios. The total drag coefficient decreases as the Reynolds number rises. The average Nusselt number is determined to have climbed by more than 76 percent. The biggest congestion of constant temperature is detected on the front area, implying the greatest Nusselt number.

Zhang et al [12] carried out a numerical research of the convection of flow via two tandem cylinder with curved edges in a channel. It looks at how two important characteristics, the gap ratio and radius of curvature, affect the flow unpredictability and thermal expansion capabilities of a hitherto unstudied tandem configuration. The gap flow between the two cylinders is classified into two regimes: constant at lower GR and unsteady at high GR. The flow across tandem cylinders may be completely stabilised in the constant gap stream flow by round the edges. By rounding the corners, the amplitude of the local peak of the time-averaged Nusselt number is lowered, while localized heat transmission at the leading static pressure is enhanced.

Using a greater discontinuity Galerkin technique, Shana et al.[13] examined the development of the fluid flow in the gaps and close wake of a series tandem cylinder alternately in the interval (AG) domain. The transient characteristics of the flows, vorticity, and pressure regions, the transient circular pressure distribution, and the fluid velocity along the centre - line of the wake are explored at a Reynolds number of 200 and a pitching ratio of 2.3. The results show that a gap-flow forms when the gap-flow meets with nonlinear static vortices in the AG domain and the adjacent tandem cylinder in the AG domain gap, leading the split boundary layers to reattach independently or bilateral. In the AG domain, the actual mechanism of boundary layers reattachment and gap-flow creation.

Monfared et al. [14] studied the thermal lattice Boltzmann system and the refute heat energy concentration technique practically. The issue of constant and unsettled mixed convection across linear tandem square cylinders with such a constant heat flow at the blocked border is examined for the first time. The rise in mRi causes a rise in the drag coefficient, particularly inside the downstream stages. At intermediate Reynolds numbers, increasing causes a modest rise in dropping frequency, but the influence is muted at greater Re. Nusselt is enhanced by Reynold ’s number in the same way as drag coefficient does. In the AG domain, the actual mechanism of boundary layers reattachment and gap-flow creation.

For a fixed Reynolds number, Ahmed et al.[15] numerically analysed under the influence of the spacing ratio, the flow around two parallel square cylinders and splitter plate length. Single-bluff body, shear layers reattachment, steady, completely formed two-row vortex shedding, and fully developed flow regimes were the five different flow regimes observed. When g/D was changed from 0.5 to 4, the CDmean2 was negative. The change of flow regime from shear layers' reattachment flow regime to fully developed flow regime (g/D= 4.5) causes a discontinuity in CDrms2 without splitter plate for g/D=4.5.

Incompressible unstable viscous two-dimensional finite volume was researched by Harichandanet al.[16]. Unstructured mesh with collocated nodes consisting of triangular cells, a Navier–Stokes solver is created utilising the "consistent flux reconstruction" technique. For Reynolds numbers (Re) = 100 and 200, the solver was applied to unconfined flow past a single cylinder, two cylinders, and three cylinders. A numerical technique for solving two-dimensional Navier–Stokes equations for incompressible viscous flows has been suggested. For unstructured triangular meshes, the solver uses a cell-centered finite-volume technique.

For flow past a tandem cylinder array, the flow structures and heat transfer were computationally examined by Hsu et al. [17], as well as an example of influence of a slit on heat transfer amplification. To investigate the impact on the flow pattern and heat transmission, a variety of cylinder distances and slit inclination angles were simulated.. In a tandem array, different distances between the cylinders cause one of three flow patterns: reattachment, unstable reattachment, or coshedding. The flow is in reattachment mode when the distance ratio is less than the critical distance. The flow is in co-shedding mode when the distance ratio is greater than the critical distance ratio.

Danman[18] explored the impact of two circular cross-sectional barriers situated in a rectangular channel parallel to the heated surface and normal to the flow direction on heat transfer and friction characteristics. FLUENT, a commercial CFD software, was used to model the numerical calculations. Heat transfer is influenced by the flow regime, which can be modified by the Reynolds number. The Reynolds numbers increase the overall amount of heat transfer from the barriers. By the way, the research reveals that as the Reynolds number rises, the drag coefficient and skin friction coefficient drop.

Kanna et al. [19]. Invistigated flow through two isothermal offset square cylinders in a channel is simulated for variable offset distance (OD) and Reynolds number (Re) L=0.6 Re 150, L=1.5d Flow offset was done over two isothermal square cylinders organised in a confined channel for a two-dimensional incompressible fluid in the laminar region.

Chaitanya et al [20] conducted a comparison between two comparable circular cylinders to see how rotation affects flow characteristics in an unconfined medium at low pressures. To investigate the Reynolds Number, two-dimensional numerical simulations are utilised. 3.0 g 0.2 g Prandtl number (Pr=0.71), Reynold number (Re=100); Prandtl number (Pr=0.71), Reynold number (Re=100); Prandtl number (Pr=0.71), Prandt The cylinders that are upstream and downstream spin in opposite directions.

The study of flow through staggered rows of square cylinders has not been addressed, according to the explanation above. Because it represents the extreme case of an isolated cylinder, the rows of staggered cylinders are of basic importance. Furthermore, researchers have used numerical models to investigate flow transitions, although their knowledge of the flow field is restricted. By adjusting Re and g, we shall analyse the change in flow regimes. Kumar et al.7 found that for a given Reynolds number, various flow patterns exist. This motivated us to look into the impacts of Re having a range of 0.5 g 3 on the flow regimes.The critical Re as well as the resulting flow regimes are discovered. Changes in the wake structure and the merger of emerging jets have traditionally been examined separately in the literature, but our goal is to move beyond these perspectives. We specifically examine how the jets created between the cylinders have a substantial impact on the dynamics of wake contact. Furthermore, this research will undoubtedly broaden our understanding of fluid-structure interactions.

Material and methods:

The governing equations for this project are the momentum equation, continuity equation, and energy equation. Flow through a pair of cylinders that are misaligned with varied angles was originally simulated using Ansys Fluent 19.2 software. The model's capacity was also tested at higher Reynolds numbers. The geometrical information for the 2-D geometrical model of circular cylinders was acquired from the experimental findings, which we used to justify the mathematical model. The computational domain, details of boundary The viscous-laminar and incompressible flow were addressed, as well as their conditions and governing fluid flow equations. It depicts the three separate examples that were solved in the current study, being the same in all of them. To acquire appropriate submerged In the streamwise direction, near the cylinder, boundary points on the cylinder's surface, the Cartesian grid, and a refined and uniform mesh were used.

CHAPTER 3

METHODOLOGY AND FORMULATION

As illustrated in Figure 3.1, we studied theunstable two-dimensional flow of an incompressible fluid around a circular cylinder (with uniform velocity (U) and temperature (T) at the input). he surface temperature of the cylinder is kept constant. The streaming fluid's thermo-physical characteristics are unaffected by The effects of temperature and viscous dissipation likewise ignored. The temperature difference between the cylinder's surface and the flowing liquid is believed to be small, therefore temperature variations in physical qualities, such as density and viscosity, can be ignored. We model this problem by limiting the obstruction in a box with fictitious borders because it is an unconfined flow.We select the box's size such that the flow pattern and temperature profile are unaffected. The flow and heat transport processes in the aforementioned issue are regulated by continuity, momentum, and thermal energy equations. The geometry and meshing are done in ANSYS 15, while the simulation is done in FLUENT.

THE PROBLEM'S BOUNDARY CONDITIONS:

There is a fluid flow of condition at the inlet border.

$U_y = 0$ and $U_x = U$ T_∞=300K

On the circular cylinder, no-slip flow is taken into account:

$U_x = 0$ and $U_y = 0$. T_w=310KReynolds number:

$Re=\varrho Ud/\mu$

Where: ϱ=Density

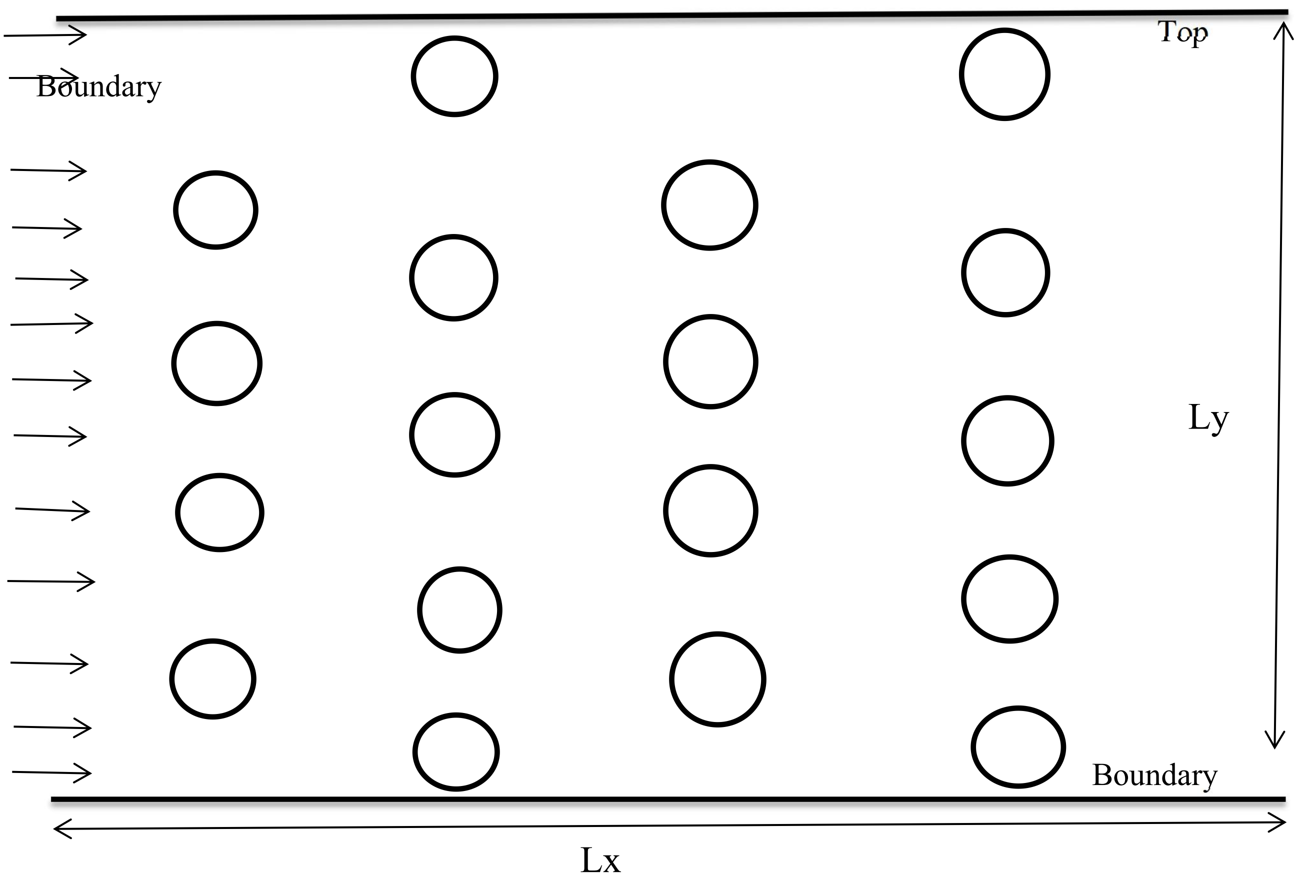
μ Viscosity

U=Inlet velocity

d=Diameter

3.1 SCHEMATIC OF STAGGERED ROWS OF CYLINDERS :

This section will focus to is organized around the problem statement and mathematical data. Flowing through square cylinders in asymmetrical rows of comparable size is seen in Figure 1. In the first, second, and third columns, there are four, five, and six cylinders, respectively. At the computational domain's intake, uniform velocity (u = U, v = 0) is utilized, which has relatively low compressibility effects. We used a convective boundary condition at the outflow to keep the fluid from travelling backward.The momentum exchange approach is used to compute the fluid forces. The flow for the indicated problem is at a stop at the start of the simulation. as well as the engineer and paper applications.



FIG(1)

The graph of the geometry and mesh of the flow passing across the array of circular cylinders is shown in Figure 1. The computation area is 80D (D bore annular) and the origin (0, 0) is the annular positioning matrix's central place. The computation area's input is 40 degrees upstream and 40 degrees downstream of the starting location. The upper and lower boundaries are 40D from the origin of the coordinates, as determined by an equality constraint of and a mathematical model of flow across an array of circular cylinders. The present velocity gradient is 0, and the lateral speed is the zero constraint, according to the manual definition of the equality constraint used in 2.For Re = ten and twenty, stratified mimicry was utilized, as well as the dimensionless tensor variants of the Stokes equation. The movement of an incompressible viscous liquid is stated as ui, where is a two-dimensional velocity, p as pressure, t as time, and is the scale of Sir Joshua Reynolds. If the Reynolds number is less than 20, the simulation may be done immediately.The trial, a finite volume approach used to avoid calculations in the computational domain. The most widely studied Simple algorithmic programme that handles the mating of two fields, which may be important in getting a move on equation convergence and must be accurate throughout computations. Regarding the discretization of convection under pressure conditions within saving, Lam et al. proposed a second-order draught differencing concept. In comparison to the first-order upwind theme, the second-order upwind differentiation scheme was chosen because of its great stability and precision. to minimize the time it takes for convergence for the time, an implied second-order direct discretization was applied. Because the continuity equation is more difficult to converge than other equations, a standard for convergence has been established that is 10-4 for the continuity equation and 10-5 for other calculations.

3.2 GAP SPACINGS AND REYNOLDS NUMBER AS A FACTORIN FLOW REGIMES:

In this part, we'll look at how Re affects flow regimes using Flow field patterns, CD and CL time-trace studies, and CL energy spectrum studies To prevent duplication, a few popular examples will be described.This research separates distinct flow regimes depending on the nature of the the fluid flow and the originating nozzles between the cylinders.

3.3 EXPLAINATION OF THE PROBLEM AND NUMERICAL INFORMATION:

This part is built around the problem statement and associated mathematical data. The flow through misalignment rows of square cylinders of equal dimensions that are uniformly spaced and subjected to a uniform flow with a starting velocity U is depicted in Fig. 1. In the first, second, and third columns, there are four, five, and six cylinders, respectively. Lu and Ld stand for upstream and downstream, respectively. The surface-to-surface distance's' between the cylinders determines the channel length, Lx. For variable g, the computational domain is provided.

3.4 ANALYSIS OF GRID SENSITIVITY, DOMAIN SENSITIVITY, AND VALIDATION:

For a solitary square cylinder, an unstable flow is recreated in the Re range in order to use it as a reference for more advanced numerical studies for staggered structures.For the simulation, Lx Ly is used as the computational domain. and Ld = are the inlet and outflow boundaries, respectively. The cylinder lies at the middle of the channel, with a height of Ly . A uniform inflow velocity (u = U, v = 0) and a convective boundary condition are specified at the inlet and outflow borders, respectively.The no-slip boundary condition is applied to the cylinder surface as well as the channel's lateral edges. The fluid forces acting on the cylinder are calculated using the momentum-exchange method. The current code is tested for different grid points   around the surface of a square cylinder to guarantee grid sensitivity.

3.5 STEADY TO UNSTABLE FLOW TRANSITION:

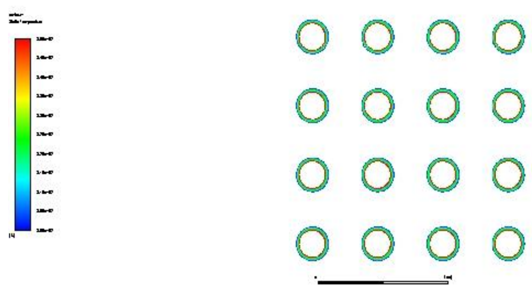
A crucial Re No.s (Re) at which the transition from steady to unsteady flow occurs is defined in detail in this section. Re is chosen between by raising Re by 0.5 increments acquired through systematic study. The Re grows as g increases. At g the maximum value of Re . As a result, we established the lowest limit of Re to be   for future inquiry. the flow shifts from two-dimensional to three-dimensional for an isolated square cylinder between , hence the top limit of Re has been fixed . Therefore, in step of with $.5 \leq g \leq 3$ The range covered for further investigation in step one.

Boundary conditions used in fluent 19.2:

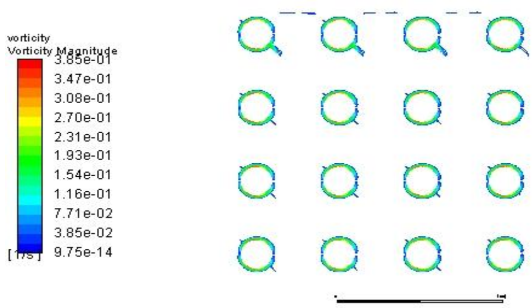
Boundary	Assigned
Entrance	Inlet speed
Exit	Pressure exit
Cylinder located upstream	Wall
Cylinder located downstream	Wall
Bottom	Symmetry
Topside	Symmetry

4 . ANALYSIS OF MODEL AT DIFFERENT GAP RATIO:

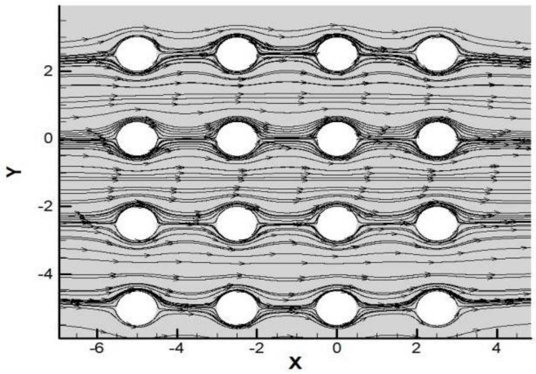
(1).MODEL AT GAP RATIO=0.5



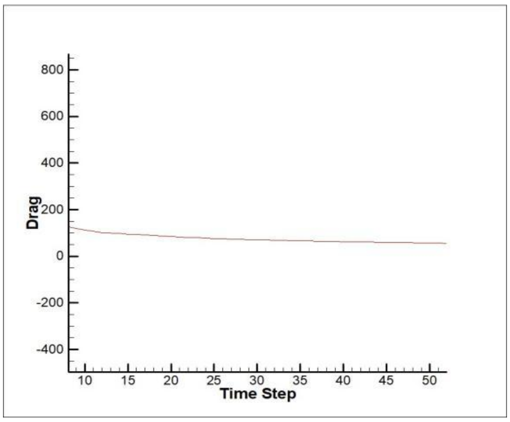
FIG(2).Temperature Contour



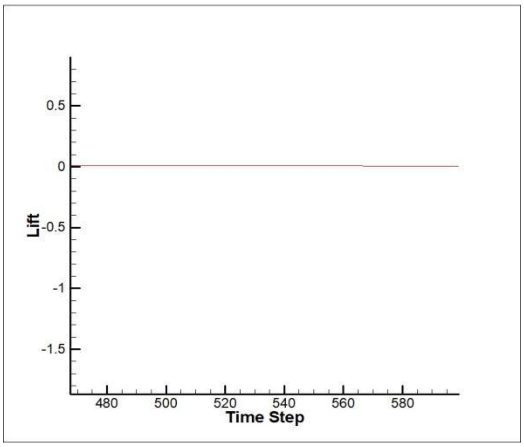
FIG(3).Vorticity Contour



FIG(4). Streamline Contour

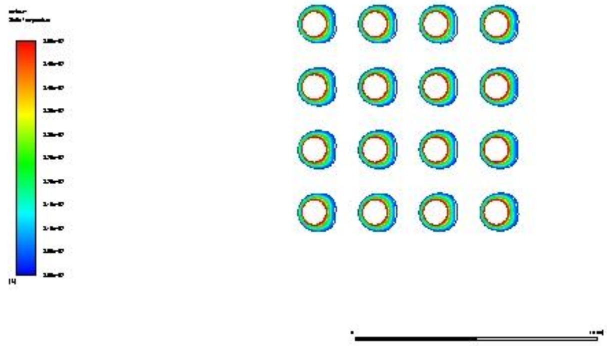


FIG(5).Coefficient of Drag Plot

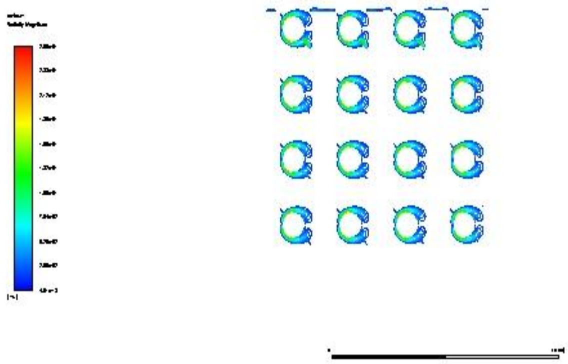


FIG(6).Coefficient of Lift Plot

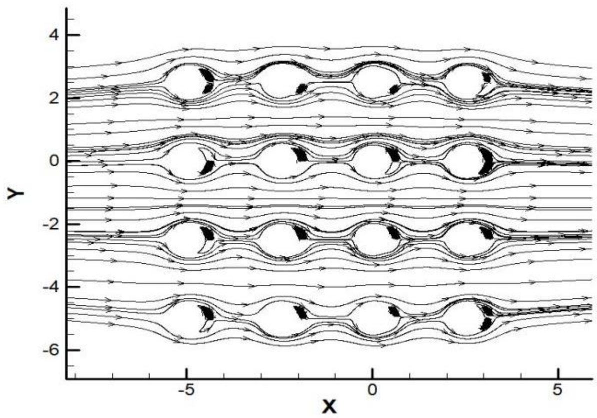
(2).MODEL AT GAP RATIO= 1



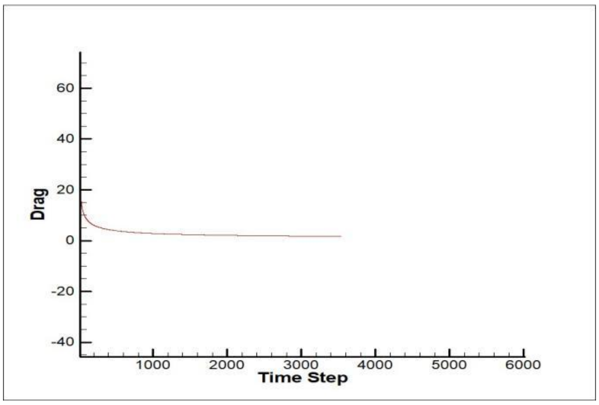
FIG(7). Temperature Contour



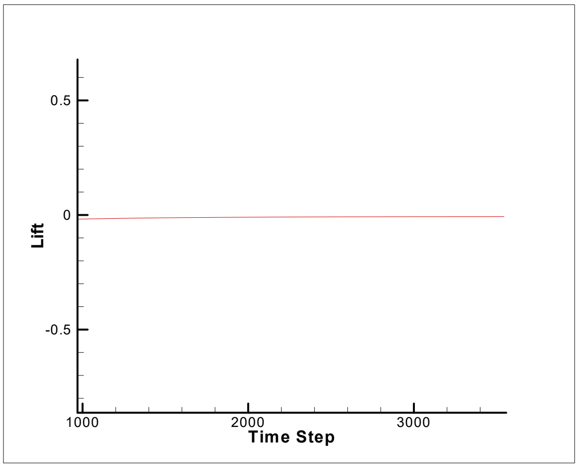
FIG(8). Vorticity Contour



FIG(9). Streamline Contour

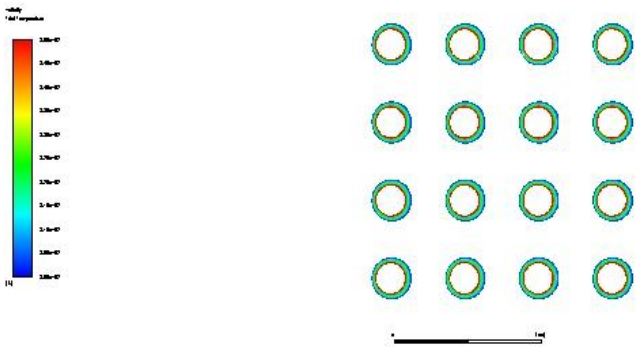


FIG(10). Coefficient of Drag Plot

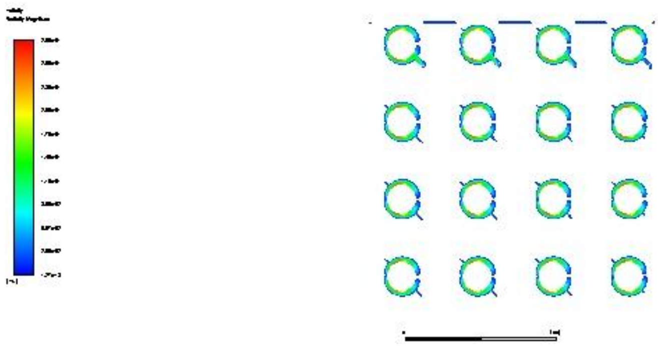


FIG(11). Coefficient of Lift Plot

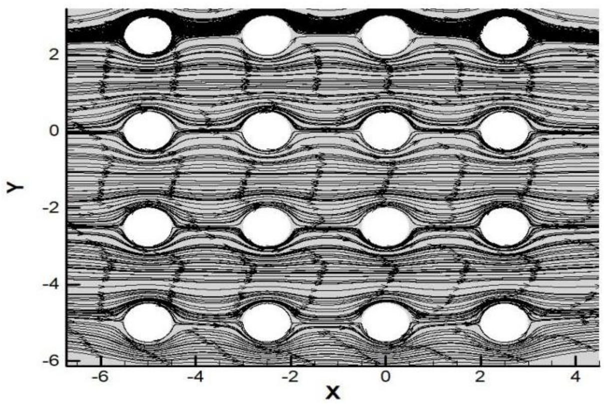
(3).MODEL AT GAP RATIO=3



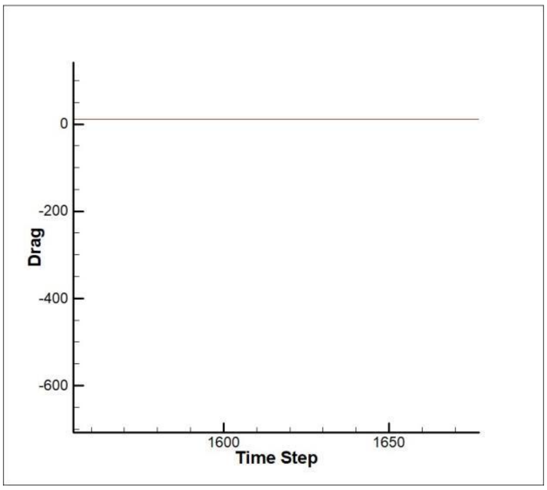
FIG(12). Temperature Contour



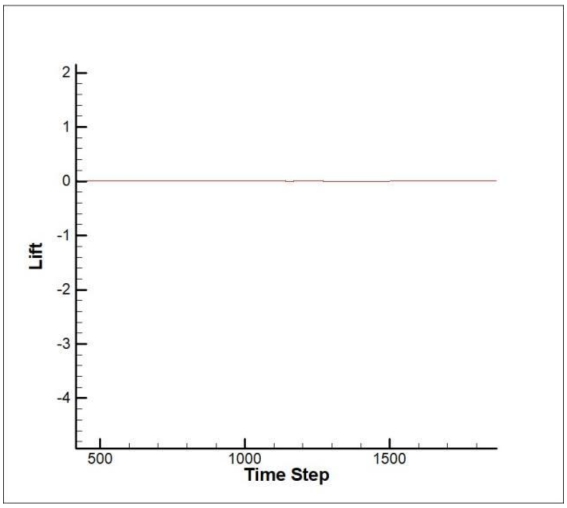
FIG(13). Vorticity Contour



FIG(14).Streamline Contour



FIG(15). Coefficient of Drag Plot

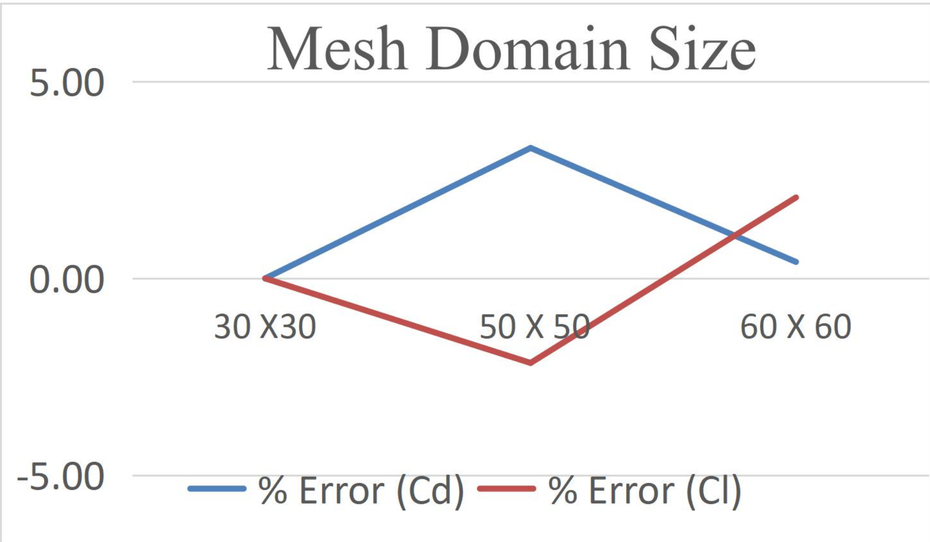


FIG(16).Coefficient of Lift Plot

CHAPTER 5

RESULT AND DISCUSSION

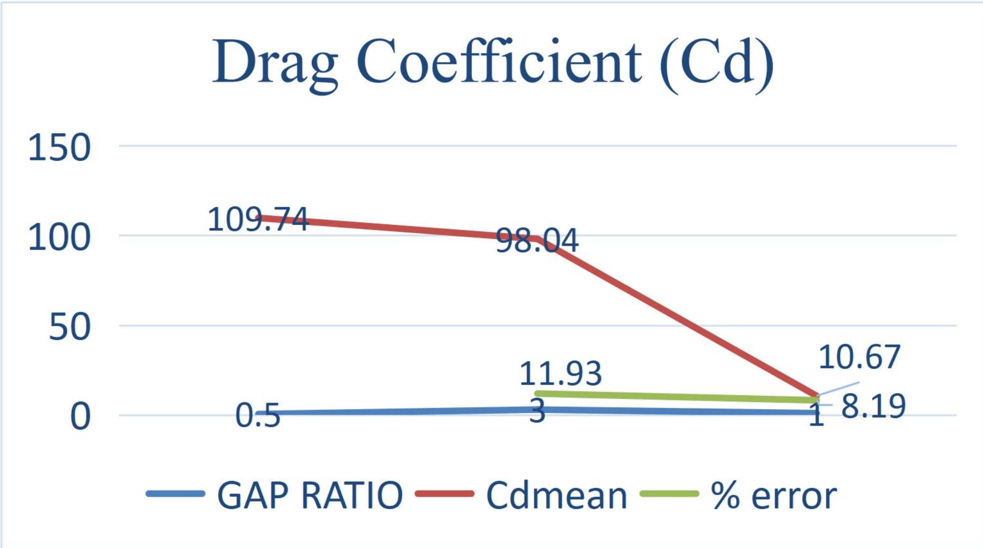
(5.1)Comparison of % error in coefficient of drag and lift at different gap ratios



FIG(17).

Due to less % error from moving to 50X50 mesh domain size to 60X60, we prefer the adoption of 60X60 mesh domain size for the simulation of 16-circular cylinder array Heat Exchanger, leads to efficient working.

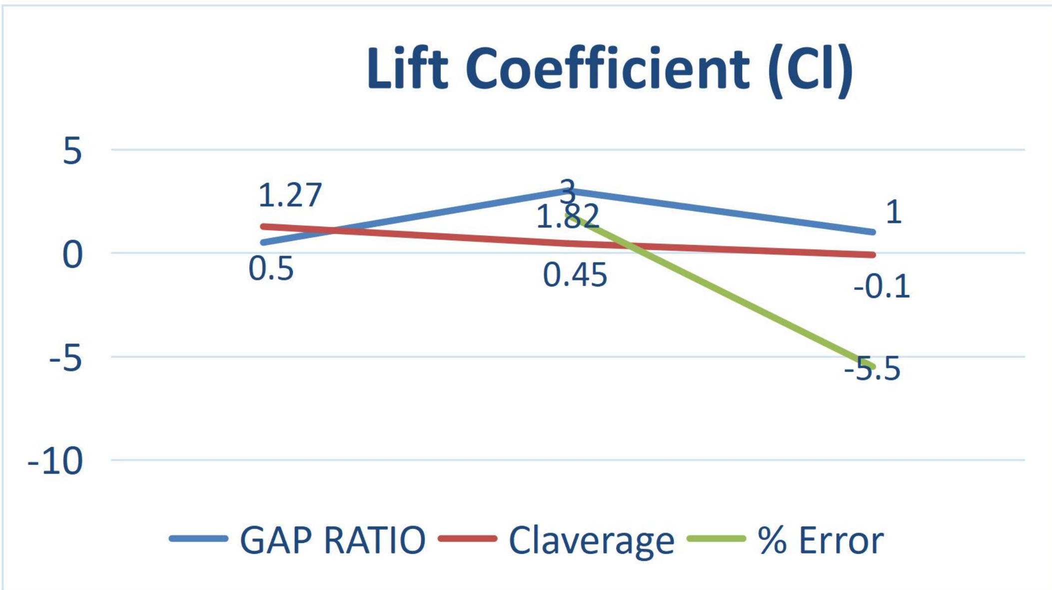
(5.2) Comparison of % error in coefficient of drag at different gap ratios.



FIG(18).

Due to less %error in drag coefficient at gap ratio 1, we prefer the adoption of gap ratio 1 between 16-circular cylinder array Heat Exchanger for the simulation, leads to better efficiency

(5.3) Comparison of % error in coefficient of lift at different gap ratios.



FIG(19).

Due to less %error in lift coefficient at gap ratio 1, we prefer the adoption of gap ratio 1 between 16-circular cylinder array Heat Exchanger for the simulation, leads to better efficiency.

(6).Conclusion:

From the analysis of this study, the following conclusions have been drawn.

- Heat transfer will increase as the coefficient of drag and lift decreases.
- Using large domain size around the array of 16-circular cylinder increases the efficiency of heat transfer in the heat exchanger.
- Using 0.88≤gap ratio ≤1 around the array of 16-circular cylinder, % error in coefficient of drag and lift decreases.
- HEs work well in terms of lift and drag coefficient at gap ratio i.e. (≡1m)

Based on this analysis it can be concluded that Heat Exchangers can work efficiently at Re = 100 with unity gap ratio around the array of 16-circular cylinder of diameter equals to 1, and domain size 60m.

Reference:

[1] Oyewala, "Numerical simulation of forced convection flows over a pair of circulars in tandam arrangement," *Jordan Journal of mechanical and industrial engineering,* october 2019.

[2] A. R. dwivedi, "Flow heat transfer analysis around tandem cylinder : critical gap ratio and thermal cross buoyancy," *Journal of the Brazilian society of mechanical sciennces and Engineering ,* september 2019.

[3] N. Mahi_r, "Numerical investigation of convective heat transfer in unsteady flow past two cylinder in tandem arrangements," *International Journal of Heat and fluid flow ,* may 2008 .

[4] S. L. Xu Sun, "Effects of flow induced vibration on forced convection heat transfer from two tandem circular cylinders in laminar flow .," *International Journal of Mechanical Science ,* dec 2020.

[5] N. AYDIN, "Numerical investigation of heat and flow characteristics in a Laminar flow past two tandem cylinders," *Mechanical Engineering , Uludag University , Bursa,16059,Turkey,* April 2021.

[6] N. moshkin*, "Numerical simulation of flow and forced convection heat transfer in cross flow of incompressible fluid over two rotating circulars," *Institute of science , School of Mathematics Suranaree University of Technology, Nakhon Ratchasima , Thailand.*

[7] M. K. A.sohankar, "Control of flow and heat transfer over two inline square cylinders," *Department of Mechanicla Engineering , Isfahan University of Technology , Isfahan, Iran Department of Mechanical Engineering Yazd University Yazd , Iran .*

[8] S. Vikram, "A study of flow interference and heat transfer between two-cylinder at different orientations," *Journal of Ocean Engineering and Science,* January 2021.

[9] D. Chatterjee, "The Effect of reynolds and prandtl number on flow and heat transfer across tandem square cylinders in the steady flow regime," *An International Journal of computation and Methodology,* March 2011 .

[10] D. Chatterjee, "Forced convection heat transfer from tandem square cylinders for Various spacing Ratios," *Journal of computation and Methodology ,* Feb 2012.

[11] A. V. durga, "CFD Analysis of momentum and heat transfer around a pair of square cylinders in side-by-side arrangement," *Department of chemical Engineering , Indian Institute of Technology Roorkee , Roorkeee , india ,* sep 2013 .

[12] W. Zhang, "Forced convection for flow across two tandem cylinders with rounded corners in a channel," *Internaltional Journal of Heat and Mass transfer ,* october 2018 .

[13] X. Shana, "Evolution of the flow structure in the Gap and Near Wake of two tandem Cylinders in the AG Regime," December1 2020.

[14] A. E. F. Monfared, "Thermal Flux simulations by lattice Boltzman method investigation of high Richardson number cross flows over tandem square cylinders," *International Journal of Heat and mass transfer,* march 2015.

[15] a. w. Ali ahmad, "Flow characteristic and fluid forces Reduction of Flow past two tandem cylinder in presence of Attached splitter plate," *Hindawi Mathematical Problem in Engineering ,* september 2021.

[16] A. b. harichandan, "Numerical investigation on of low reynolds number flow past two and three circular cylinders using unstructure grid CFR scheme," *International Journal of Heat and fluid flow .*

[17] L.-c. hsu, "Heat transfer in flow past two cylinder in tandem and enhancement with a slit," *Multidisciplinary Digital Publishing Institute/2021 .*

[18] Danigman, "Heat transfer and fluid flow analysis with CFD and HD Modal around isothermal two tandem circular cylinders," *january 2009 .*

[19] P. R. kanna, "Numerical Simulation of steady Flow and forced convection Heat transfer from two off set square cylinder placed in a channel," june 2015.

[20] N. K. Chaitanya, "Influence counter Rotation on Fluid Flow and Heat Transfer around Tandom circular cylinders at low Reynolds number," *Journal of the brazalian Society of mechanical science of engineering ,* 2021 .