Numerical Investigation Of Flow Around Double, Single And No Slit Cylinder

Author

HASSAAN IDREES

Registration Number

329505

Supervisor

Dr. Muhammad Sajid

DEPARTMENT OF MECHANICAL ENGINEERING

SCHOOL OF MECHANICAL & MANUFACTURING ENGINEERING

NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY

ISLAMABAD

MAY 2023

Numerical Investigation of flow around double, single and no slit cylinder

Author

HASSAAN IDREES

Registration Number

329505

A thesis submitted in partial fulfillment of the requirements for the degree of

MS Mechanical Engineering

Thesis Supervisor:

Dr. Muhammad Sajid

Thesis Supervisor's Signature: ____

DEPARTMENT OF MECHANICAL ENGINEERING

SCHOOL OF MECHANICAL & MANUFACTURING ENGINEERING

NATIONAL UNIVERSITY OF SCIENCES AND TECHNOLOGY, ISLAMABAD

MAY, 2023

FORM TH-4

MASTER THESIS WORK

We hereby recommend that the dissertation prepared under our supervision by: Hassaan Idrees, Titled: Numerical Investigation of flow around double, single and no slit cylinder be accepted in partial fulfillment of the requirements for the award of Masters degree.

Examination Committee Members

1. Name: Dr. Sara Ali Signature:________________

2. Name: Dr. Waqas Khalid Signature:________________

Supervisor's name: Dr. Muhammad Sajid Signature: ________________

Date:____________________

____
Head of Department

____
Date

COUNTERSINGED

____
Dean/Principal

Date:___________

Thesis Acceptance Certificate

Certified that final copy of MS thesis written by **<u>Hassaan Idrees</u>** Registration No. **<u>329505</u>**, of SMME has been vetted by undersigned, found complete in all respects as per NUST Statutes / Regulations, is free of plagiarism, errors, and mistakes and is accepted as partial fulfillment for award of MS degree. It is further certified that necessary amendments as pointed out by GEC members of the scholar have also been incorporated in the said thesis.

Signature: ______________________________

Name of Supervisor: Dr. Muhammad Sajid

Date: ___________________________________

Signature (HOD): ________________________

Date: ___________________________________

Signature (Principal): ____________________

Date: ___________________________________

Certificate for Plagiarism

It is certified that MS Thesis Titled: Numerical Investigation of flow around double, single and no slit cylinder by Hassaan Idrees (329505) has been examined by me. I undertake the follows:

a. Thesis has significant new work/knowledge as compared already published or are under consideration to be published elsewhere. No sentence, equation, diagram, table, paragraph, or section has been copied verbatim from previous work unless it is placed under quotation marks and duly referenced.
b. The work presented is original and own work of the author (i.e., there is no plagiarism). No ideas, processes, results, or words of others have been presented as Author own work.
c. There is no fabrication of data or results which have been compiled/analyzed.
d. There is no falsification by manipulating research materials, equipment, or processes, or changing or omitting data or results such that the research is not accurately represented in the research record.
e. The thesis has been checked using TURNITIN (copy of originality report attached) and found within limits as per HEC plagiarism Policy and instructions issued from time to time.

Name & Signature of Supervisor

Name: Dr. Muhammad Sajid

Signature: ________________

Declaration

I certify that this research work titled "*Numerical Investigation of flow around double, single and no slit cylinder*" is my own work. The work has not been presented elsewhere for assessment. The material that has been used from other sources it has been properly acknowledged / referred.

Signature of Student

Hassaan Idrees

2023-NUST -Mech-000329505

Copyright Statement

Acknowledgements

I am thankful to my Creator Allah Subhana-Watala to have guided me throughout this work at every step and for every new thought which You setup in my mind to improve it. Indeed I could have done nothing without Your priceless help and guidance. Whosoever helped me throughout the course of my thesis, whether my parents or any other individual was Your will, so indeed none be worthy of praise but You.

I am incredibly grateful for the support and guidance I have received throughout my thesis journey, and I would like to take this opportunity to express my heartfelt appreciation to those who have contributed to my success. Firstly, I would like to thank my supervisor, Dr. Muhammad Sajid, for his invaluable guidance, mentorship, and unwavering support throughout my research. Dr. Sajid not only taught me the fundamentals of basic and applied CFD but also helped me develop problem-solving skills that have been critical to my academic and professional growth. His encouragement and faith in my abilities have been a constant source of motivation and inspiration, and I am deeply indebted to him for his guidance and mentorship.

I would also like to express my gratitude to my parents, who have always been my pillars of strength and support. Their unconditional love, guidance, and unwavering belief in me have been the driving force behind my academic achievements. Without their encouragement and unwavering support, I would not have been able to achieve my academic goals, and for that, I will always be grateful.

I would like to extend my sincere appreciation to my colleague and teacher, Engr. Bushra Rasheed, for attending to my queries and patiently bearing my many questions throughout the research process. Her insights and feedback have been invaluable in shaping my research, and I am thankful for her support and guidance. Finally, I would like to thank my team IOSE and my friends Engr. Mateen Ahmad and Engr. Shehzad Ahmad, who have supported and motivated me throughout my research journey. Their encouragement and positivity have made the long hours and hard work worthwhile, and I am grateful for their friendship and support.

Dedicated to my Parents

Abstract

This study presents a numerical investigation of the flow characteristics around a single and double-slotted cylinder using computational fluid dynamics (CFD). The simulations were conducted using CFD software OpenFOAM on a structured grid. The governing equations for incompressible, laminar flow were solved using the SIMPLE FOAM solver. A mesh-independent study led to a structured grid with 0.7m cells which was sufficient to capture the flow features accurately. The results show that the addition of double slits to the cylinder significantly reduces the drag coefficient as compared to a normal cylinder. The reduction in drag coefficient was observed to be more significant for wider slits, with a maximum reduction of 22% observed for a gap between slits and a width-to-diameter ratio of 0.1. The streamlines and pressure contours for the double-slotted cylinder show a distinct change in the flow pattern due to the presence of the slits. The flow around the slotted cylinder is observed to be more streamlined with less vortex shedding as compared to the no-slit cylinder. Additionally, the slits help in reducing the size of the wake region behind the cylinder, resulting in a reduction of pressure drag. The effect of the Reynolds number on the flow characteristics was also studied. It was observed that the increase in Reynolds number resulted in a decrease in drag coefficient for each set of cases, similar to that of a no-slit and single-slotted cylinder. The results of this study suggest that the presence of double slits can significantly alter the flow characteristics of a cylinder and improve its aerodynamic performance, especially in laminar conditions. The findings of this study can have potential applications in various fields, including aerospace, wind energy, and marine engineering.

Key Words: *Slotted Cylinders, Drag Coefficient, Passive Control, Flow characteristics, Force Characteristics*

Table of Contents

List of Figures

List of Tables

CHAPTER 1: INTRODUCTION

Extensive engineering research has been focused on the flow around cylinders because of its potential to improve performance in various applications, including aerospace, hydrodynamics, power generation, and heat exchangers. The complex flow dynamics of these structures are governed by changing geometry and operating conditions, making them an intriguing subject for scientists and engineers. The knowledge gained from this research is applicable to improving the design of tall buildings and underwater structures, as there are similarities in the physical characteristics of the flow around cylinders and these structures.

1.1 Background and Motivation

Flow over a bluff body is a topic of interest among the researchers pursuing CFD related studies. Flow over cylinder being an example to flow over a bluff body is extensively studied. Unfortunately, being a topic of vast practical presence, there is minimum literature that can help us predict flow over the cylinder with multiple slits. So, the topic being novel as no previous work has been done was chosen.

In addition, Computational Fluid Dynamics (CFD) is an important area not only in research but of vast practical applications. Unfortunately to perform numerical and computational solutions for complex problems requires licensing of commercial solvers such as ANSYS, COMSOL etc. This restricts researchers from pursuing this field as licensing can cost them a fortune. This project aims to conduct a study based on open-source software such as OpenFoam. This would include the development of code specific to this topic, which can be openly used improved based on the problem. It would help the researchers and developers in future to benefit from an already existing literature and code and could improve it for a vast number of practical uses. Apart from a research perspective, the results could be applied to various engineering systems around the country and aerodynamic improvements could be made.

1.2 Objectives of the Research

The objectives of this research are as follows:

1. To study the reduction of drag in a multiple slotted cylinder.

2. To evaluate the effect of the gap between slits and the slits-to-diameter ratio on the drag of the cylinder.
3. To identify the optimal combination of slits-to-diameter ratio and gap between slits for double slotted cylinders.

1.3 Scope and Applications of this research

Flow over a bluff body is conducted for several purposes such as to,

i. Reduce the drag force

ii. Study the flow-induced vibrations

iii. Control the lift force

iv. Enhance Heat Transfer

All these factors are well understood and researched so that flow-induced problems may be reduced, and an engineering design can be optimized, resulting in considerable improvements in public safety/comfort.

Flow around bluff bodies is of important practical significance in various fields such as

i. Aerodynamic Forces of Cooling Towers and Stack.

ii. Dispersion around high-rise buildings.

iii. Forces on underwater structures.

iv. Flow in a Heat Exchanger

v. Electronic Cooling

CHAPTER 2: LITERATURE REVIEW

2.1. Overview

Flow around cylinders has been the focus of extensive research in engineering due to its potential to enhance performance in various engineering applications such as power generation, aerospace, hydrodynamics, and heat exchangers. The flow dynamics around these structures are complex and governed by changing geometry and operating conditions, making them a fascinating subject for engineers and scientists [1]. The research is used in improving engineering designs of skyscrapers [2] and underwater structures [3], [4] due to the similarity of the physical characteristics between flow around cylinder and buildings. An important physical parameter for these structures is the cylindrical drag force which is related to the vortex shedding at the wake region of the cylinder. The Vortex-Induced Vibrations (VIVs) resulting from the drag have the potential to cause significant structural damage in engineering systems with cylindrical cross-sections, such as wired bridges and overpasses, towers, onshore and offshore structures, and heat-dissipating devices [5]. The risk accompanying the VIVs induced by drag has led researchers to find active and passive techniques to reduce or eliminate them. The objective of developing such techniques is to reduce drag and prevent variations in the lift by altering the way a bluff body sheds its wake vortex.

2.2 Types of drag control methods

The two main methods to control drag coefficient in flow around cylinder are, Active control and Passive control methods.

2.2.1 Active Control Methods

Active control methods previously developed include suction control [6], synthetic jet [7], rotating cylinders [8], heated cylinders [9], and electromagnetic force control [10], [11]. These methods required an external energy source to control the process and eventually alter the drag-around cylinders. Koutmos et al.[12] used both computational and experimental methods of planer jet injection at Reynolds number (Re) of 8400 and discovered that at an injection rate of 0.75, the jet does not separate the wake zone which results in four circulation sections it's downstream. Close to the wake region, a square cylinder experiences two wakes due to the jet's separation of the wake zone at injection rate 1.54. Huai-long et al.[13] approximated a flow field around a slotted cylinder at Reynolds Number (Re) 200. Constant suction and blowing at the slit orifice acted as a disturbance, the study found, decreasing the time it took for the boundary layer flow to divide along the back surface of the slotted cylinder and speeding up the time it took for the vortex to form. The effect of Re, slits' location and inclination in a slotted cylinder were studied by Zhu et al [14]. using PIV experiments. According to the results of the experiments, the boundary layer may be strongly impacted by blowing and suction if the slit vent is in front of the flow separation point for the base cylinder.

2.2.2 Passive Control Method

A few examples of passive control techniques discussed by Choi et al. [15] are dimpled surfaces, surface protrusion, grooves, the slotted cylinder, and splitter plates which work without an external energy source and are usually modifications to the cylinder surface.

For square cylinders researchers have used various geometric configurations for drag reduction, Aydin et al. [16] used modified square cylinders, i.e., square cylinders with holes and resulted that the holes in the cylinders stretched the vortex formation. He et al. [17] used square cylinders with rounded edges and reported that with increase of the round cut, reduction in drag coefficient was observed. A square cylinder with varied angle of orientation was tried by Deepak et al. [18] and concluded that the maximum drag reduction was observed at 45 degrees tilt angle. For circular cylinder without slits, Wu et al. [19] carried out simulations with a hinge plate and resulted that the addition of the hinge plate resulted in decrease in drag coefficient. Donglai et al. [20] used a circular cylinder with splitter plate reported the splitter plates decreased the drag coefficient by 51.5 %. 3D flow around cylinder with various aspect ratios was investigated by Prashant et al. [21] and reported three flow regimes with decreased drag coefficient. and Shi and Feng [22] modified the normal cylinder with narrow cuts at upper and lower stagnation points. This modification resulted in reduction of the vortices and decrease in drag coefficient.

In slotted cylinders, parallel and inclined slits were primarily used in the literature, as the parameters to reduce the drag coefficient. Tahir Durhasan [23] experimentally investigated slotted square cylinders at Re 6000 and concluded that the self-issuing jet due to the slot resulted in reduction of drag coefficient by 42%. Hangan and Kim [24] also used slotted square cylinders and reported 10% drag reduction due to penetration of near wake region and decrease in pressure fluctuations. Circular slotted cylinders with parallel slits are equally famous among researchers [25]–[28] and results shows that the increase in the width of the slits decreases the drag coefficient and suppress the vortex shedding. Alshareef et al. [29] explored the fluid dynamic performance of slotted cylinders and concluded that the due to the addition of slots in the the drag coefficient decreased by 45%. The slit's effectiveness was over low speed laminar flows was explored by Goodarzi and Asadi [30] at Re 150 and reported that the slits induced additional stagnation points and decreased the drag coefficient by 5%. Verma et al. [31]and Sharma et al. [32], [33] used parallel converging and diverging slits for flow around cylinder for Re 500 and 60-180 respectively. It was concluded that the modifications in the cylinder led to a decreased pressure drag and low Strouhal number.

Slits orientation in flow around cylinder are also prevalent in literature to improve flow aerodynamics. Peng et al. [34] and Hsu et al. [35] used slits perpendicular to the flow and reported that normal slits result in increase in drag. The later performed simulations on Re 100 to 500 and reported that increase in the slits width resulted in the increase in the drag coefficient. Bao et al. [36] studied inclined slits and found that for a slit's width of 0.3 a decrease of 28% is observed, while Olsen and Rajagopalan [37] used a combination of six cylinders with slits and resulted in reduced drag and improved aerodynamics parameters.

Table. 1: Summary of Passive Control Method

S. No	**Slits Information**	**Cylinder Geometry**	**Reference**	**Findings**
1.	**No Slits**	**Square cylinder**	Aydin et al.	Study was carried out on square cylinder with holes. The holes on the cylinder stretched the vortex formation.
			He et al.	The geometry chosen for the study was square cylinder with rounded edges. The increase in the round cut decreases the drag coefficient.
			Deepak et al	At 45% tilt angle, the drag coefficient was reduced with increase in Re.

		Circular cylinder	Wu et al.	The study was carried out on a circular cylinder with a hinge plate. It was found that, Addition of the hinge plate reduced the drag coefficient.
			Donglai et al	A circular cylinder with splitter plate was used and it was found that the Addition of splitter plates decreased the drag coefficient of 51.5 %.
			Prashant et al	3D flow around circular cylinder with various aspect ratio was investigated, which resulted in three flow regimes at decrease in drag coefficient observed.
			Shi and Feng	Narrow cuts at cylinder's upper and lower stagnant points were made to decrease the vortices and reduction in drag was observed.
2.	**Parallel Slits**	**Square Slotted Cylinder**	Tahir Durhasan	Experimental investigations were performed on a slotted square cylinder at Re 6000 and it was concluded that the self-issuing jet impacts the vortex shedding and drag coefficient by 42%.
			Hangan and Kim	The slot in the square penetrates the near wake region and contributes in drag reduction and decrease in pressure fluctuations. The reduction in drag is up to 10%.
		Circular slotted cylinder	Jian sheng et al.	The increase in the slits width decreased the drag coefficient.
			Wang et al	The suitable gap ratio for reduced heat transfer was identified as 2.0.
			Ordia et al	The increase in width of the cylinder has effect on vortex shedding.
			Gao et al	With increase of slits width from 0.05 to 1.15D decrease drag coefficient by 0.025.

		Parallel, Converging and Diverging Slits	Verma et al	Effects of shape, size and position of slit at Re500 and resulted that addition of slits suppresses VIV and drag.
			Sharma et al	Simulations were carried out at Re 60-180, on cylinder with parallel, converging and diverging slits. The modified cylinders were able to decrease the pressure drag and resulted in a low Strouhal number.
3.	**Perpendicular and inclined Slits**	**Perpendicular Slits**	Peng et al	The optimum slit to width ratio found for drag coefficient was 0.1 to 0.5.
			Hsu et al	At Re 100 to 500 the drag for the cylinder with normal slit increases with the increase in slit's width. The drag also increases with the increase in slit's width.
		Slotted Cylinder with Parallel, Inclined and Perpendicular Slits	Bao et al	The highest decrease in drag coefficient was observed to be 28% with slit to dia ration of 0.3.
		6 modified cylinders with slits at different angles	Olsen and Rajagopalen	Six modified cylinders with slits were used at different angles. It Resulted in constant vortex shedding and reduced drag.

2.3 Focus of current study

The current study focuses on reducing the drag coefficient with a passive control method i.e., multiple slits parallel to the flow. It is hypothesized that the use of multiple slits parallel to the flow in the form of double slits would further reduce the drag coefficient for flow around a cylinder, compared to the use of a single slit. This is because the double slits provide multiple passages for flow through the cylinder, leading to a split wake region and reduced drag. This hypothesis is supported by previous studies on flow control using multiple slits and will be investigated in the current study through numerical simulations. The paper is organized in such a way that section 2 presents the methodology of the paper including the governing equations and mesh independent study performed. In Section 3 results and outcomes are discussed followed by the Conclusion and Future work.

CHAPTER 3: METHODOLOGY

3.1 Governing Equations

The governing equations for our problem were the incompressible Momentum and continuity equations written in the vector form are as,

Continuity Equation

$$\nabla . \bar{v} = 0 \tag{1}$$

Momentum equation

$$\frac{\partial \bar{v}}{\partial t} + \bar{v} . \nabla \bar{v} = -\nabla p + Re^{-1} \nabla^2 \bar{v} \tag{2}$$

Where, v represents the velocity, t time, p density and Re Reynold's number.

3.2 Numerical Details

In this paper, we utilized the open-source CFD software OpenFOAM to solve the fluid flow equations. We employed the incompressible solver SimpleFOAM for the pressure-velocity coupling of the fluid flow in the domain of interest. The cylinder domain is illustrated in Fig. 1. The cylinder's center in the computational domain was chosen as the origin of the coordinate system with a downstream direction indicating the positive x-axis. The top, bottom, inlet, and outlet boundaries of the domain were placed at y=20D, y=-20D, x=-20D and x=30 respectively. The boundary conditions at inlet, outlet, top and bottom were specified as a uniform flow, Neuman boundary condition and wall respectively. On the cylinder surface, a no-slip boundary condition was applied. The computational domain and boundary conditions adopted in this study are consistent with those presented in the author's previously published work [35], [36].

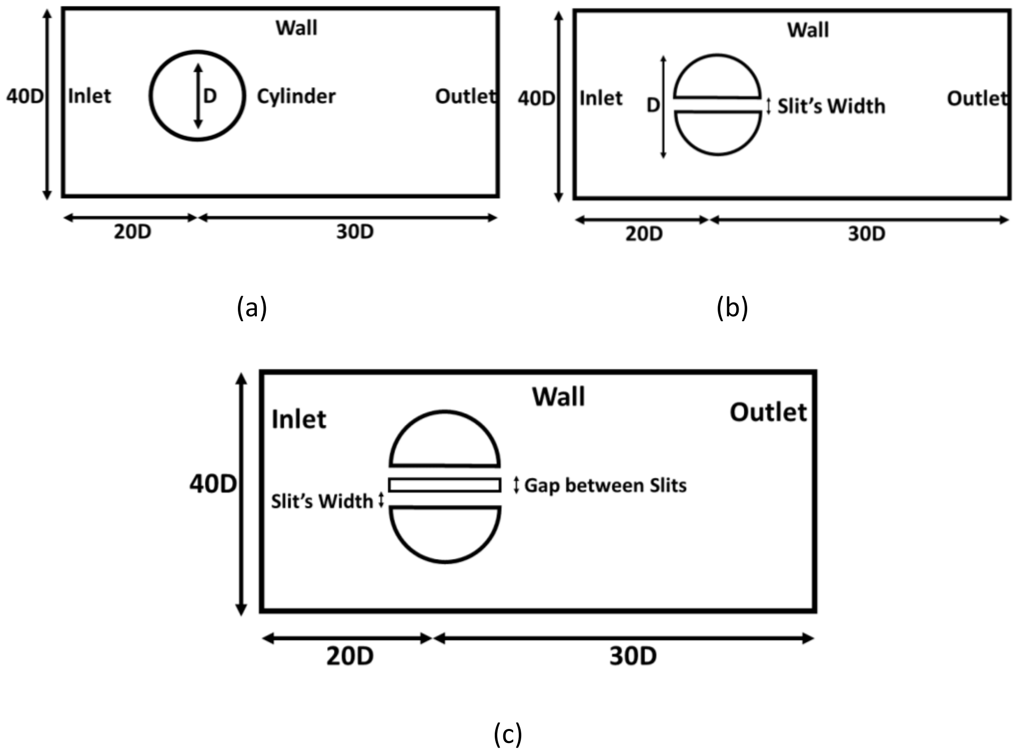


Figure. 1: Computational Domain, (a) For No Slits, (b) For Single Slit, (c) For Double Slits

3.3 Mesh Independent Study

The mesh independent study is presented in Table 2 which was conducted on a no-slit cylinder at Re 200 using five different grid types: Grid A, B, C, D, and E. The mean drag coefficient was employed to assess the sensitivity of the grid independence study, with Grid D and E exhibiting similar results up to two decimal points and no significant differences. To illustrate the mesh used for these simulations, Fig. 3 displays the resulting structured mesh.

Table. 2: Mesh Independent Study

Mesh	Number of Elements	Cd
A	47,500	1.1503
B	190,000	1.2
C	545,000	1.22
D	760,000	1.2515
E	1,200,000	1.256

In Fig. 2, the results obtained from Grid D were compared with those published in the literature [13], [40]–[46], demonstrating close agreement between the computational data and previously reported findings. Based on these results, Grid D was selected for further detailed simulations.

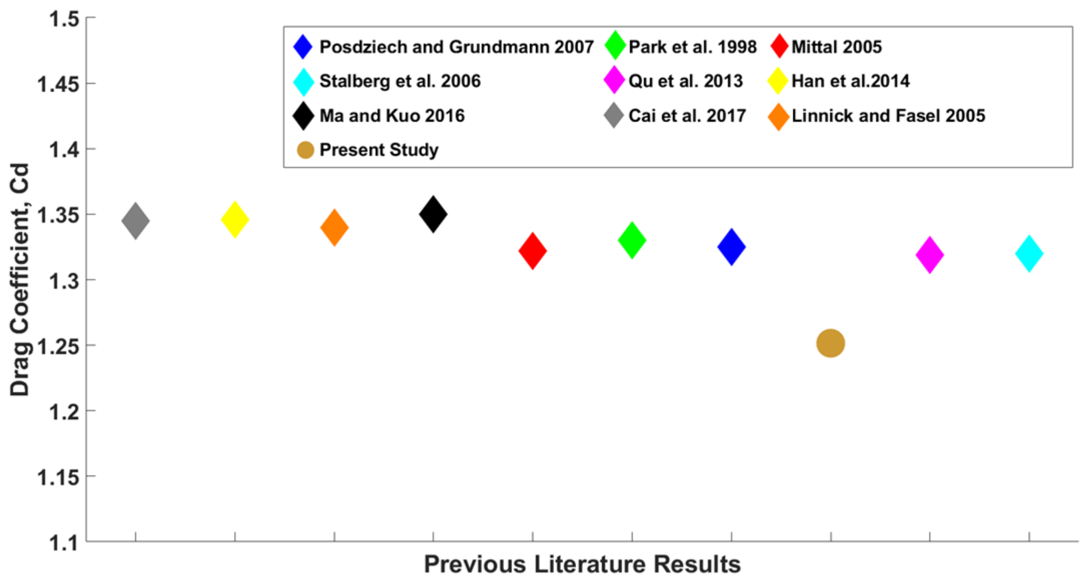


Figure. 2: Comparison of Present Study with published literature

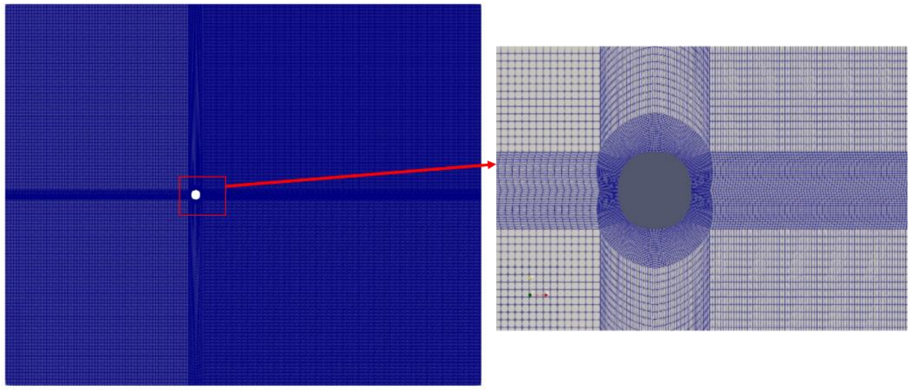


Figure 3 Final Mesh after Mesh Independent Study

3.4 Simulation parameters

This study involved conducting two-dimensional simulations on no, single, and double-slit cylinders with slits aligned in the flow direction. The study was limited to the laminar range of Reynolds numbers ranging from 50-200. Other parameters used were the width to dia ratio (W/D) and the gap between the slits (L) and were set to 0.1 and 0.15, respectively. In total, 28 simulations were performed, comprising four for no slit, eight for a single slit, and 16 for a double-slotted cylinder. A summary of the simulation parameters used in our study is listed in Table

Table. 3: Simulation parameters and variations

Parameters	Variations
Geometry	No Slit
	Single Slit
	Double Slit
Width to Dia Ratio (W/D)	0.1
	0.15
Gap between slits (L)	0.1
	0.15
Reynolds Number (Re)	50
	100
	150
	200

CHAPTER 4: Results and Discussion

4.1 No Slit and Single Slit Cylinder

4.1.1 Force Statistics

Fig. 4 and 5 depict the variation of the coefficient of drag over time for the no slit and single slit cylinder cases, respectively. The graphs display a steady-state behavior of the coefficient of drag, indicating that the flow field has reached a stable state. This observation is supported by the literature and shows a low level of fluctuation in the coefficient of drag value over time which is consistent with this behavior.

In Fig. 4, the drag force coefficient variations for no slit cylinder were analyzed and plotted against the Reynolds number in Fig. 4. The results showed that the drag coefficient reduced significantly with an increase in Reynolds number. With an increase of Reynolds number from 50 to 100, then 150 and 200 a reduction in drag coefficient of be 9%, 2%, and 7%respectively was observed. The reduction of in drag coefficient can be attributed to the increased turbulence intensity at higher Reynolds numbers that promotes momentum transfer, leading to reduced drag. The observed reduction in drag coefficient agrees with previously published literature.

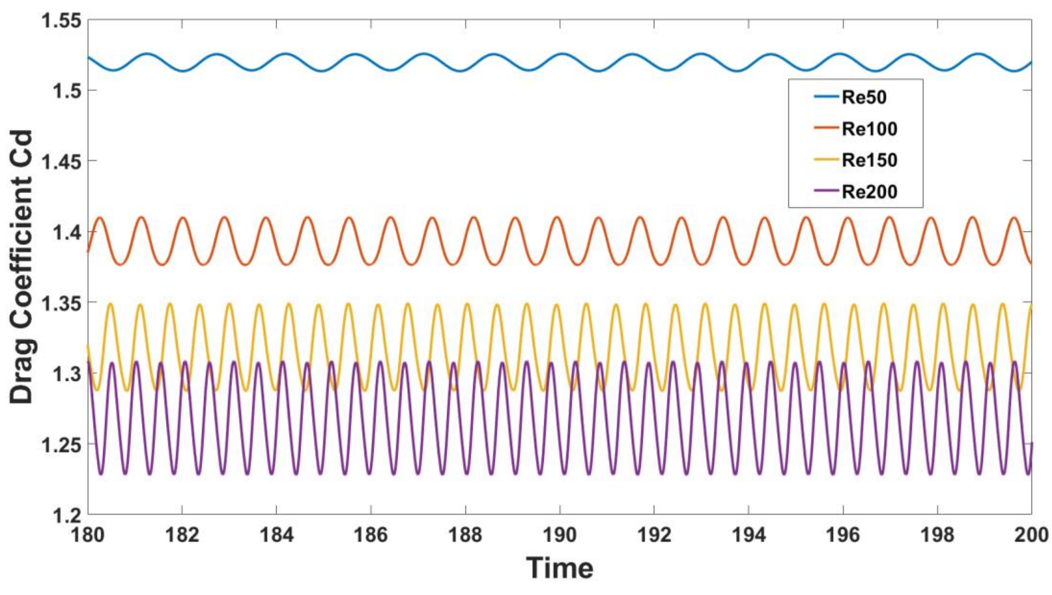
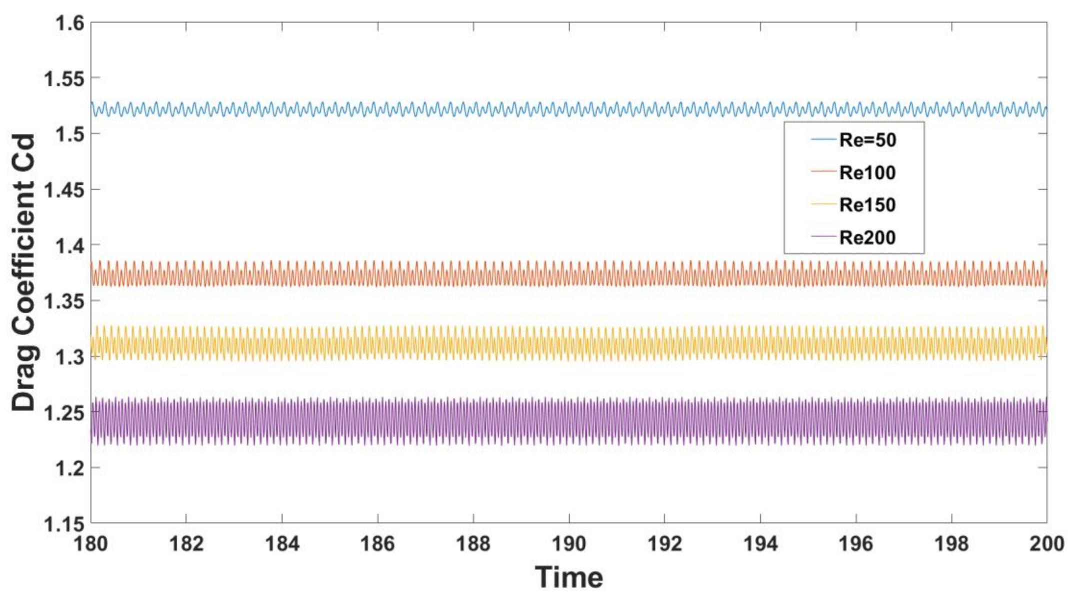
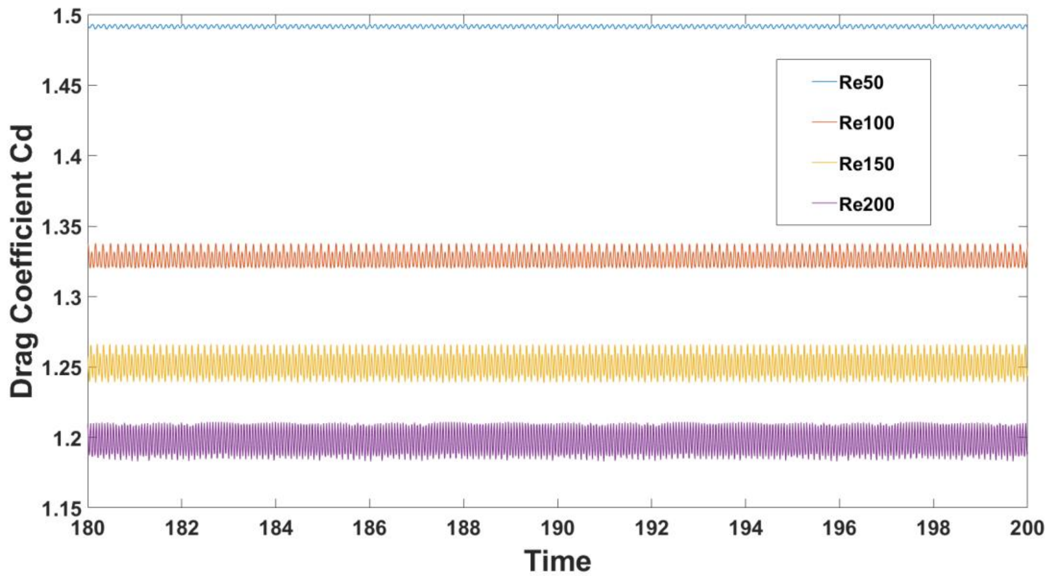


Figure. 4: Force statistics of flow around no slit cylinder

The variations of the drag coefficient for a single slit cylinder are presented in Fig. 5(A) and (B). Fig. 5(A) shows the drag coefficient variations for a width ratio of 0.1, while Fig. 5(B) shows the variations for a width ratio of 0.15 at the specified range of Reynolds numbers. It was observed from the results that the addition of a slit to the cylinder significantly reduced the drag coefficient. At a Reynolds number of 200, the drag coefficient for a single slit cylinder decreased from 1.25 to 1.24 for a width ratio of 0.1 and from 1.25 to 1.18 for a width ratio of 0.15, which corresponds to a 5% reduction. The maximum reduction in the drag coefficient was observed for a Reynolds number of 150 and a width ratio of 0.15, where the drag coefficient decreased by 7%. The trend of the drag coefficient was like that of the no-slit cylinder, i.e., an increase in the Reynolds number from 50 to 200 resulted in a significant decrease in the drag coefficient for both width ratios. "The results obtained from the single slit cylinder were in comparison to the previously published literature and were in good agreement [33],[38]".



(a)



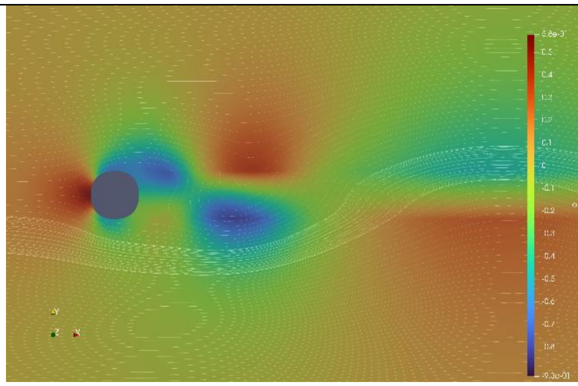
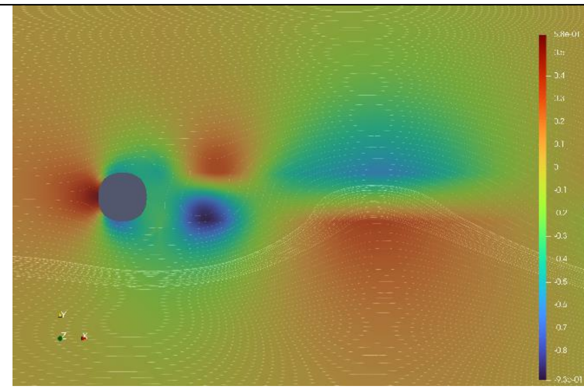
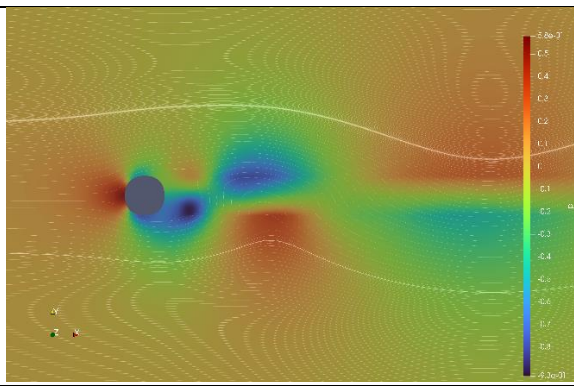
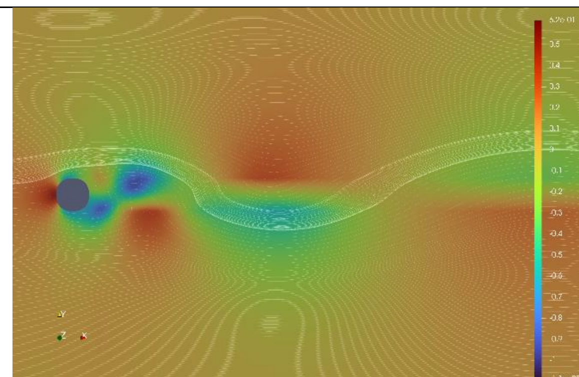
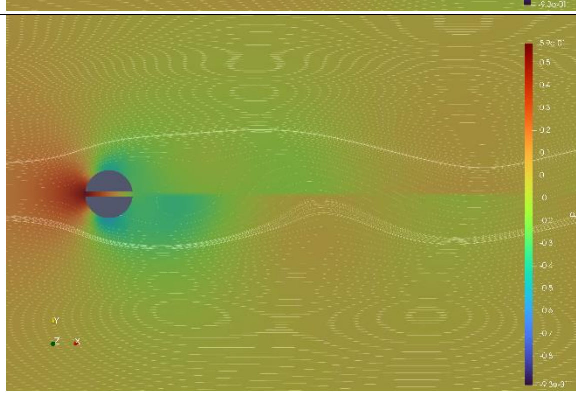
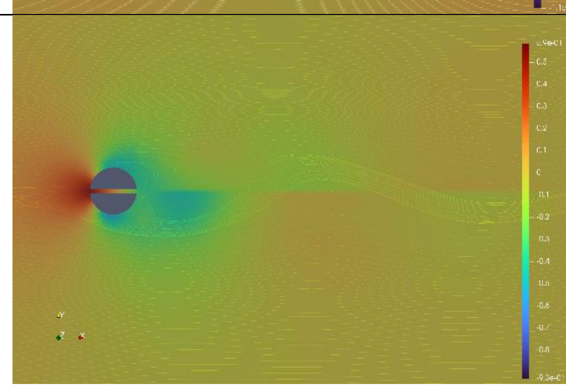
(b)

Figure. 5: Drag coefficients for flow around single slit cylinder, (a): Width to dia ratio = 0.1, (b): Width to dia ratio = 0.15.

4.1.2 Flow Statistics

The computational results for pressure and velocity contours along with streamlines, for no slit and single slits at the range of Reynolds numbers, are given in Fig. 6 and 7. The results for the no-slit cylinder indicate that at low Reynolds numbers, the flow is smooth and symmetric, with streamlines remaining parallel to the axis of the cylinder. As the Reynolds number increases, however, the flow becomes more complex, and the streamlines start to deviate from their original parallel paths, eventually separating from the surface of the cylinder and forming vortices in the wake region. These vortices cause a decrease in pressure behind the cylinder, resulting in a drag force. The velocity contours show that the maximum velocity occurs at the top and bottom of the cylinder, while the minimum velocity occurs at the surface of the cylinder due to the no-slip boundary condition.

The addition of slits to the cylinder leads to a more uniform and less turbulent flow as compared to the no-slit cylinder. The results indicate that the presence of slits causes a significant change in the flow pattern, which can be attributed to the jet injection effect created by the slits. The reduction in turbulence is observed to be more significant for wider slits. The streamlines also show a distinct change in the flow pattern due to the presence of the slits. The flow around the slotted cylinder is observed to be more streamlined with less vortex shedding as compared to the no-slit cylinder. Additionally, the slits help in reducing the size of the wake region behind the cylinder, resulting in a reduction of pressure drag. The results presented in this study highlight the importance of the presence of slits in altering the flow characteristics of a cylinder and improving its aerodynamic performance, especially in laminar conditions.

(i)		(ii)	
(iii)		(iv)	
(v)		(vi)	

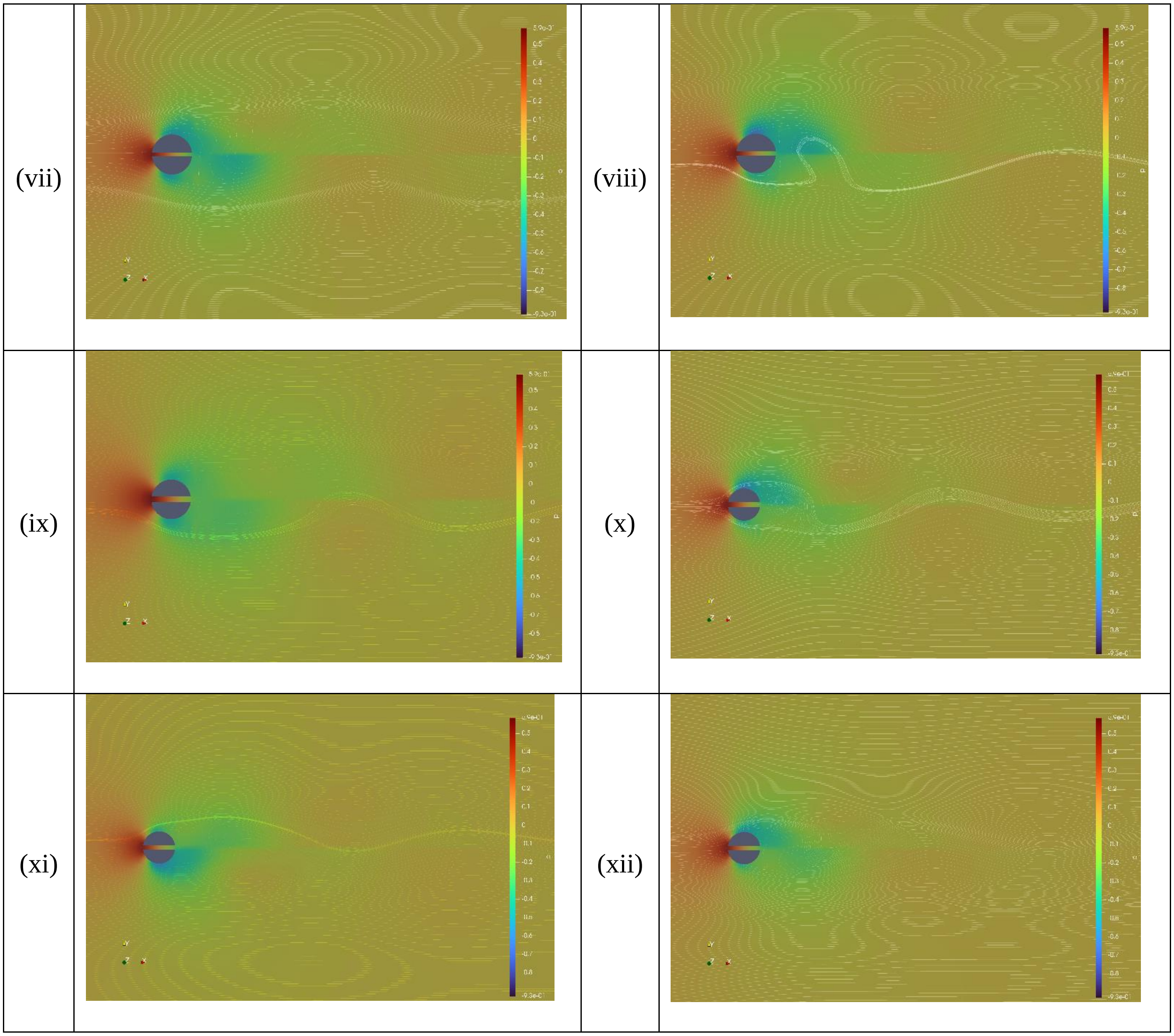
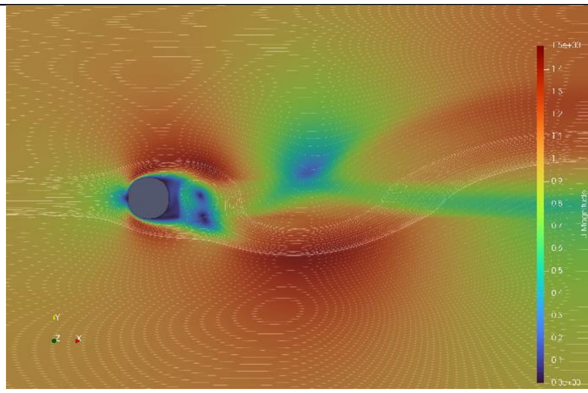
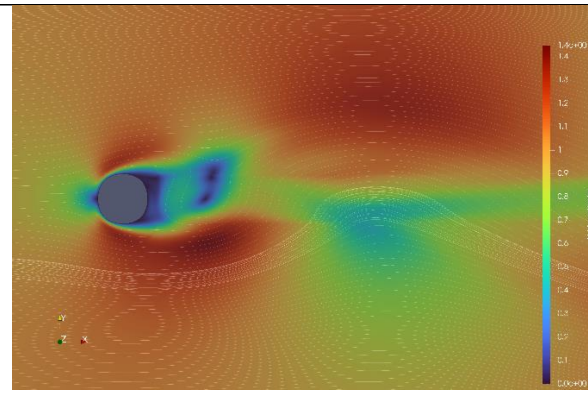
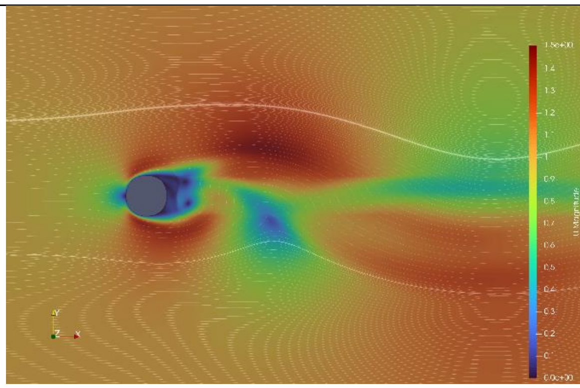
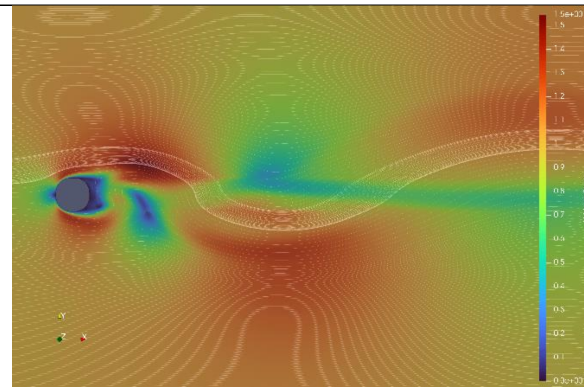
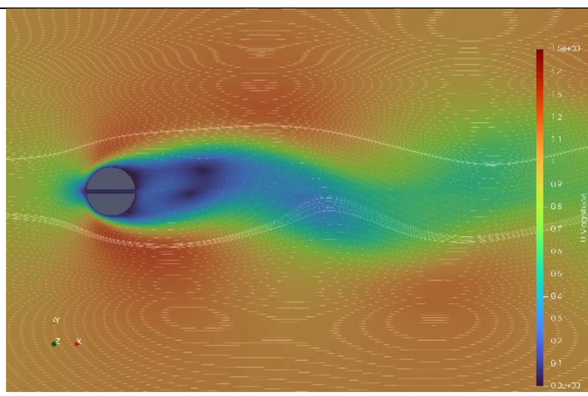
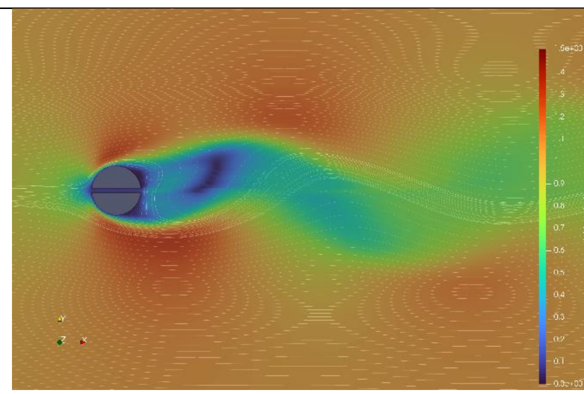
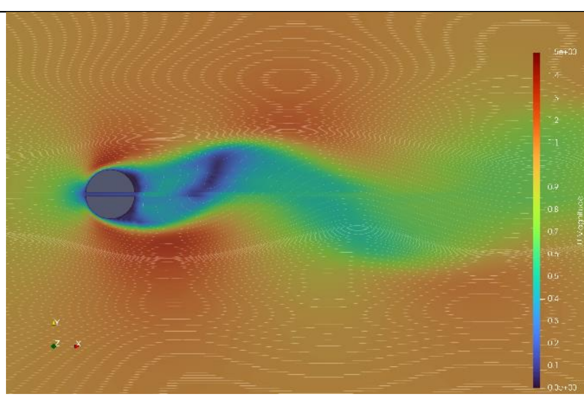
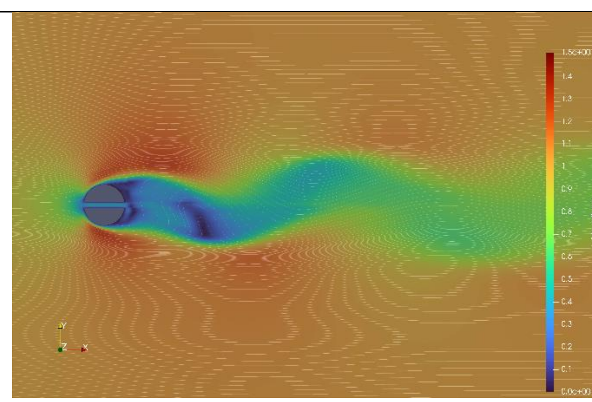
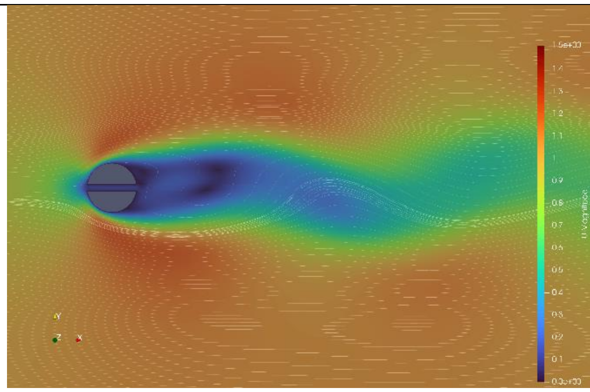


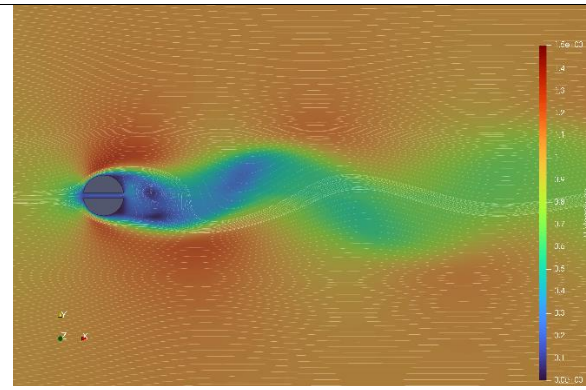
Figure. 6: Pressure contours and Streamlines for no slit and single slit: (i) c0r1 (ii) c0r2 (iii) c0r3 (iv) c0r4 (v) c1w1r1 (vi) c1w1r2 (vii) c1w1r3 (viii) c1w1r4 (ix) c1w2r1 (x) c1w2r2 (xi) c1w2r3 (xii) c1w2r4

(i)		(ii)	
(iii)		(iv)	
(v)		(vi)	
(vii)		(viii)	

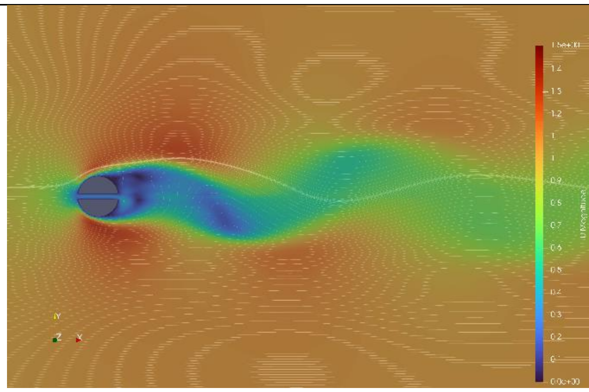
(ix)



(x)



(xi)



(xii)

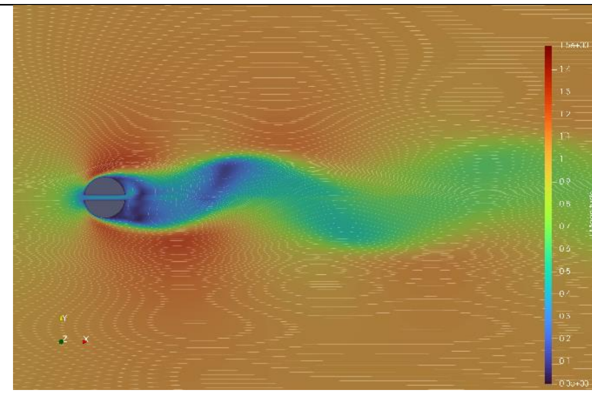
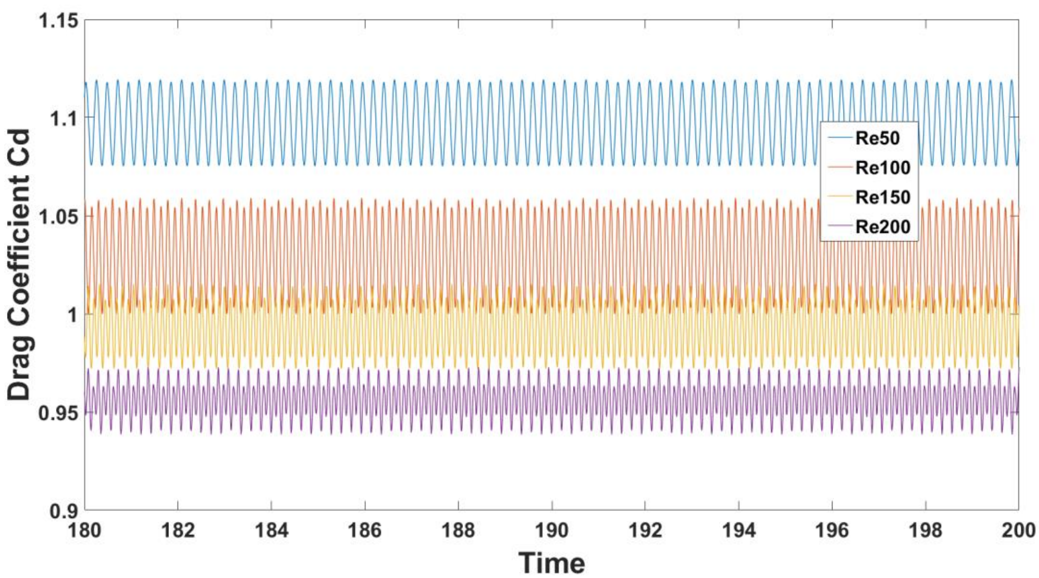


Figure. 7: Velocity contours and streamlines for no slit and single slit (i) c0r1 (ii) c0r2 (iii) c0r3 (iv) c0r4 (v) c1w1r1 (vi) c1w1r2 (vii) c1w1r3 (viii) c1w1r4 (ix) c1w2r1 (x) c1w2r2 (xi) c1w2r3 (xii) c1w2r4

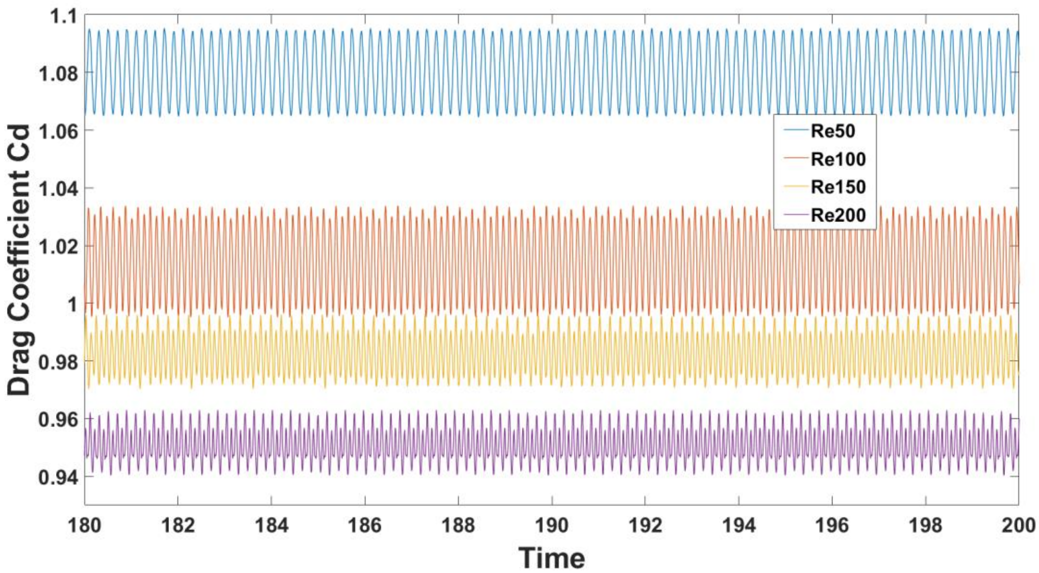
4.2 Double Slits

4.2.1 Force statistics for double slits cylinder

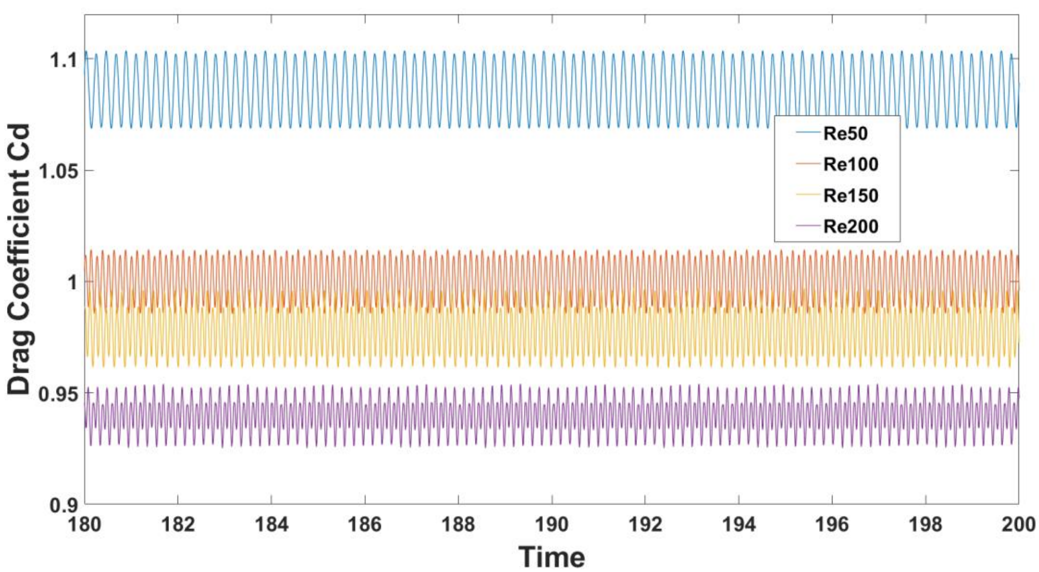
In Fig. 7, we present the force characteristics of the double-slotted cylinder, where the effect of two main parameters - the gap between the slits and the width-to-diameter ratio - on the drag coefficient is investigated. The drag coefficient results are shown in Fig. 7(A-D), where Fig. 7(A) depicts the drag coefficient at width ratio and slits spacing of 0.1, Fig. 7(B) shows the drag coefficient at width ratio 0.1 and spacing of 0.15, Fig. 7(C) shows the drag coefficient at width ratio 0.15 and spacing of 0.1, and Fig. 7(D) shows the drag coefficient at width ratio and slits spacing of 0.1. It is observed that the presence of double slits remarkably reduces the drag coefficient compared to that of a normal cylinder. For instance, at a Reynolds number of 200, the drag coefficient of the double-slotted cylinder with a width ratio and gap of 0.1 was reduced by 22% compared to the normal cylinder. When compared to a single-slotted cylinder, the reduction in drag coefficient was observed to be 21%. While, for the same geometric parameters, at Re 50 the reduction in the drag coefficient for a double-slotted cylinder was recorded as 28%. Furthermore, the minimum value of the drag coefficient, 0.92, was obtained for the double-slotted cylinder with a width ratio and gap between slits of 0.15 for Reynolds number 200. On the other hand, the maximum drag coefficient value for a double-slotted cylinder, 1.1, was obtained for the width ratio and gap between slits of 0.1. Notably, both the width ratio and gap between the slits significantly impacted the drag coefficient, and an increase in either parameter resulted in a decrease in the drag coefficient. Moreover, the Reynolds number effect on the double-slotted cylinder was similar to that of the single-slotted and no-slit cylinders, where an increase in the Reynolds number decreased the drag coefficient. For instance, by increasing the Reynolds number from 50 to 200, the drag coefficient for each set of cases decreased by 10%, 11%, 12%, and 13%, respectively. Overall, the results suggest that the addition of double slits to a cylinder can effectively reduce the drag coefficient, and the width ratio and gap between the slits are significant parameters that impact the drag coefficient.



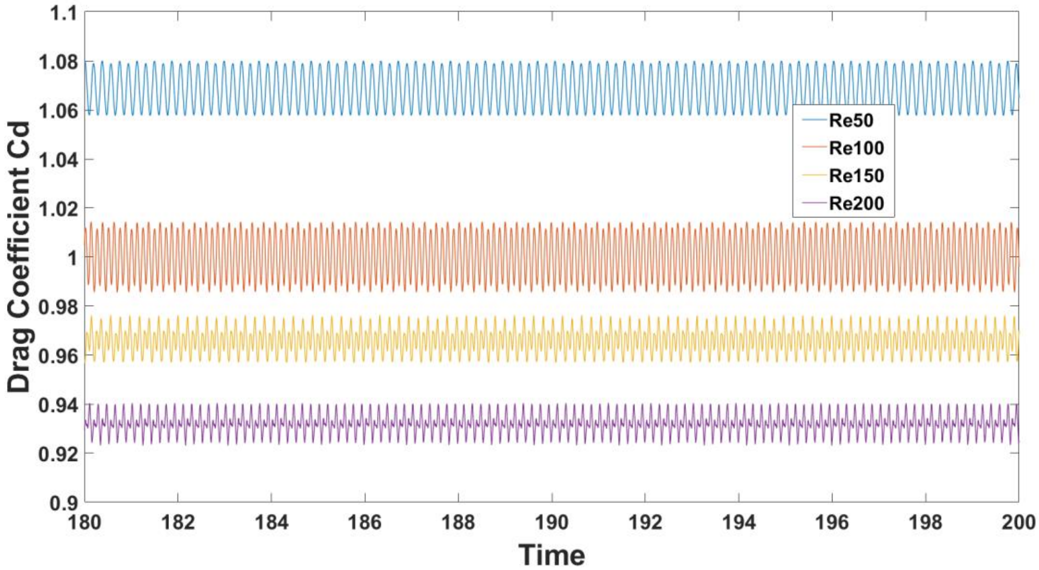
(a)


Drag Coefficient Cd
Time
Re50
Re100
Re150
Re200
1.1
1.08
1.06
1.04
1.02
1
0.98
0.96
0.94
180
182
184
186
188
190
192
194
196
198
200

(b)


Drag Coefficient Cd
Time
Re50
Re100
Re150
Re200
1.1
1.05
1
0.95
0.9
180
182
184
186
188
190
192
194
196
198
200

(c)



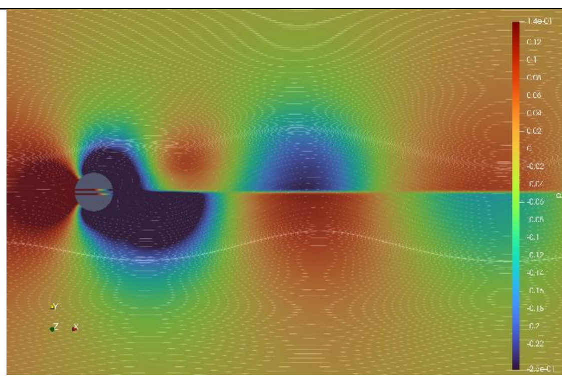
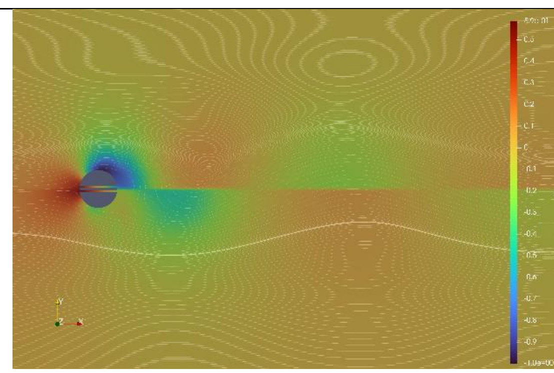
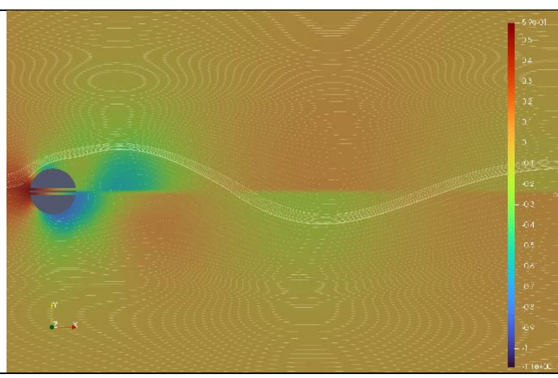
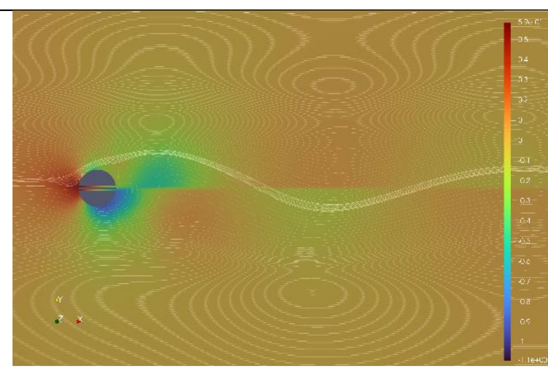
(d)

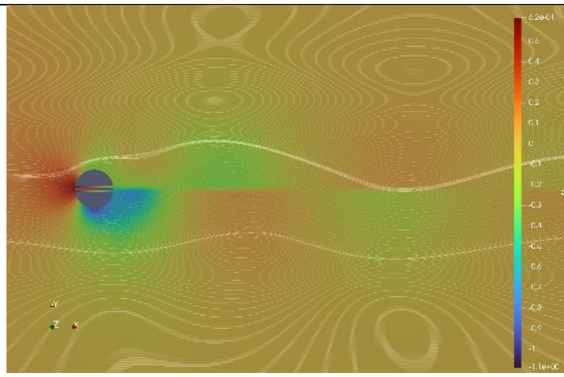
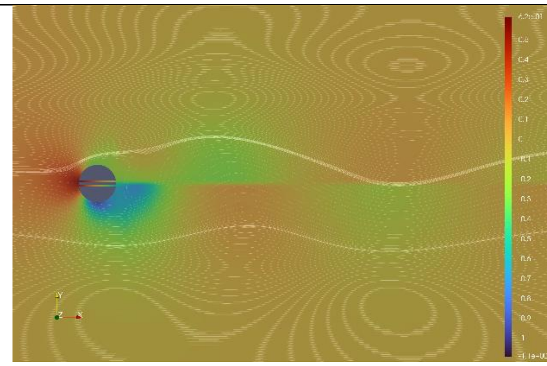
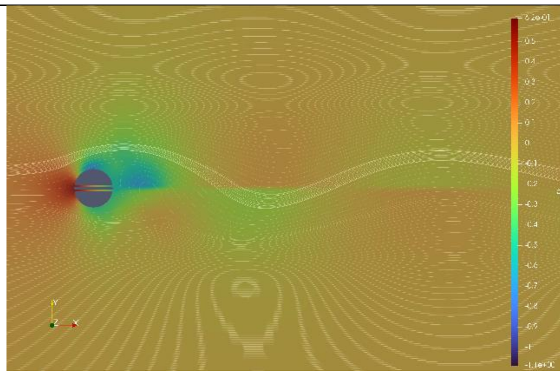
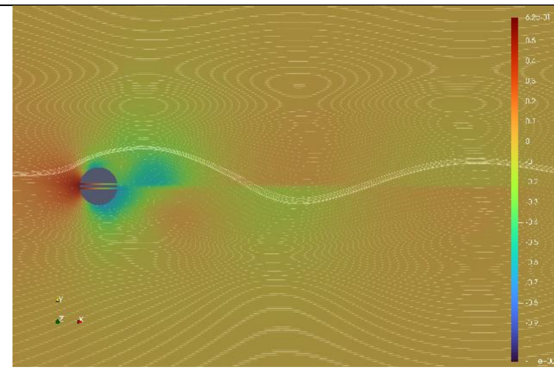
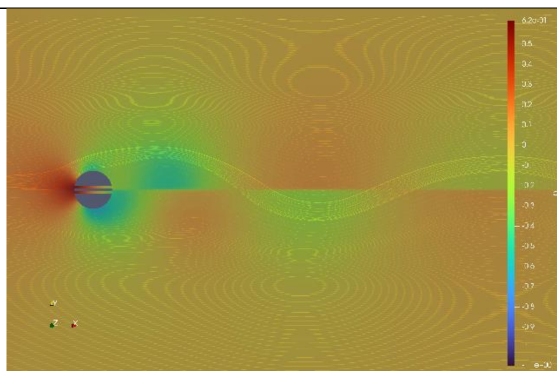
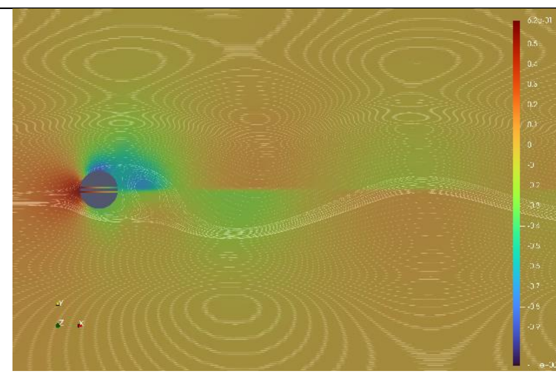
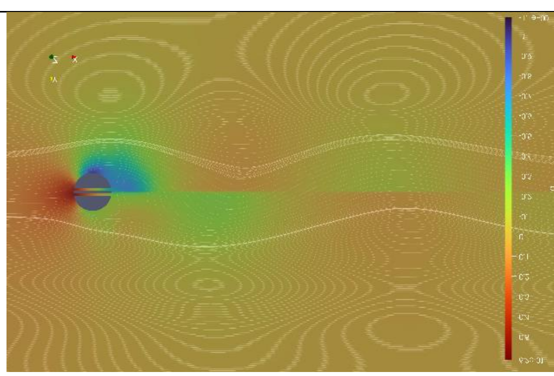
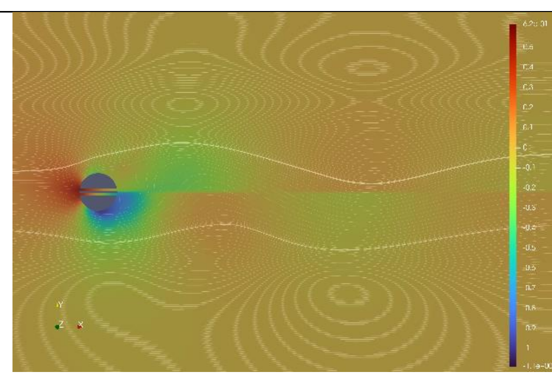
Figure. 8: Drag coefficients for of flow around a double-slotted cylinder, (a) For a width to dia ratio and gap between slits = 0.1, (b) For a width to dia ratio = 0.1 and gap between slits = 0.1, (c) For a width to dia ratio = 0.15 and gap between slits = 0.1, (d) For a width to dia ratio and gap between slits = 0.15

4.2.3 Flow Characteristics for double slits cylinder

The addition of double slits to the cylinder also significantly impacted the flow characteristics compared to the normal and single-slotted cylinders. As shown in Fig. 7, the drag coefficient for the double-slotted cylinder was remarkably reduced compared to the normal cylinder, with reductions of up to 22% observed at Re 200 for the width ratio and gap between slits of 0.1. The simulations as shown in Fig. 8, also revealed that the double-slotted cylinder had a similar effect on the flow pattern to the single-slotted cylinder. The flow was more uniform and less turbulent, with a distinct change in flow pattern due to the jet injection effect created by the slits. The wake region behind the double-slotted cylinder exhibits a distinct change in flow characteristics compared to the no-slit and single-slotted cylinders. The results indicate that the presence of double slits helps in reducing the size of the wake region behind the cylinder, resulting in a reduction of pressure drag. This reduction in the size of the wake region can be attributed to the jet injection effect created by the double slits, which helps in entraining high-velocity fluid from outside the boundary layer into the wake region, thus reducing the size of the wake. The pressure distribution around the double-slotted cylinder is also affected by the presence of double slits. The computational results show that the pressure distribution behind the double-slotted cylinder is less negative as compared to the no-slit and single-slotted cylinders. The pressure coefficient at the center of the double-slotted cylinder is observed to be lower than that of the no-slit and single-slotted cylinders. This reduction in pressure coefficient indicates a reduction in the pressure drags for the double-slotted cylinder. Similarly, the velocity contours around the double-slotted cylinder exhibit a distinct change in flow pattern compared to the no-slit and single-slotted cylinders. The results show that maximum velocity occurs at the top and bottom of the cylinder, while the minimum velocity occurs at the surface of the cylinder due to the no-slip boundary condition. The velocity contours also indicate the formation of a jet-like flow in the region between the two slits, which is responsible for reducing turbulence and entraining high-velocity fluid from outside the boundary layer into the wake region. This jet-like flow is observed to be more pronounced for wider slits.

Overall, the computational results suggest that the addition of double slits to a cylinder can significantly alter the wake region, pressure distribution, and velocity contours, leading to a reduction in drag and an improvement in aerodynamic performance. The results highlight the importance of considering the effect of double slits in the design and optimization of aerodynamic systems.

(i)		(ii)	
(iii)		(iv)	

(v)		(vi)	
(vii)		(viii)	
(ix)		(x)	
(xi)		(xii)	

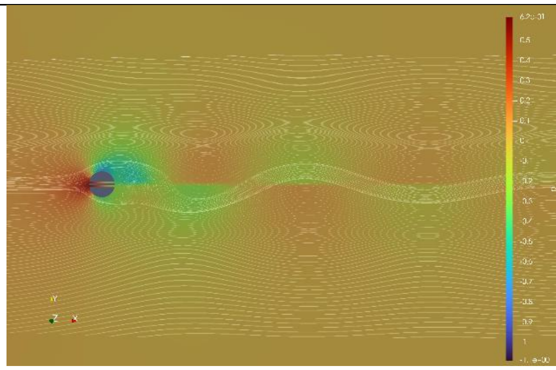
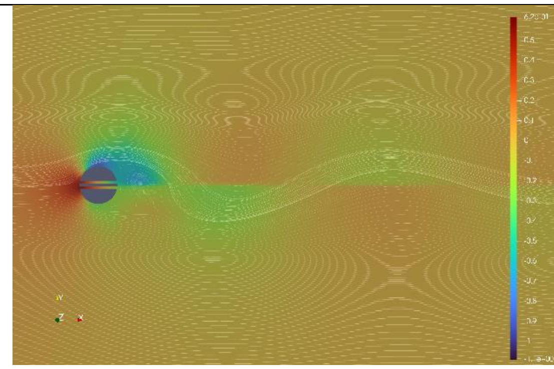
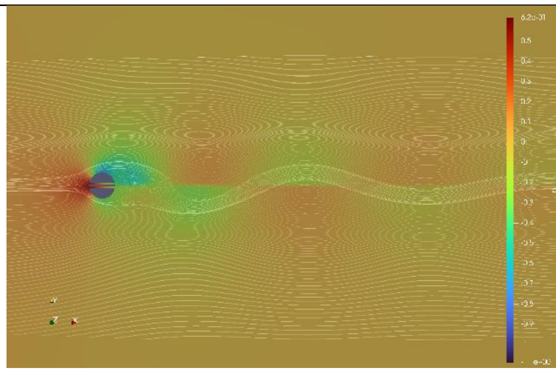
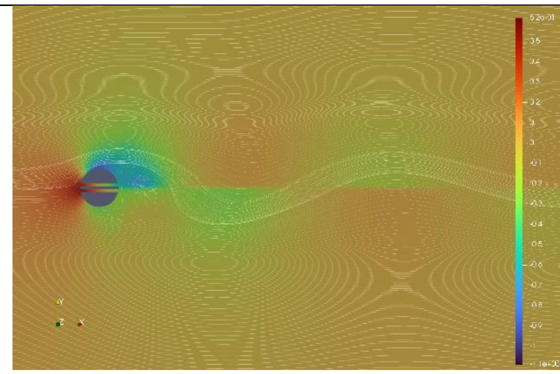
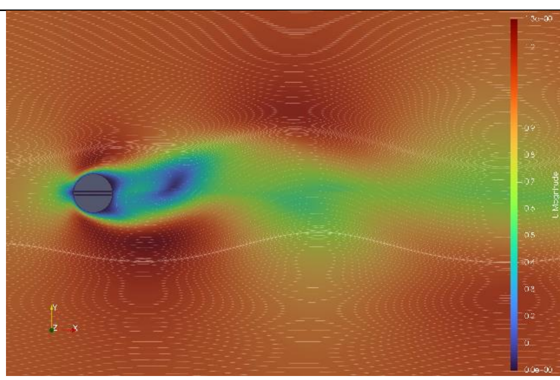
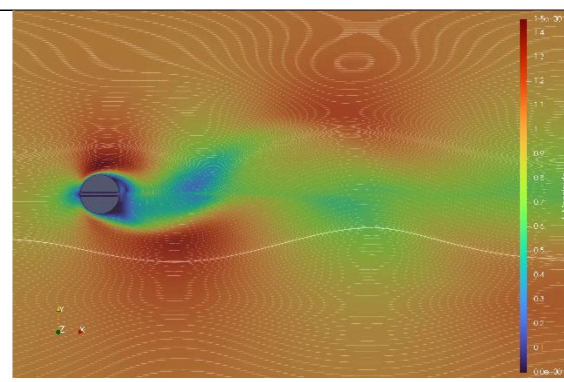
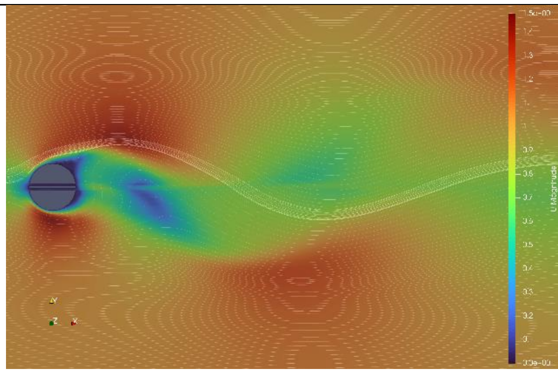
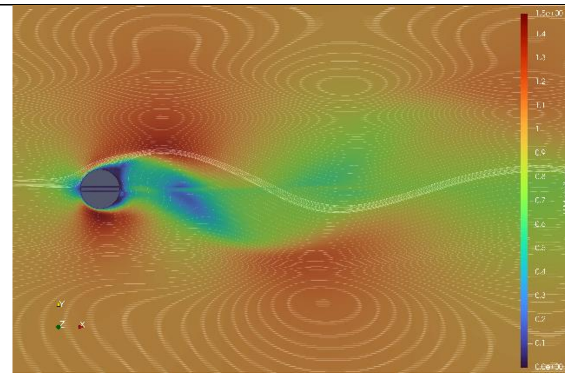
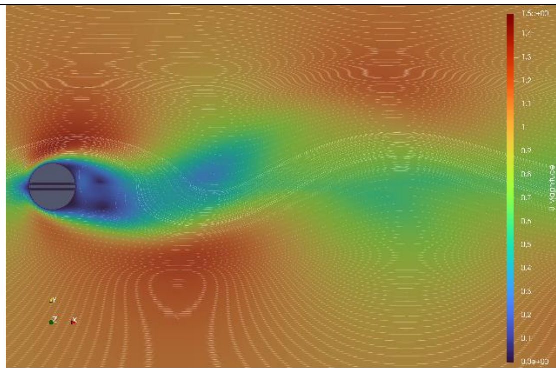
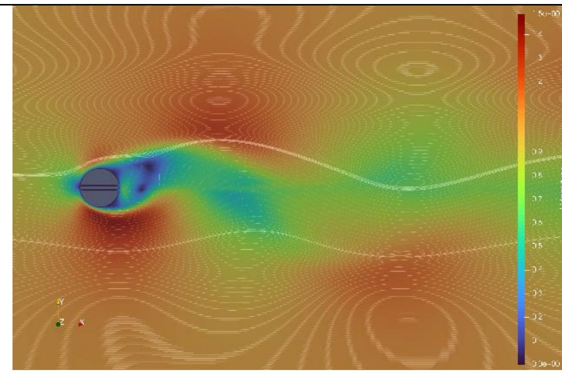
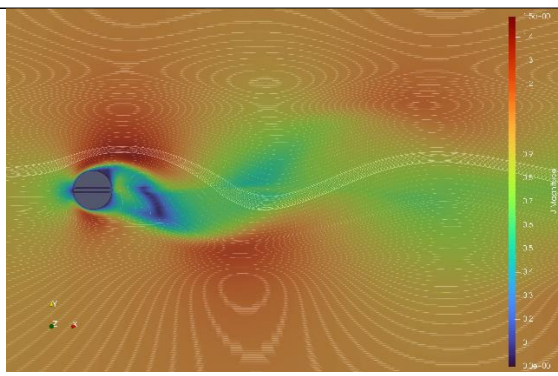
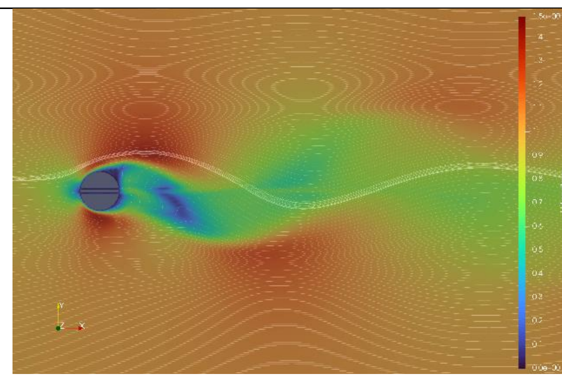
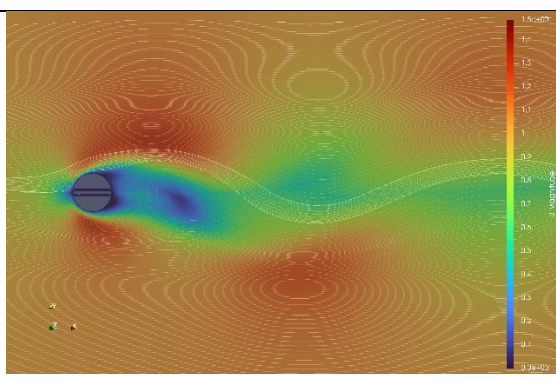
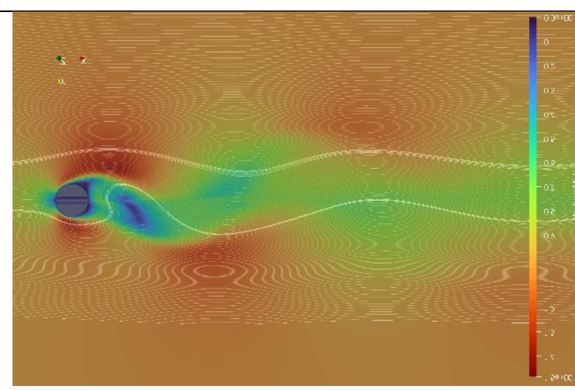
(xiii)		(xiv)	
(xv)		(xvi)	

Figure. 9: Pressure contours and streamlines for double slit cylinders (i) c2w1l1r1 (ii) c2w1l1r2 (iii) c2w1l1r3 (iv) c2w1l1r4 (v) c2w1l2r1 (vi) c2w1l2r2 (vii) c2w1l2r3 (viii) c2w1l2r4 (ix) c2w2l1r1 (x) c2w2l1r2 (xi) c2w2l1r3 (xii) c2w2l1r4 (xiii) c2w2l2r1 (xiv) c2l2r2 (xv) c2w2l2r3 (xvi) c2w2l2r4

(i)		(ii)	

(iii)		(iv)	
(v)		(vi)	
(vii)		(viii)	
(ix)		(x)	

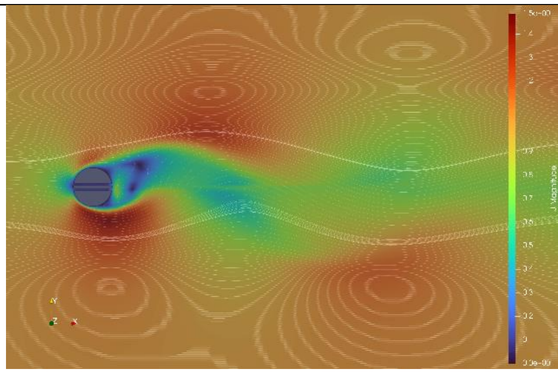
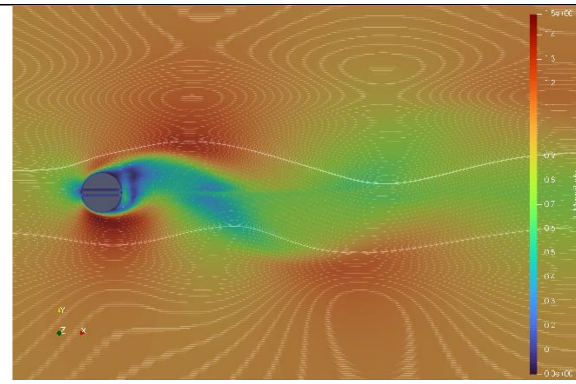
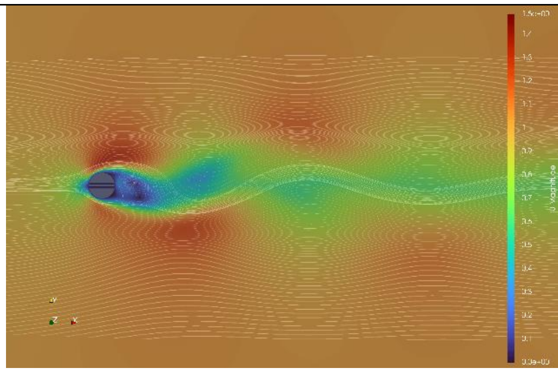
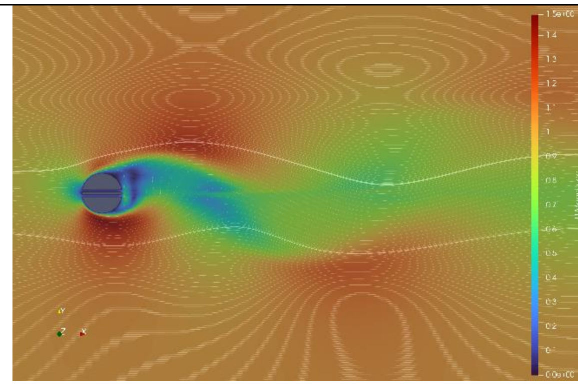
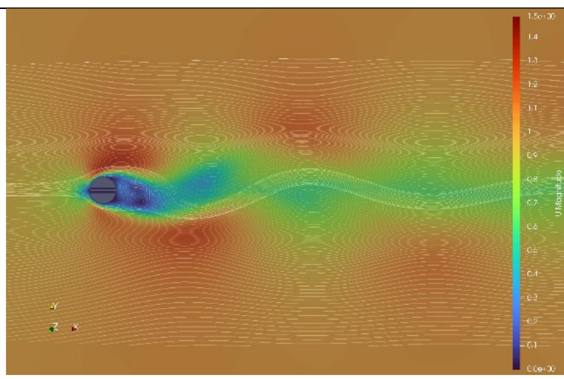
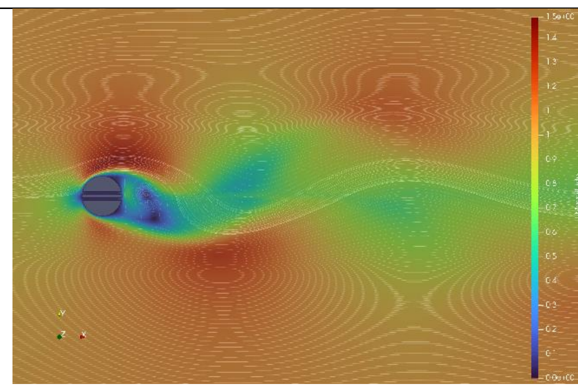
(xi)		(xii)	
(xiii)		(xiv)	
(xv)		(xvi)	

Figure. 10: Velocity contours and streamlines for double slit cylinders (i) c2w1l1r1 (ii) c2w1l1r2 (iii) c2w1l1r3 (iv) c2w1l1r4 (v) c2w1l2r1 (vi) c2w1l2r2 (vii) c2w1l2r3 (viii) c2w1l2r4 (ix) c2w2l1r1 (x) c2w2l1r2 (xi) c2w2l1r3 (xii) c2w2l1r4 (xiii) c2w2l2r1 (xiv) c2w2l2r2 (xv) c2w2l2r3 (xvi) c2w2l2r4

CHAPTER 5: CONCLUSION

Based on the results and discussions presented in this study, it can be concluded that the addition of slits to a cylinder can significantly alter its flow characteristics and improve its aerodynamic performance. The present study investigated the effects of single and double slits on the flow around a circular cylinder using computational fluid dynamics. The simulations revealed that the presence of slits causes a reduction in drag coefficient, turbulence, and a decrease in the size of the wake region behind the cylinder, resulting in a reduction of pressure drag. Additionally, the slits caused a significant change in the flow pattern, which can be attributed to the jet injection effect created by the slits.

The results showed that the reduction in drag coefficient was more significant for double-slotted cylinders than for single-slotted cylinders. The drag coefficient was observed to decrease by 22% for the double-slotted cylinder with a width-to-gap ratio of 0.1 and Reynolds number of 200, compared to a 21% reduction for the single-slotted cylinder. The lowest value of drag coefficient was obtained for the double-slotted cylinder with a width-to-gap ratio of 0.15 and a Reynolds number of 200, while the maximum drag coefficient was obtained for the width-to-gap ratio of 0.1. The increase in the width ratio and gap between slits decreased the drag coefficient.

Overall, the findings of this study have important implications for the design of various engineering systems such as heat exchangers, turbines, and wind turbines, where reducing drag and increasing efficiency is critical. Future studies can further explore the effects of slits on cylinders of different shapes and sizes and investigate the flow characteristics at higher Reynolds numbers.

REFERENCES

[1] C. Xu, Y. Mao, and Z. Hu, "Control of cylinder wake flow and noise through a downstream porous treatment," *Aerosp Sci Technol*, vol. 88, pp. 233–243, 2019, doi: 10.1016/j.ast.2019.03.027.

[2] L. H. Feng and J. J. Wang, "Circular cylinder vortex-synchronization control with a synthetic jet positioned at the rear stagnation point," *J Fluid Mech*, vol. 662, pp. 232–259, 2010, doi: 10.1017/S0022112010003174.

[3] U. Latif *et al.*, "Experimental investigation of energy harvesting eel in the wake of bluff body under ocean waves," *Proc IMechE Part M: J Engineering for the Maritime Environment*, vol. 2021, no. 1, pp. 81–92, doi: 10.1177/1475090220949334.

[4] U. Latif *et al.*, "Experimental and numerical investigation of the energy harvesting flexible flag in the wake of a bluff body," *Wind and Structures*, vol. 26, no. 5, p. 279, May 2018, doi: 10.12989/WAS.2018.26.5.279.

[5] W. L. Chen, Q. Q. Zhang, H. Li, and H. Hu, "An experimental investigation on vortex induced vibration of a flexible inclined cable under a shear flow," *J Fluids Struct*, vol. 54, pp. 297–311, 2015, doi: 10.1016/j.jfluidstructs.2014.11.007.

[6] S. Dong, G. S. Triantafyllou, and G. E. Karniadakis, "Elimination of vortex streets in bluff-body flows," *Phys Rev Lett*, vol. 100, no. 20, pp. 1–4, 2008, doi: 10.1103/PhysRevLett.100.204501.

[7] Y. G. Liu and L. H. Feng, "Suppression of lift fluctuations on a circular cylinder by inducing the symmetric vortex shedding mode," *J Fluids Struct*, vol. 54, pp. 743–759, 2015, doi: 10.1016/j.jfluidstructs.2015.01.011.

[8] Y. Bao, Y. Lin, W. Chen, C. K. Rheem, and X. Li, "Numerical investigation of wake and flow-induced vibrations of a rotating cylinder in flow," *Ocean Engineering*, vol. 262, p. 112207, Oct. 2022, doi: 10.1016/J.OCEANENG.2022.112207.

[9] U. Ali, M. D. Islam, and I. Janajreh, "Flow over rotationally oscillating heated circular cylinder at low Reynolds number," *Ocean Engineering*, vol. 265, p. 112515, Dec. 2022, doi: 10.1016/J.OCEANENG.2022.112515.

[10] A. S. Chan, P. A. Dewey, A. Jameson, C. Liang, and A. J. Smits, “Vortex suppression and drag reduction in the wake of counter-rotating cylinders,” *J Fluid Mech*, vol. 679, pp. 343–382, 2011, doi: 10.1017/jfm.2011.134.

[11] W. L. Chen, D. B. Xin, F. Xu, H. Li, J. P. Ou, and H. Hu, “Suppression of vortex-induced vibration of a circular cylinder using suction-based flow control,” *J Fluids Struct*, vol. 42, pp. 25–39, 2013, doi: 10.1016/j.jfluidstructs.2013.05.009.

[12] P. Koutmos, D. Papailiou, and A. Bakrozis, “Experimental and computational study of square cylinder wakes with two-dimensional injection into the base flow region,” *European Journal of Mechanics, B/Fluids*, vol. 23, no. 2, pp. 353–365, 2004, doi: 10.1016/j.euromechflu.2003.09.004.

[13] H. L. Ma and C. H. Kuo, “Control of boundary layer flow and lock-on of wake behind a circular cylinder with a normal slit,” *European Journal of Mechanics, B/Fluids*, vol. 59, pp. 99–114, 2016, doi: 10.1016/j.euromechflu.2016.05.001.

[14] R. Zhu, Q. Zhuang, S. Li, N. Su, F. Bao, and Z. Liu, “Overall flow dynamics characteristics of slit-vent cylinders,” *Aerosp Sci Technol*, vol. 105, p. 106074, 2020, doi: 10.1016/j.ast.2020.106074.

[15] H. Choi, W. P. Jeon, and J. Kim, “Control of flow over a bluff body,” *Annu Rev Fluid Mech*, vol. 40, pp. 113–139, 2008, doi: 10.1146/annurev.fluid.39.050905.110149.

[16] B. T. Aydin, O. Cetiner, and M. F. Unal, “Effect of self-issuing jets along the span on the near-wake of a square cylinder,” *Exp Fluids*, vol. 48, no. 6, pp. 1081–1094, 2010, doi: 10.1007/s00348-009-0793-4.

[17] G. S. He, N. Li, and J. J. Wang, “Drag reduction of square cylinders with cut-corners at the front edges,” *Exp Fluids*, vol. 55, no. 6, 2014, doi: 10.1007/s00348-014-1745-1.

[18] D. Kumar, K. Sourav, P. K. Yadav, and S. Sen, “Understanding the secondary separation from an inclined square cylinder with sharp and rounded trailing edges,” *Physics of Fluids*, vol. 31, no. 7, 2019, doi: 10.1063/1.5097595.

[19] J. Wu, C. Shu, and N. Zhao, “Numerical investigation of vortex-induced vibration of a circular cylinder with a hinged flat plate,” *Physics of Fluids*, vol. 26, no. 6, 2014, doi: 10.1063/1.4880135.

[20] D. Gao, Y. Huang, W. L. Chen, G. Chen, and H. Li, “Control of circular cylinder flow via bilateral splitter plates,” *Physics of Fluids*, vol. 31, no. 5, 2019, doi: 10.1063/1.5097309.

[21] P. Kumar, N. P. Manelil, and S. Tiwari, "Effects of shear intensity and aspect ratio on three-dimensional wake characteristics of flow past surface mounted circular cylinder," *Physics of Fluids*, vol. 31, no. 4, 2019, doi: 10.1063/1.5081795.

[22] X. D. Shi and L. H. Feng, "Control of flow around a circular cylinder by bleed near the separation points," *Exp Fluids*, vol. 56, no. 12, pp. 1–17, 2015, doi: 10.1007/s00348-015-2083-7.

[23] T. Durhasan, "Flow topology downstream of the hollow square cylinder with slots," *Ocean Engineering*, vol. 209, no. May, p. 107518, 2020, doi: 10.1016/j.oceaneng.2020.107518.

[24] H. Hangan and J. Kim, "Aerodynamic slot-control for 2D square prisms," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 91, no. 12–15, pp. 1847–1857, 2003, doi: 10.1016/j.jweia.2003.09.023.

[25] W. Jian Sheng and W. Chen, "Features of flow past a circular cylinder with a slit," *Scientia Iranica*, vol. 23, no. 5, pp. 2097–2112, 2016, doi: 10.24200/sci.2016.3941.

[26] J. Wang and C. Wang, "Heat transfer and flow characteristics of a rectangular channel with a small circular cylinder having slit-vent vortex generator," *International Journal of Thermal Sciences*, vol. 104, pp. 158–171, 2016, doi: 10.1016/j.ijthermalsci.2016.01.006.

[27] L. Ordia, A. Venugopal, A. Agrawal, and S. V. Prabhu, "Vortex shedding characteristics of a cylinder with a parallel slit placed in a circular pipe," *J Vis (Tokyo)*, vol. 20, no. 2, pp. 263–275, 2017, doi: 10.1007/s12650-016-0398-y.

[28] D. L. Gao, W. L. Chen, H. Li, and H. Hu, "Flow around a circular cylinder with slit," *Exp Therm Fluid Sci*, vol. 82, pp. 287–301, 2017, doi: 10.1016/j.expthermflusci.2016.11.025.

[29] S. Alshareef, T. Harman, and T. Ameel, "Fluid Dynamic and Thermal Performance of a Slotted Cylinder at Low Reynolds Number," *SSRN Electronic Journal*, 2023, doi: 10.2139/SSRN.4360289.

[30] M. Goodarzi and K. Asadi, "Slit Effectiveness Analysis on the Lift Fluctuation Caused by Low-Speed Laminar Flow over a Circular Cylinder," *J Aerosp Eng*, vol. 35, no. 2, p. 04021138, Dec. 2021, doi: 10.1061/(ASCE)AS.1943-5525.0001392.

[31] M. Verma, A. Mishra, and A. De, "Flow characteristics of elastically mounted slit cylinder at sub-critical Reynolds number," *Physics of Fluids*, vol. 33, no. 12, p. 123612, Dec. 2021, doi: 10.1063/5.0073368.

[32] B. Sharma and R. N. Barman, "Steady laminar flow past a slotted circular cylinder," *Physics of Fluids*, vol. 32, no. 7, 2020, doi: 10.1063/5.0007958.

[33] B. Sharma and R. N. Barman, "Unsteady flow past slotted circular cylinders in laminar regime: Effect of slit shapes and Reynolds number," *Physics of Fluids*, vol. 35, no. 1, Jan. 2023, doi: 10.1063/5.0130736/2867606.

[34] B. H. Peng, J. J. Miau, F. Bao, L. D. Weng, C. C. Chao, and C. C. Hsu, "Performance of vortex shedding from a circular cylinder with a slit normal to the stream," *Flow Measurement and Instrumentation*, vol. 25, pp. 54–62, 2012, doi: 10.1016/j.flowmeasinst.2011.07.003.

[35] L. C. Hsu and C. L. Chen, "The drag and lift characteristics of flow around a circular cylinder with a slit," *European Journal of Mechanics - B/Fluids*, vol. 82, pp. 135–155, Jul. 2020, doi: 10.1016/J.EUROMECHFLU.2020.02.009.

[36] Z. Bao, G. Qin, W. He, and Y. Wang, "Numerical investigation of flow around a slotted circular cylinder at low Reynolds number," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 183, no. November, pp. 273–282, 2018, doi: 10.1016/j.jweia.2018.11.010.

[37] J. F. Olsen and S. Rajagopalan, "Vortex shedding behind modified circular cylinders," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 86, no. 1, pp. 55–63, 2000, doi: 10.1016/S0167-6105(00)00003-9.

[38] Z. Bao, G. Qin, W. He, and Y. Wang, "Numerical investigation of flow around a slotted circular cylinder at low Reynolds number," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 183, no. November, pp. 273–282, 2018, doi: 10.1016/j.jweia.2018.11.010.

[39] Z. Han, D. Zhou, X. Gui, and J. Tu, "Numerical study of flow past four square-arranged cylinders using spectral element method," *Comput Fluids*, vol. 84, pp. 100–112, 2013, doi: 10.1016/j.compfluid.2013.05.008.

[40] O. Posdziech and R. Grundmann, "A systematic approach to the numerical calculation of fundamental quantities of the two-dimensional flow over a circular cylinder," *J Fluids Struct*, vol. 23, no. 3, pp. 479–499, 2007, doi: 10.1016/j.jfluidstructs.2006.09.004.

[41] J. Park and H. Choi, “Numerical solutions of flow past a circular cylinder at reynolds numbers up to 160,” *KSME International Journal*, vol. 12, no. 6, pp. 1200–1205, 1998, doi: 10.1007/BF02942594.

[42] Z. Han, D. Zhou, J. Tu, C. Fang, and T. He, “Flow over two side-by-side square cylinders by CBS finite element scheme of Spalart-Allmaras model,” *Ocean Engineering*, vol. 87, pp. 40–49, 2014, doi: 10.1016/j.oceaneng.2014.05.006.

[43] S. Mittal, “Excitation of shear layer instability in flow past a cylinder at low Reynolds number,” *Int J Numer Methods Fluids*, vol. 49, no. 10, pp. 1147–1167, 2005, doi: 10.1002/fld.1043.

[44] E. Stålberg, A. Brüger, P. Lötstedt, A. V. Johansson, and D. S. Henningson, “High order accurate solution of flow past a circular cylinder,” *J Sci Comput*, vol. 27, no. 1–3, pp. 431–441, 2006, doi: 10.1007/s10915-005-9043-y.

[45] S. G. Cai, A. Ouahsine, J. Favier, and Y. Hoarau, “Moving immersed boundary method,” *Int J Numer Methods Fluids*, vol. 85, no. 5, pp. 288–323, 2017, doi: 10.1002/fld.4382.

[46] M. N. Linnick and H. F. Fasel, “A high-order immersed interface method for simulating unsteady incompressible flows on irregular domains,” *J Comput Phys*, vol. 204, no. 1, pp. 157–192, 2005, doi: 10.1016/j.jcp.2004.09.017.