



Justin du Plessis

Application of Energy and Exergy Balances to Drag
Decomposition in Unsteady Incompressible Flows

School of Aerospace, Transport and Manufacturing
Aerospace Computational Engineering

MSc

Academic Year: 2024–2025

Supervisor: Dr Drewan Sanders and Dr Tamas Józsa
August 2025



School of Aerospace, Transport and Manufacturing
Aerospace Computational Engineering

MSc

Academic Year: 2024–2025

Justin du Plessis

Application of Energy and Exergy Balances to Drag
Decomposition in Unsteady Incompressible Flows

Supervisor: Dr Drewan Sanders and Dr Tamas Józsa
August 2025

This thesis is submitted in partial fulfilment of the
requirements for the degree of MSc.

© Cranfield University 2025. All rights reserved. No part of
this publication may be reproduced without the written
permission of the copyright owner.

Academic Integrity Declaration

I declare that:

- the thesis submitted has been written by me alone.
- the thesis submitted has not been previously submitted to this university or any other.
- that all content, including primary and/or secondary data, is true to the best of my knowledge.
- that all quotations and references have been duly acknowledged according to the requirements of academic research.

I understand that to knowingly submit work in violation of the above statement will be considered by examiners as academic misconduct.

Abstract

The accurate evaluation of aerodynamic drag in unsteady flows presents an ongoing challenge for aerodynamic optimisation in tightly coupled engine-airframe configurations. This thesis investigates the use of energy and exergy balance formulations as alternative approaches for drag evaluation, with a focus on their ability to decompose drag power into recoverable and irreversible contributions. Three two-dimensional laminar flow test cases were examined using CFD simulations in ANSYS Fluent. The first, a stationary cylinder at $Re = 140$ undergoing vortex shedding, served as a benchmark and confirmed that both balances reproduce near-field drag values with high accuracy while revealing the conversion of mechanical energy into viscous dissipation along the wake. The second case, involving an oscillating cylinder in static fluid, demonstrated that the balances remain applicable even in the absence of net aerodynamic drag force, capturing the redistribution of energy through viscous effects in agreement with analytical predictions. The third case combined vortex shedding with azimuthal wall oscillation, producing a more complex flow where the balances showed a modest increase in mean drag but no fundamental changes to dissipation mechanisms. Together, these results establish the robustness of energy and exergy balances for unsteady drag evaluation and highlight their potential for usage in optimisation of aerodynamic performance.

Keywords:

Aerodynamic Drag, Unsteady Flow, Vortex Shedding, Power Balance, Energy Balance, Exergy Balance.

Acknowledgements

The Author wishes to thank Dr Drewan Sanders and Dr Tamas Józsa for their supervision and guidance throughout the project. Their expertise and support were invaluable in the completion of this research. The Author would also like to thank Antoine Germaneau for providing analytical data for validation purposes, and Dameon Westley from the University of Pretoria for his assistance with debugging the far-field balances.

Contents

Academic Integrity Declaration	i
Abstract	ii
Acknowledgements	iii
Contents	iv
List of Figures	vii
List of Tables	ix
1 Introduction	1
1.1 General Introduction	1
1.2 Objectives of the Research	4
2 Formulations used in the Research	6
2.1 Absolute and Relative Reference Frames	6
2.2 Momentum Balance	7
2.3 Energy Balance	7
2.4 Exergy Balance	8
2.5 Further Decompositions	9
3 Vortex Shedding Test Case	10
3.1 Test Case Definition	10
3.1.1 Numerical Setup	11
3.1.2 Control Volume Definition	13
3.2 Grid Study and Results Validation	14
3.3 Far-Field Results	17
3.3.1 Control Volume Accuracy Evaluation	17
3.3.2 Flow Decompositions	20
3.4 Test Case Conclusions	27
4 Oscillating Cylinder Test Case	28
4.1 Test Case Definition	28
4.1.1 Numerical Setup	29
4.1.2 Control Volume Definition	31
4.2 Near-Field Results Validation	32

4.3	Far-field Balances	35
4.4	Far-Field Decompositions	37
4.5	Test Case Conclusions	40
5	Combined Test Case	41
5.1	Test Case Definition	41
5.1.1	Numerical Setup	42
5.1.2	Control Volume Definition	42
5.2	Results	43
5.2.1	Near-Field Evaluation	44
5.2.2	Far-Field Balances	45
5.2.3	Far-Field Decompositions	47
5.2.4	Wake Analysis	50
5.3	Test Case Conclusions	51
6	Thesis Conclusions	52
	References	55
A	Ethical Approval Letter	58
B	Laminar Vortex Shedding Grid Information	60
B.1	Mesh Images	60
B.2	Mesh Quality Metrics	62
C	Oscillating Cylinder Mesh Information	63
C.1	Mesh Images	63
C.2	Mesh Quality Metrics	65
D	Combined Test Case Exergy Plots	66
D.1	CV1 Decomposition	66
D.2	CV3 Decomposition	67
D.3	CV7 Decomposition	67
E	Fluent Named Expressions	68
E.1	Momentum Balance	68
E.1.1	Steady Term	68
E.1.2	Unsteady Term	68
E.2	Energy Balance	68
E.2.1	$\mathcal{E}_{TP}$	68
E.2.2	$\frac{dEk}{dt}$	68
E.2.3	Θ	69
E.2.4	Φ	69
E.3	Exergy Balance	69
E.3.1	$\mathcal{X}_s$	69
E.3.2	$\dot{\mathcal{X}}_{th}$	69
E.3.3	$\dot{\mathcal{A}}_{VT}$	69
E.3.4	$\dot{\mathcal{A}}_{\Phi}$	69

E.3.5	$\dot{\mathcal{X}}_v(t)$ Term 2	69
-------	---	----

List of Figures

1.1	Drag Components	1
1.2	Onera Nova Boundary Layer Ingestion Concept	2
2.1	Control Volume ARF vs RRF	6
3.1	Vortex Street	10
3.2	Computational Domain Dimensions	13
3.3	Control Volumes Diagram	13
3.4	Near-Field C_d and C_l vs Flow Time	14
3.5	Grid Convergence Index for Time Averaged C_d	15
3.6	Time Averaged C_d vs Reynolds Number Validation	16
3.7	Strouhal Number vs Reynolds Number Validation	17
3.8	RMS of Error Between Near-Field and Far-Field C_d results for Each CV	18
3.9	C_d vs Time for CV1 and CV7	19
3.10	Far-Field Balance for CV1	20
3.11	Far-Field Balance for CV2	21
3.12	Far-Field Balance for CV3	21
3.13	$\mathcal{E}_{TP}$ Decomposition at CV2 and CV3	22
3.14	Kinetic Energy Contour at $t/T=1.35$	22
3.15	Kinetic Energy Contour at $t/T=1.75$	23
3.16	Transverse Energy Contour at $t/T=1.75$	23
3.17	Far-Field Balance for CV6	24
3.18	Mechanical Energy Wake Line Visualisation	25
3.19	Potential for Energy Recovery in the Wake	26
4.1	Guage Pressure around Cylinder	29
4.2	Computational Domain Dimensions	31
4.3	Control Volumes Diagram	31
4.4	Wall Radial Velocity vs Time	32
4.5	Velocity Comparison with Analytical Solution	33
4.6	Gauge Pressure Comparison with Analytical Solution	34
4.7	C_d comparison with Momentum Balance	35
4.8	$\mathcal{P}_S$ Comparison with Analytical Solution	36
4.9	Far-Field Comparison with Analytical $\mathcal{P}_S$	36
4.10	Far-Field Decomposition for CV1	37
4.11	Far-Field Decomposition for CV2	38
4.12	Far-Field Decomposition for CV3	38

4.13	Mechanical Energy Decomposition for CV1	39
4.14	Mechanical Energy Decomposition at CV2 and CV3	39
5.1	Control Volumes Diagram	43
5.2	Near-Field C_d Comparison for Case 3 (Oscillating) and Case 1 (Standard)	44
5.3	Energy and Exergy Balances for CV1	45
5.4	Energy and Exergy Balances for CV3	45
5.5	Energy and Exergy Balances for CV7	46
5.6	Energy Decomposition for CV1	47
5.7	Mechanical Energy Decomposition for CV1	48
5.8	Energy Decomposition for CV3	49
5.9	Energy Decomposition for CV7	49
5.10	$\mathcal{E}_{TP}$ Wake Analysis	50
5.11	$\mathcal{PER}$ Solid Line: Oscillating Case, Dashed Line: Standard Case	50
B.1	Mesh Image for Laminar Vortex Shedding Test Case	60
B.2	Mesh Near-Field Region for Laminar Vortex Shedding Test Case	61
B.3	Mesh Boundary Layer Region for Laminar Vortex Shedding Test Case	61
C.1	Mesh Image for Oscillating Cylinder Test Case	63
C.2	Mesh Nearfield Region for Oscillating Cylinder Test Case	64
C.3	Mesh Wall Region for Oscillating Cylinder Test Case	64
D.1	Exergy Decomposition for CV1	66
D.2	Exergy Decomposition for CV3	67
D.3	Exergy Decomposition for CV7	67

List of Tables

3.1	Numerical Solver Inputs	12
3.2	Control Volume Wake Lengths	14
3.3	GCI Study Inputs	15
3.4	Time Averaged Drag Coefficient Results	18
4.1	Numerical Solver Inputs	30
4.2	Control Volume Radii	32
5.1	Numerical Solver Inputs	42
5.2	Control Volume Wake Lengths	43
5.3	Near-Field Drag Comparison	44
5.4	Energy Balance Time Averaged C_d	46
5.5	Exergy Balance Time Averaged C_d	46
5.6	Mechanical Energy Decomposition Time Averaged Mean Values	48
B.1	Mesh Quality Metrics	62
C.1	Mesh Quality Metrics	65

Nomenclature

$\bar{\bar{\tau}}$	Viscous Stress Tensor
$\dot{E}_\tau$	Viscous Boundary Work Deposition Rate
$\dot{E}_a$	Stream-Wise Kinetic Energy Deposition Rate
$\dot{E}_p$	Pressure Boundary Work Deposition Rate
$\dot{E}_v$	Transverse Kinetic Energy Deposition Rate
$\frac{dEk}{dt}$	Rate of Change of Kinetic Energy
$\hat{n}$	Normal Vector
$\mathcal{E}_{TP}$	Mechanical Energy Deposition Rate on Wake Plane
$\mathcal{P}_s$	Propulsive Power
Φ	Viscous Dissipation Rate
Θ	Volumetric Pressure Work Rate
C_d	Drag Coefficient
C_l	Lift Coefficient
$C_{()}$	Non-dimensional Coefficient of a Quantity of Interest
p_g	Gauge Pressure
PER	Potential For Energy Recovery
T	Temperature
V	Velocity in Absolute Reference Frame
V'	Velocity in Relative Reference Frame

Chapter 1

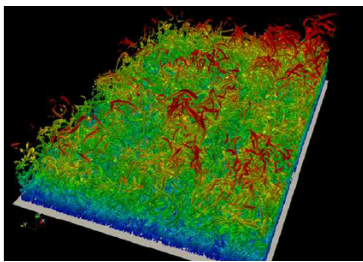
Introduction

1.1 General Introduction

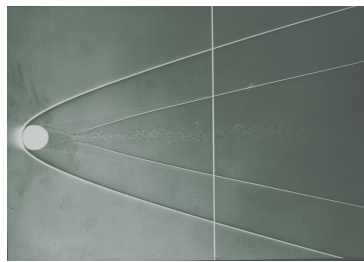
The aviation industry is undergoing a period of rapid transformation, driven by the urgent need to reduce emissions and lower the operating costs of commercial flights. While modern aircraft are significantly more efficient than earlier generations, as a consequence of large improvements in engine technology, further improvements in performance and sustainability remain a priority. Aerodynamic drag still accounts for a significant portion of the fuel consumption of an aircraft, and as such, it is a key target for future design improvements. In response, both academia and emerging aerospace companies have begun to shift away from the conventional tube-and-wing configuration. This traditional layout, although proven and reliable, is increasingly viewed as having reached its aerodynamic and performance limits.

For an aircraft, the aerodynamic drag forces can be broken down into three main types [1]:

- **Parasitic Drag:** A combination of Viscous Drag between the fluid and the surface of the aircraft (Figure 1.1a), and Profile Drag which is influenced by the shape of the aircraft or component.
- **Wave Drag:** Generated by the formation of shockwaves, due to the compressibility of air (Figure 1.1b).
- **Lift-Induced Drag:** Introduced through the formation of downwash and tip vortices as a reaction to the generation of the lift force (Figure 1.1c).



(a) Boundary Layer [2]



(b) Shock Wave Formation [3]



(c) Tip Vortex Formation [4]

Figure 1.1: Components of Drag

To achieve larger improvements in drag reduction and fuel consumption, both academia and industry have begun proposing more radical design approaches for next generation aircraft. Many of these emerging concepts feature a close integration of the propulsion system with the airframe and aim to optimise by considering the interactions of the propulsion and airframe, rather than treating them as separate systems [5, 6, 7]. Boundary Layer Ingestion (BLI) in particular has gained attention for its potential to both aid in drag reduction and improve engine efficiency, through using the engine to re-energise low momentum boundary layer air.

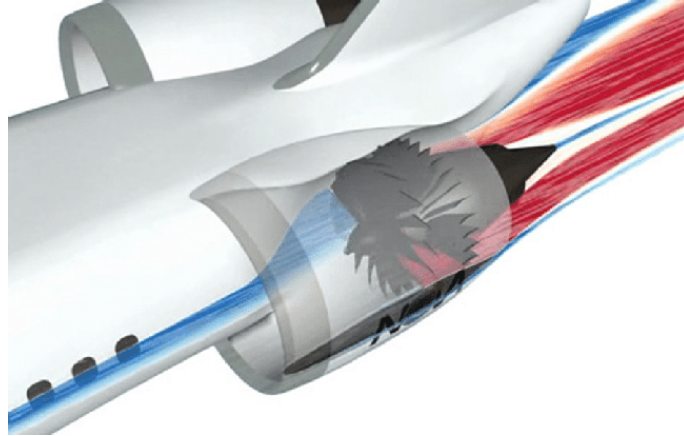


Figure 1.2: Onera Nova Boundary Layer Ingestion Concept
Obtained from Doll et al. [8]

However, these tightly coupled designs introduce new complexities in aerodynamic analysis. The interaction between propulsion and the surrounding airframe affects both the local flow-field and the global flow, making it difficult to isolate the contributions of thrust and drag [9]. While modern CFD and experimental techniques can accurately measure the total drag force acting on an aircraft, they remain limited in their ability to resolve individual drag components. This limitation is especially pronounced in tightly integrated configurations, where complex interactions between the airframe and propulsion system hinder traditional analysis methods [10].

This limitation poses a challenge for performance analysis and design optimisation. To address this, there is growing interest in alternative diagnostic methods, particularly those based on far-field control volume approaches. These methods, which rely on integral balances of momentum, energy, and exergy, offer a promising way to assess drag and energy transfer mechanisms in complex, integrated flow fields.

Destarac and van der Dooren [11] were the first to address this challenge of utilising a far-field control volume approach to separate the drag and thrust forces from one another. They proposed a momentum-based far-field method, which not only allowed for the separation of thrust and drag, but allowed them to separate the drag into its sub components. Although the framework provided a robust and technically sound method for drag analysis, it was limited to steady-state simulations, with static geometry. Furthermore the method offered very little insight into energy movement within the flow, particularly with

regards to energy transfer and dissipation. To improve on the steady state limitations of Desatrac and van der Dooren's method, Toubin [1] further developed the momentum-based methods to allow for transient formulations and the inclusion of moving geometries. Toubin's method was able to provide a more detailed analysis of the drag force, but still lacked the ability to quantify the energy transfer and dissipation within the flow field. To address this limitation, Drela developed the power balance method utilising the conservation of energy [10].

Drela's power balance expressed drag as a result of reversible and irreversible power losses within the flow field, which allowed for a more detailed analysis of the energy transfer and dissipation within the flow field. These losses, referred to as drag power are associated with mechanisms such as viscous dissipation, pressure work, and kinetic energy transfer. Furthermore, the power balance was able to separate drag and thrust components from one another and it was able to detect interference effects which were not necessarily captured in the momentum balance approach. Although Drela's power balance method represented a large step forward it still lacked the ability to represent thermal effects within the flow-field [12], which lead to Artz et al. identifying the need for a further improvement to the method, named the Exergy Balance [13].

The concept of exergy is based on the second law of thermodynamics, where exergy represents the useful portion of energy within a system [14]. To view the aerodynamic drag power in terms of this exergy quantity, Artz et al. identified that the thermal effects in form of temperature gradients and the quantification of entropy must be introduced into the analysis. Furthermore he identified that the Volumetric Pressure Work is not considered, as a stand alone term and is instead represented as a result of thermal changes within the flow-field.

Although there has been a large amount of work done in the development of the momentum, energy and exergy balances, there has been very little work done in the application of these methods to unsteady flows. The majority of the work has been done in the context of steady-state flows, with the exception of Toubin's extension of the momentum balance. The only work done in the context of unsteady flows using the energy or exergy balances has been work done by Ruscio et al. [15], where the authors performed an exergy analysis of a vortex shedding cylinder. Although their work was done in an unsteady flow, it was limited to only two sequential time steps. Furthermore, their work did not evaluate the usefulness of the energy being lost to the wake, nor did it compare the results of the exergy balance with the energy or momentum balances.

1.2 Objectives of the Research

The primary aim of this research is to assess the applicability of the energy, and exergy balance methods for drag evaluation in unsteady aerodynamics flows and to compare their accuracy against the more established momentum balance method. While the momentum method remains widely used for aerodynamic force quantification, it provides limited insight into the mechanisms of energy transfer and dissipation within the flow field. In contrast, energy and exergy-based formulations theoretically offer a deeper understanding of how energy is transformed, lost, or irreversibly dissipated, potentially revealing flow structures and losses that are not captured through momentum analysis alone.

To investigate these methods, three unsteady flow configurations are considered. The first is a classical drag analysis problem where the drag is dominated by a periodically unsteady wake, while the second test case explores a situation where no net force or wake is generated allowing the study of energy and exergy transport in the absence of classical definitions of thrust or drag. Finally the third case explores the effects of a moving geometry on a unsteady flow-field by combining test case one and two together.

Test Case 1: Vortex Shedding Cylinder

The first case focuses on a two-dimensional cylinder undergoing unsteady vortex shedding at a Reynolds number of $Re = 140$. This canonical case features periodic drag fluctuations and well-understood wake dynamics, making it ideal for testing and validating the methods. The objectives of this case are to:

- Apply the momentum, energy, and exergy balance methods to the unsteady cylinder wake and compare the accuracy of their drag predictions.
- Extend the exergy analysis conducted by Ruscio et al. [15] to include the energy balance, and to perform the analysis with respect to the periodic nature of the flow, rather than just two sequential time steps.
- Quantify the amount of energy that is transferred in the wake and evaluate the usefulness of this energy in terms of its ability to perform work or contribute to irreversible loss.

Test Case 2: Oscillating Cylinder

The second case examines a cylinder oscillating azimuthally in a static fluid with no freestream velocity. The motion is prescribed as a sinusoidal oscillation and generates unsteady flow features entirely due to wall motion. This configuration produces zero net aerodynamic force over time, posing a challenge to conventional force-based methods. The objectives of this case are to:

- Demonstrate the capability of energy and exergy balance methods to analyse and interpret flow behaviour in the absence of net drag force.

- Investigate how energy is introduced, redistributed, and dissipated within the flow-field due to body-induced motion.
- Assess the utility of the exergy framework in identifying irreversible losses and entropy-generating mechanisms in flows where classical drag definitions are no longer applicable.

Test Case 3: Combined Case

The third case investigates a two-dimensional cylinder that both oscillates azimuthally and experiences unsteady vortex shedding in a freestream flow. This configuration combines the wake dynamics of Test Case 1 with the body-induced motion of Test Case 2, resulting in a flow field where unsteady periodic wake structures interact with motion-driven near-wall effects. By combining these two mechanisms, the case aims to produce a more complex and coupled unsteady aerodynamic flowfield. The objectives of this case are to:

- Apply the momentum, energy, and exergy balance methods to a flow field where wake-induced and motion-induced unsteadiness coexist.
- Explore how these interactions influence irreversible loss mechanisms, and determine whether the combined case amplifies or suppresses recoverable energy compared to the individual cases.

Together, these three test cases provide a detailed study to evaluate the strengths and limitations of energy and exergy balances compared to the momentum method. The research aims to determine whether these energy and exergy methods can provide a more comprehensive understanding of unsteady aerodynamic flows, in both traditional unsteady flow scenarios and in cases where the energy transfer mechanisms are not directly linked to classical definitions of drag.

Chapter 2

Formulations used in the Research

2.1 Absolute and Relative Reference Frames

In the context of this research, the formulations are applied within the Absolute Reference Frame (ARF), rather than the Relative Reference Frame (RRF) which is typically used within aerodynamic analysis. The ARF is defined as a fixed position at which the observer is stationary and viewing the object of interest moving through space at velocity V_∞ , whereas in the RRF the observer is in plane with the object of interest and views the fluid moving past the object at a speed of V_∞ . This principle is illustrated in Figure 2.1.

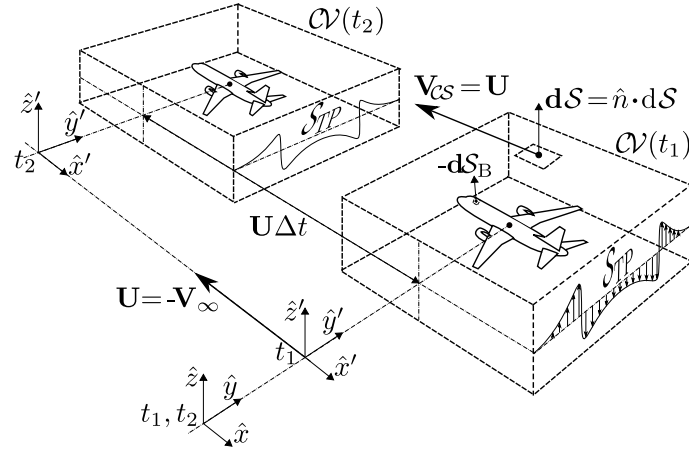


Figure 2.1: Control Volume ARF($\hat{x}, \hat{y}, \hat{z}$) vs RRF($\hat{x}', \hat{y}', \hat{z}'$)
Adapted from [16]

The reference frame change enables a more physically meaningful analysis of the flow field when using the energy and exergy methods. It allows for a direct evaluation of work done by the object of interest on the surrounding fluid within the control volume. The change from the RRF to the ARF is achieved through the usage of a velocity shift, where the fluid velocity at any point of the control volume (V) is defined as the velocity in the RRF (V') minus the freestream velocity of the fluid (V_∞). This is expressed mathematically in Equation 2.1.

$$V = V' - V_\infty \quad (2.1)$$

The shift is only required for the velocity term, as the other terms are Galilean invariant, meaning that they are not altered with the change in reference frame. The velocity shift is applied to the momentum, energy, and exergy balance, which are detailed in the subsections 2.2-2.5.

2.2 Momentum Balance

As mentioned in the previous sections, the momentum balance has been included in the study to provide a reference point for evaluating the energy and exergy balances. The momentum balance shown in equation 2.2, where the left hand side represents the drag calculated using the forces on the body (Near-Field), and the right hand side represents the Far-Field balance in the ARF.

$$\underbrace{\iint (p_g \hat{n} - \bar{\tau} \cdot \hat{n}) dS_B}_{\text{Near-Field}} = \underbrace{\iint (\rho \mathbf{V} \mathbf{V}' \cdot \hat{n} + p_g \hat{n} - \bar{\tau} \cdot \hat{n}) dS_{TP}}_{\text{Steady Term}} - \underbrace{\frac{d}{dt} \iiint \rho \mathbf{V} d\mathcal{V}}_{\text{Unsteady Correction Term}} \quad (2.2)$$

The Far-Field part of the balance is comprised of two terms, the first term is the steady term, which is the integral of the momentum flux across the wake plane (S_{TP}). The second term is the unsteady correction term, which accounts for the change in momentum within the control volume over time, this term is ignored in a steady-state simulation but necessary to account for fluctuations in the flow field when a transient simulation is performed.

2.3 Energy Balance

The energy balance, which is based on the conservation of energy, is the first of the two primary formulations evaluated within the study. The energy balance differs to the momentum balance in that it is not a direct evaluation of the drag force, but rather an evaluation of the energy transfer between the object of interest and the surrounding fluid, in the form of Drag Power (W). The balance is shown in equation 2.3, which again features the Near-Field formulation on the left hand side and the Far-Field formulation on the right hand side.

$$\underbrace{- \iint (p_g \mathbf{V}'_{cs} - \bar{\tau} \cdot \mathbf{V}'_{cs}) \cdot d\mathbf{S}_B + \iiint \mathcal{P}_f \mathbf{f} \cdot \mathbf{V}' d\mathcal{V}}_{\mathcal{P}_s} + \underbrace{\iint (p_g \mathbf{V}_\infty - \bar{\tau} \cdot \mathbf{V}_\infty) \cdot d\mathbf{S}_B + \iiint \mathcal{P}_f \mathbf{f} \cdot \mathbf{V}_\infty d\mathcal{V}}_{-F_B \cdot U} = \underbrace{\frac{d}{dt} \iiint \left(\frac{\mathbf{V}^2}{2} \rho \right) d\mathcal{V}}_{d\dot{E}k/dt} + \underbrace{\iint \left(\frac{\mathbf{V}^2}{2} \rho \mathbf{V}' + p_g \mathbf{V} - \bar{\tau} \cdot \mathbf{V} \right) \cdot d\mathbf{S}_{TP}}_{\dot{E}_{TP}} + \underbrace{\iiint -p_g \nabla \cdot \mathbf{V} d\mathcal{V}}_{\Theta} + \underbrace{\iiint \bar{\tau} : \nabla \mathbf{V} d\mathcal{V}}_{\Phi} \quad (2.3)$$

The Near-Field portion of the balance is comprised of four terms. The first two terms represent the power input of a propulsor within the control volume, where $\mathcal{P}_s$ is the power input from a physically represented propulsor, whilst $\mathcal{P}_f$ is the power input from source

term modelled propulsor. The drag power on the body is represented by the third term, $-F_B \cdot U$, while the final term $(-F_f \cdot U)$ represents any source terms not related to propulsion.

In the far-field portion of the balance, the drag power is represented as a combination of the Mechanical Energy work rate on the boundaries of the CV ($\mathcal{E}_{TP}$), the unsteady kinetic energy rate of change ($d\dot{E}_k/dt$), the volumetric pressure work rate (Θ), and the viscous dissipation rate (Φ).

2.4 Exergy Balance

Based on the second law of thermodynamics, the exergy balance is an extension of the energy balance, which considers the thermal effects within the control volume. The left hand side of the expression is extended to include the effects of exergy deposition to the fluid via thermal conduction ($\dot{\mathcal{X}}_q$) and heating effects in the volume ($\dot{\mathcal{X}}_{\dot{q}}$). The full balance is shown in equation 2.4.

$$\begin{aligned}
 & \overbrace{\mathcal{P}_s + \mathcal{P}_f - F_B \cdot U - F_f \cdot U - \underbrace{\iint \left(1 - \frac{T_\infty}{T}\right) \mathbf{q} \cdot d\mathbf{S}_B}_{\dot{\mathcal{X}}_q} - \underbrace{\iiint \left(1 - \frac{T_\infty}{T}\right) \rho \dot{q} d\mathcal{V}}_{\dot{\mathcal{X}}_{\dot{q}}}}^{\text{Near-Field}} = \\
 & \underbrace{\frac{d\dot{E}_k}{dt} + \frac{d}{dt} \iiint \rho \mathcal{X}_s d\mathcal{V}}_{\dot{\mathcal{X}}_{v(t)}} + \underbrace{\iint \rho \mathcal{X}_s \mathbf{V}' \cdot d\mathbf{S}_{TP}}_{\dot{\mathcal{X}}_{th}} + \underbrace{\dot{\mathcal{E}}_{TP}}_{\dot{\mathcal{X}}_m} + \underbrace{\iiint \frac{T_\infty}{T} (\bar{\boldsymbol{\tau}} : \nabla \mathbf{V}) d\mathcal{V}}_{\dot{\mathcal{A}}_\Phi} + \underbrace{\iiint T_\infty \mathcal{K} \left(\frac{\nabla T}{T} \right) d\mathcal{V}}_{\dot{\mathcal{A}}_{\nabla T}} \\
 & \hspace{15em} (2.4)
 \end{aligned}$$

The right hand side of the balance also includes thermal modifications to some of the energy terms. $\dot{\mathcal{X}}_{v(t)}$ exists as an extension of $d\dot{E}_k/dt$ by including the mass specific internal energy and losses to entropy. The static mass specific exergy ($\mathcal{X}_s$) is calculated with equation 2.5.

$$\mathcal{X}_s = (e - e_\infty) - T_\infty(s - s_\infty) + p_\infty \left(\frac{1}{\rho} - \frac{1}{\rho_\infty} \right) \quad (2.5)$$

The viscous dissipation term is modified to include the temperature ratio between the freestream flow and the local fluid, this modification allows for the calculation of the anenergy portion of the viscous dissipation ($\dot{\mathcal{A}}_\Phi$), where the anenergy represents the irreversibly lost component of it. Additionally the right hand side of the balance includes the energy that is irreversibly lost to thermal conduction within the fluid volume ($\dot{\mathcal{A}}_{\nabla T}$), and the static exergy ($\dot{\mathcal{X}}_{th}$), passing through the wake plane (dS_{TP}).

2.5 Further Decompositions

The energy and exergy balances can be further decomposed to provide a more detailed analysis of the energy and exergy transfer within the fluid. The mechanical energy term ($\mathcal{E}_{TP}$) can be decomposed into its kinetic energy (E_k), pressure boundary work (E_p), and viscous work (E_τ) components as shown in equation 2.6. Furthermore, the kinetic energy term can be split into its transverse component (E_v) and stream-wise component (E_a), as shown in the same equation.

$$\dot{\mathcal{E}}_{TP} = \iint \left(\underbrace{\frac{u^2}{2} \rho u'}_{E_a} + \underbrace{\frac{(v+w)^2}{2} \rho u'}_{E_v} + \underbrace{p_g u}_{E_p} - \underbrace{(\bar{\tau} \cdot \mathbf{V}) \cdot \hat{n}}_{E_\tau} \right) dS_{TP} \quad (2.6)$$

Similarly the fluctuations in kinetic energy (dE_k/dt) can be decomposed into its stream-wise (dE_a/dt) and transverse (dE_v/dt) components as shown in equation 2.7.

$$\frac{dE_k}{dt} = \frac{d}{dt} \iiint \underbrace{\frac{\rho u^2}{2}}_{dE_a/dt} + \underbrace{\frac{\rho(v^2+w^2)}{2}}_{dE_v/dt} d\mathcal{V} \quad (2.7)$$

To quantify the amount of usable energy that is lost to the flow field, Antz et al. developed the exergy waste coefficient (EWC) [13], defined as the ratio between the physical exergy ($\mathcal{X}_{th} + \mathcal{X}_m$) in the system and the total exergy and anergy in the system. While the EWC allowed for a quantification of the exergy wastage, it assumed all exergy can be utilised without impacting the source of the exergy formation. This was identified by Sanders and Laskaridis [17], who introduced the potential for energy recovery metric (PER), which changed it to a ratio between the irreversibly lost viscous dissipation and the total energy within the system. Later Lamprakis et al. [16] extended it into the improved PER ($\mathcal{PER}$), where the pressure work was included in the irreversibly lost energy. Equation 2.8 shows a modification of the *PER* which has been extended to include the unsteady kinetic energy term (dE_k/dt) as well.

$$\mathcal{PER} = 1 - \frac{\Phi + \Theta}{\frac{dE_k}{dt} + \mathcal{E}_{TP} + \Phi + \Theta} \quad (2.8)$$

A similar relationship can be established using the exergy balance quantities as shown in equation 2.9.

$$\mathcal{PER}_{\mathcal{X}} = 1 - \frac{\mathcal{A}_\Phi + \mathcal{A}_{\nabla T}}{\mathcal{X}_{v(t)} + \mathcal{X}_{th} + \mathcal{X}_m + \mathcal{A}_\Phi + \mathcal{A}_{\nabla T}} \quad (2.9)$$

Chapter 3

Vortex Shedding Test Case

3.1 Test Case Definition

The test case considers two-dimensional, incompressible flow past a stationary circular cylinder at a Reynolds number of $Re = 140$. At this Reynolds number, the flow separates symmetrically along the upper and lower surfaces of the cylinder, leading to the formation of a well-defined von Kármán vortex street, visualised in Figure 3.1.



Figure 3.1: Vortex Street at $Re = 140$

This unsteady laminar flow regime is characterized by the periodic shedding of vortices from the cylinder surface, resulting in sinusoidal oscillations of the drag coefficient about a constant mean value.

Owing to its periodicity and the absence of turbulence or external forcing, the case provides an ideal platform for evaluating far-field drag estimation methods. It allows not only for a comparison of their accuracy, but also for a deeper investigation into their ability to decompose the flow field in terms of energy transport and dissipation. Although the case has previously been studied by Ruscio et al. [15] using the exergy decomposition method, that work was limited to evaluating the balance at two discrete time steps and did not assess the full periodic behaviour of the system.

The current study extends that work by examining the entire periodic cycle of vortex shedding using not only the exergy balance but also the energy and momentum balances.

These analyses are used to assess the accuracy of each far-field method in replicating the near-field drag coefficient, and to gain insight into the unsteady transport of energy and exergy within the wake. In particular, the decomposition results are used to quantify the useful energy lost to viscous dissipation and pressure work, and to estimate the potential for energy recovery ($\mathcal{PER}$) from the unsteady flow.

As the flow-field in this case is purely a result of Drag Power and does not feature any propulsion or source terms, the near-field portions of the energy and exergy balances are simplified to only include the drag power as shown in equations 3.1 and 3.2 respectively.

$$F_B \cdot V_\infty = \frac{dE_k}{dt} + \mathcal{E}_{TP} + \Theta + \Phi \quad (3.1)$$

$$F_B \cdot V_\infty = \dot{\mathcal{X}}_{th} + \dot{\mathcal{X}}_{v(t)} + \dot{\mathcal{X}}_m + \dot{\mathcal{A}}_\Phi + \dot{\mathcal{A}}_{VT} \quad (3.2)$$

For comparison purposes the values in this section are presented as non-dimensional coefficients. The lift and drag force values they are presented as the lift coefficient (C_l) and drag coefficient (C_d). For comparison with the energy and exergy drag powers, and their subsequent components, the values are nondimensionalised using equation 3.3, where $()$ is the quantity of interest.

$$C_{()} = \frac{()}{\frac{1}{2}\rho V_\infty^3 S} \quad (3.3)$$

3.1.1 Numerical Setup

The incompressible unsteady Reynolds-Averaged Navier-Stokes (URANS) equations were solved using ANSYS Fluent 2024R2. Given the test case operates at a Reynolds number of 140, compressibility effects are considered negligible, therefore, Fluent's pressure-based solver was used. Although the flow is in a laminar flow regime, the $k - \omega$ SST turbulence model was utilised, to follow the methodology utilised by Rusio et al. [15]. Due to the numerical study requiring the development of steady oscillations in lift and drag coefficients, resulting from a consistent vortex shedding pattern, a significant amount of flow time is needed to be simulated before a meaningful analysis can begin. In this case, approximately 9 seconds of simulated flow time was required for the flow to reach a fully periodic state. To improve computational efficiency during this transient simulation, a non-iterative time advancement method was utilised.

The non-iterative time advancement method differs from its traditionally used counterpart (iterative time advancement) in that it does not attempt to eliminate time stepping error from operator splitting, instead the method only aims to reduce the splitting error to the same order as the truncation error from the computational grid [18]. This approach allows for a greater computational efficiency as only one inner iteration is required per time step. The method is particularly useful in cases where a small time-step size is utilised and the limiting factor in accuracy tends to be the computational grid.

When using Fluent's non-iterative time advancement (NITA) scheme, two pressure-velocity coupling algorithms are available: the Pressure-Implicit with Splitting of Operators (PISO)

method and the Fractional Step Method (FSM). The FSM, originally developed as an improvement to Chorin’s projection method [19], is an operator-splitting approach designed to solve the pressure–velocity coupling in incompressible flows efficiently. The method proceeds by first solving an intermediate velocity field, temporarily neglecting the pressure gradient in the momentum equation. This intermediate velocity is then used to solve a pressure Poisson equation, which updates the pressure field. Finally, the velocity field is corrected using the updated pressure. This method is particularly well-suited for incompressible flow problems, as it enables accurate pressure and velocity predictions without the need for inner iterations [20]. As the pressure Poisson equation used in FSM enforces a divergence-free velocity field, the method is fundamentally suited to incompressible (or very weakly compressible) flows and is not applicable to compressible regimes.

To avoid artificially dampening out the transient flow features, which are needed for the detailed far-field analysis, a second order discretisation was utilised for both the spacial discretisation and the transient formulation, with a time step size of 1.4×10^{-4} s. The solver setup has been summarised in Table 3.1.

Table 3.1: Numerical Solver Inputs

Solver Parameter	Input
Solver Type	Pressure Based
Time Advancement	Non-Iterative Time Advancement
Pressure Velocity Coupling Scheme	Fractional Step
Transient Formulation	Second Order Implicit
Gradient	Least Squares Cell Based
Pressure Discretisation	Second Order
Density Discretisation	Second Order Upwind
Momentum Discretisation	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind
Energy	Second Order Upwind

The computational domain was defined as a two-dimensional rectangular domain, with a cylinder of diameter $0.01m$. The boundary condition on the edge of the domain were set as a pressure far-field condition with the inputs set to match a Reynolds Number of 140 by utilising a mach number constraint, whilst the cylinder was defined as a stationary no-slip wall. The domain’s dimensions are shown in Figure 3.2, where the lengths are represented as a multiple of the cylinder diameter (D).

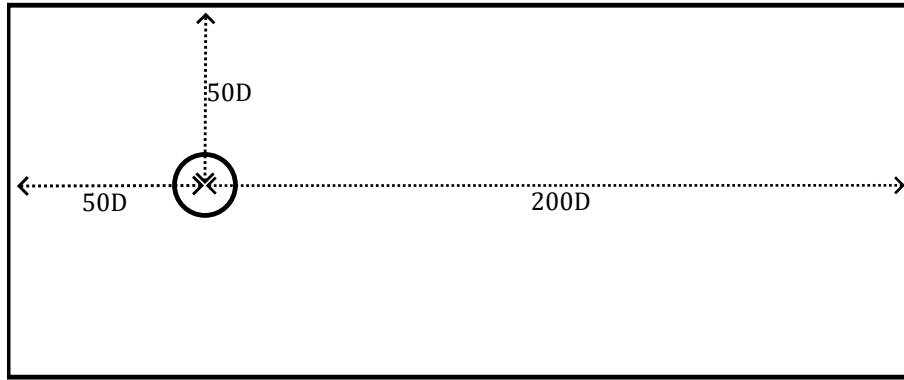


Figure 3.2: Computational Domain Dimensions
Diagram not to Scale

3.1.2 Control Volume Definition

For implementing the far-field balances and decompositions, a series of control volumes (CV) were defined around the cylinder. The control volumes were defined as rectangular prisms with the edge surfaces used as the wake planes for the surface integrals. The control volumes were defined as shown in Figure 3.3, where the upstream, top and bottom surfaces were defined at a distance of 10 diameters from the center of the cylinder.

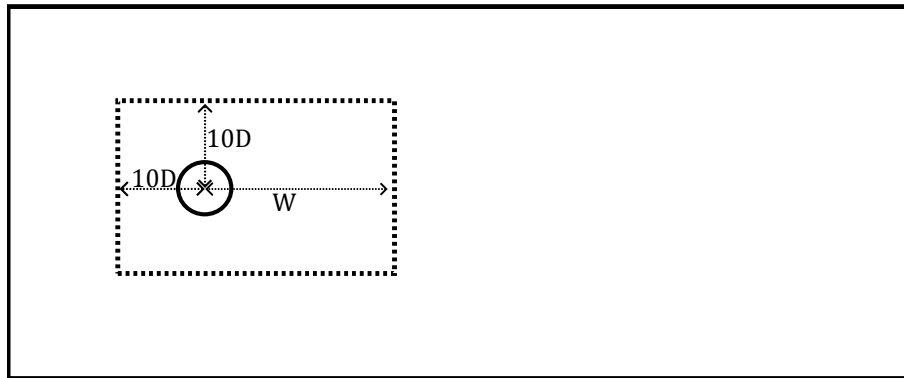


Figure 3.3: Control Volumes Diagram
Diagram not to Scale

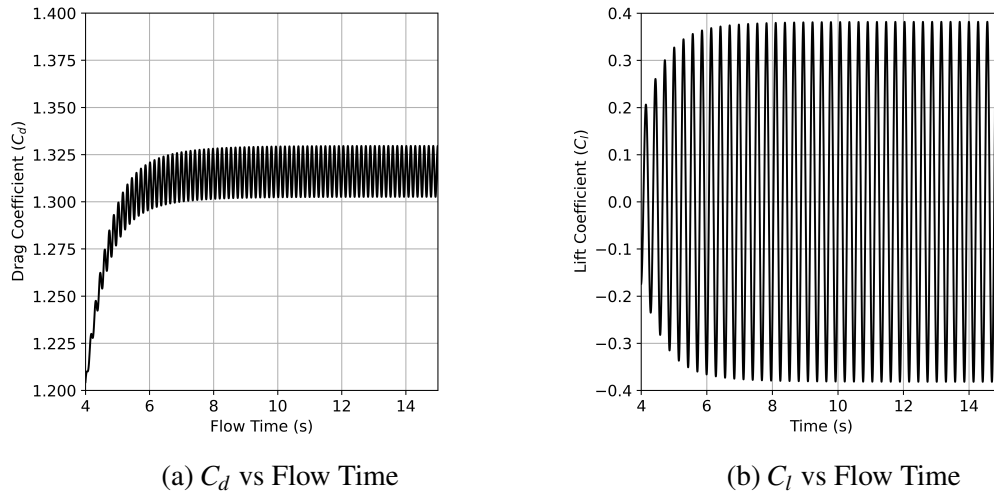
The usage of multiple control volumes allows for the evaluation of how their size affects the results of the drag calculations and also allows for evaluation of how the energy is transferred and dissipated as the distance from the cylinder increases. For these reasons the downstream portion of the control volumes were defined at seven different lengths (W in Figure 3.3), where the lengths are defined as multiples of the cylinder diameter (D) in Table 3.2.

Table 3.2: Control Volume Wake Lengths

Control Volume Name	W
CV1	2D
CV2	5D
CV3	10D
CV4	15D
CV5	20D
CV6	30D
CV7	50D

3.2 Grid Study and Results Validation

Before evaluating the results of the far-field balances, it is important to evaluate the accuracy of the numerical simulation and to establish grid independence and minimise error due to truncation effects. Before the drag coefficient (C_d) can be evaluated, the simulation needed to reach a periodic state with a well-defined vortex shedding pattern and fully developed flow field. To judge whether the simulation has reached a periodic state, the C_d and lift coefficient (C_l) were monitored over time, as shown in Figure 3.4. The figure shows after 9 seconds of simulated flow time, both the C_d and C_l values have reached a periodic state with the oscillations occurring with constant amplitude and frequency.

Figure 3.4: Near-Field C_d and C_l vs Flow Time

A structured grid was used for the simulations to allow for precise control over the mesh quality and resolution in the wake region. The grid convergence study was conducted using the time averaged C_d value for three levels of grid refinement. The C_d averaging was conducted from 10 to 15s for all of the simulations. The coarsest grid had a cell count of 108660, the medium grid had a cell count of 294820, and the finest grid had a cell count of 664100 cells. The time averaged C_d values for the three grids were plotted in Figure 3.5, where the results were used to calculate the richardson's exrtapolated value. The results are plotted against the normalised cell count, where the normalised cell count

is defined as the ratio of the cell count of the current grid to the cell count of the finest grid. With the finest grid having a normalised cell count of 1 and the coarsest grid having a normalised cell count of 6.11.

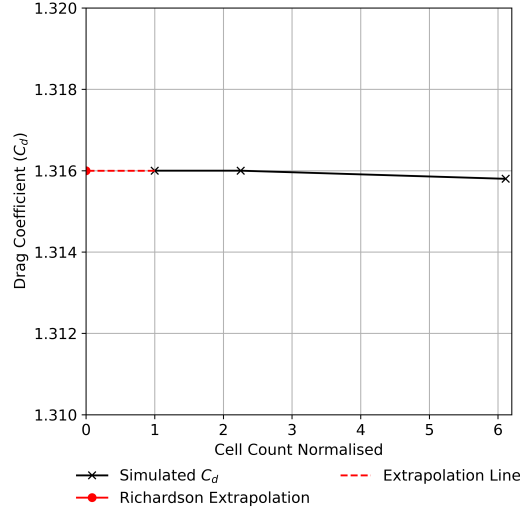


Figure 3.5: Grid Convergence Index for Time Averaged C_d

Looking at the results in Figure 3.5, it can be seen that the time averaged C_d value undergoes a monotonic convergence as the grid is refined, with the coarsest grid having a C_d value of 1.3158 and the medium and finest grid obtain a C_d value of 1.3160. The richardson's extrapolated value was calculated to be 1.3160 suggesting that there will be no further significant change in the C_d value with further grid refinement. For this reason it was decided to use the finest grid as the increased cell count would ensure a significantly reduced truncation error in the wake region where the control volumes are defined. Further details on the mesh GCI are seen in Table 3.3, while information on the quality and images of the mesh are available in Appendix B.

Table 3.3: GCI Study Inputs

Parameter	Value Obtained
Grid 1 Cell Count	108660
Grid 2 Cell Count	294820
Grid 3 Cell Count	664100
Grid 1 Averaged Cd	1.358
Grid 2 Averaged Cd	1.360
Grid 3 Averaged Cd	1.360
r_{32}	1.65
r_{21}	1.5
Convergence Ratio	0.02
Convergence Condition	Monotonic
Asymptotic Range	0.889

To further evaluate the accuracy of the numerical setup and grid combination the time averaged C_d value obtained from the simulation was compared with the data from the study by Ruscio et al. [15] and with experimental data from Roshko [21]. This is presented in Figure 3.6, where it can be seen there is a 0.2% difference to the C_d value obtained by Ruscio et al. and a 3.6% difference to the experimental data from Roshko.

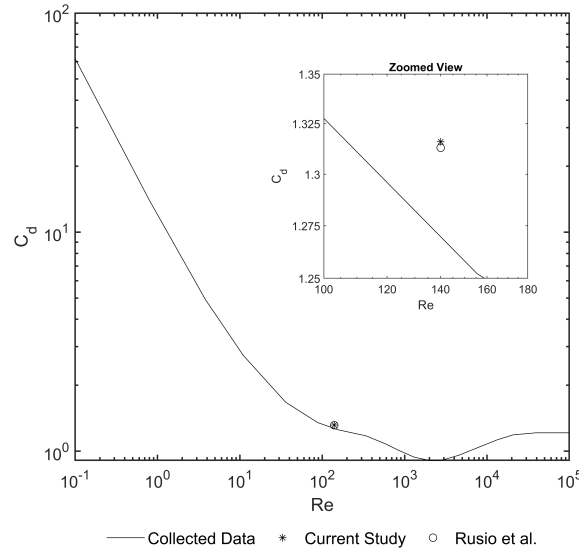


Figure 3.6: Time Averaged C_d vs Reynolds Number Validation
Validation data from Ruscio et al. [15] and Roshko [21]

Further validation of the setup was done by comparing the Strouhal number (St) obtained from the simulation with the data from various experimental and numerical studies. The Strouhal number was calculated using the frequency of the vortex shedding, obtained from the oscillatory period of the C_l vs time results (Figure 3.4b). The resulted in a strouhal number of 0.172, this is presented in Figure 3.7, where it can be seen that the value lies within the range of data from multiple sources.

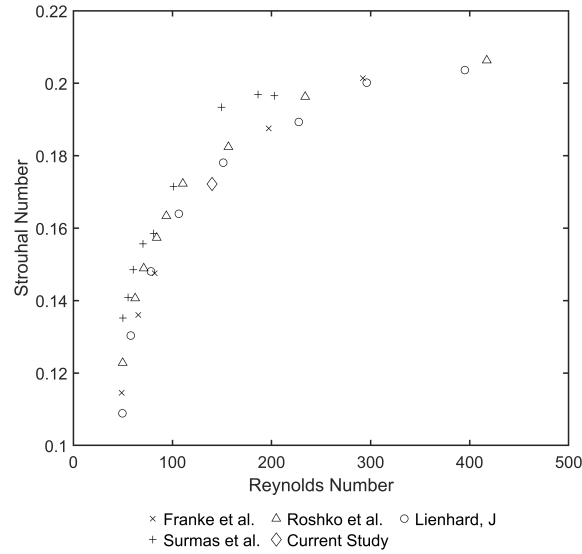


Figure 3.7: Strouhal Number vs Reynolds Number Validation
Validation data from multiple sources [21, 22, 23, 24]

Overall the grid convergence study and validation results provide a good level of confidence in the numerical setup and its ability to accurately capture the flow physics of the test case.

3.3 Far-Field Results

To evaluate the effectiveness of the far-field analysis methods introduced earlier, the results are organised into two distinct subsections. First, a control volume accuracy study is presented (Subsection 3.3.1), which compares the accuracy of each method and examines the influence of control volume size by benchmarking against the near-field result. This is followed by a detailed flow field analysis at distinct locations, based on the control volumes, using the mathematical decompositions of the energy and exergy balances (Subsection 3.3.2). Finally, the flow field analysis concludes with an study of the wake region, focusing on how the distance from the cylinder affects mechanical energy deposition and the potential for energy recovery.

3.3.1 Control Volume Accuracy Evaluation

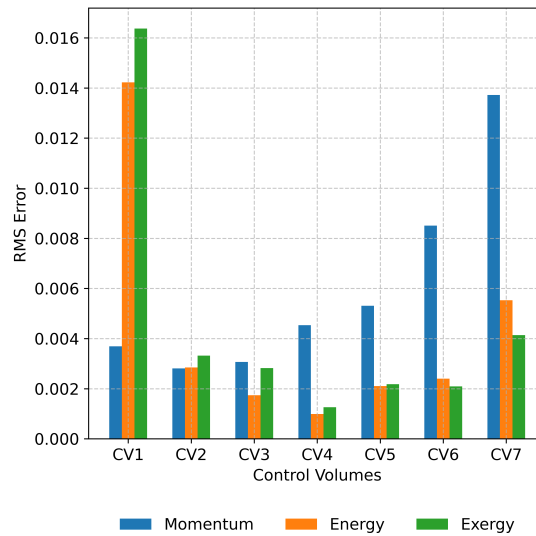
To compare the accuracy of the far-field balances and the control volume size's, the drag coefficient (C_d) results from the near-field and far-field balances were averaged averaged over 5 seconds of flow time, from 10 to 15 seconds. The results are presented in Table 3.4, where the near-field C_d value is compared with the far-field C_d values obtained from each control volume (CV) for each far-field method.

Table 3.4: Time Averaged Drag Coefficient Results

Nearfield C_d	Control Volume	Momentum C_d	Energy C_d	Exergy C_d
1.3160	CV1	1.3175	1.3018	1.2997
	CV2	1.3148	1.3160	1.3166
	CV3	1.3143	1.3170	1.3180
	CV4	1.3130	1.3164	1.3172
	CV5	1.3114	1.3156	1.3165
	CV6	1.3082	1.3142	1.3152
	CV7	1.3028	1.3105	1.3252

As shown in Table 3.4, outside of CV1, the momentum-based far-field method consistently underestimates the drag coefficient relative to the near-field value. In contrast, the energy and exergy-based methods do not exhibit a clear trend of under- or overestimation across control volumes. To quantify the accuracy, the percentage error relative to the near-field result was calculated for each method and control volume. The largest error for the momentum method occurs in CV7, with an underestimation of approximately 1.0%. For the energy and exergy methods, the largest deviation is found in CV1, at roughly 1.1% underestimation of energy and 1.2% underestimation with the exergy balance.

Overall all three methods show a good level of accuracy, when averaging the C_d results over time, but the comparison neglects the transient and periodic nature of the flow. To further evaluate the accuracy of the methods, the amount of error between the near-field and far-field C_d results was calculated at each time step and from this the root mean square (RMS) of the error was calculated. The RMS of the error is presented in Figure 3.8.

Figure 3.8: RMS of Error Between Near-Field and Far-Field C_d results for Each CV

As shown in Figure 3.8, the largest RMS errors occur in CV1 for the energy and exergy methods, and in CV7 for the momentum method. Overall, the energy and exergy balances exhibit lower RMS errors across most control volumes compared to the momentum

method, indicating greater accuracy in capturing the unsteady drag signal. Notably, the momentum method shows a clear trend of increasing error with control volume size, suggesting it is more sensitive to truncation errors as a result of grid stretching in the far-field than the energy and exergy methods. To better understand the differences in the results, the C_d vs time results for CV1 and CV7 are presented in Figure 3.9a and Figure 3.9b respectively.

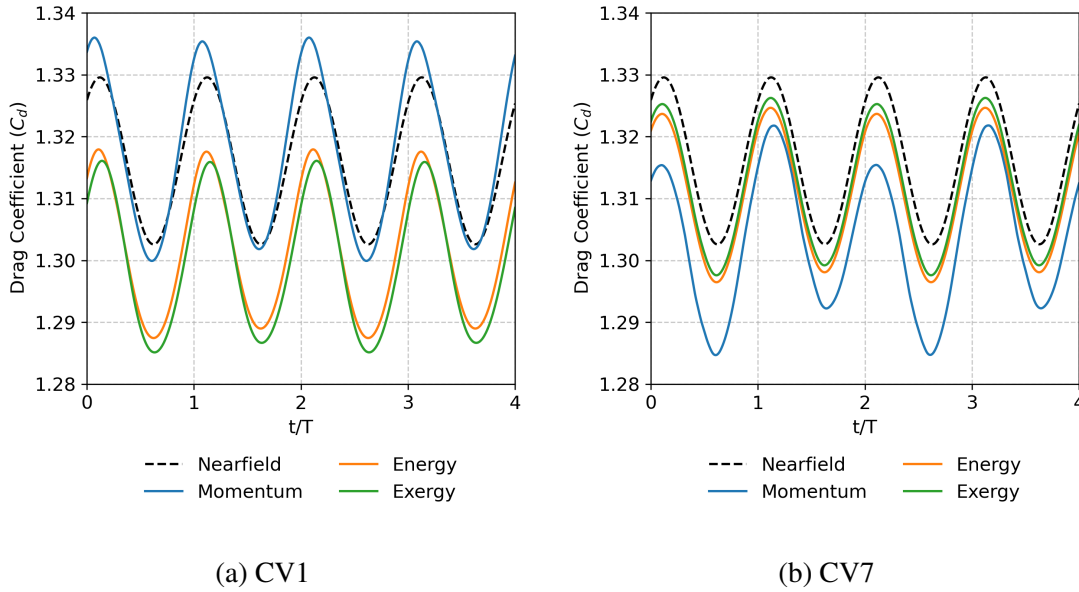


Figure 3.9: C_d vs Time for CV1 and CV7

Figure 3.9a shows that at CV1, both the energy and exergy methods consistently underestimate the drag coefficient (C_d) throughout the oscillation cycle, with only minor differences between them. In contrast, the momentum method oscillates around a mean value that closely matches the near-field C_d , albeit with a noticeably larger amplitude. Importantly, all three far-field methods maintain a phase alignment with the near-field signal, indicating no significant phase shift. This trend remains consistent at greater distances from the cylinder, as seen in Figure 3.9b, where the far-field methods continue to follow the phase of the near-field C_d . However, at CV7 the momentum method not only underestimates the drag coefficient but also exhibits a clear change in oscillation behaviour, splitting into two repeating peaks per cycle. This is not observed in the energy and exergy methods, which continue to display a single, consistent oscillation frequency.

Overall the results show that the energy and exergy methods are less sensitive to the control volume size and truncation errors than the momentum method. Although the errors are noted to be larger with momentum in most of the control volumes, it is still important to note that the error is still within a reasonable range, with the largest error being only being 1% for the momentum method in CV7.

3.3.2 Flow Decompositions

The main strength of the energy and exergy-based far-field methods is their ability to decompose the flow-field into its different energy transfer and dissipation mechanisms. This allows for a detailed understanding of the underlying physics dictating the the behaviour and formation of the vortex street and the drag. The decomposition for using the energy and exergy balances for CV1 are shown in Figure 3.10. It is important to note the exergy results are near identically to the energy results, this is a result of the low Reynolds number of the flow, which means that there is very little thermal energy transfer within the system, and its is almost all localised around the wall.

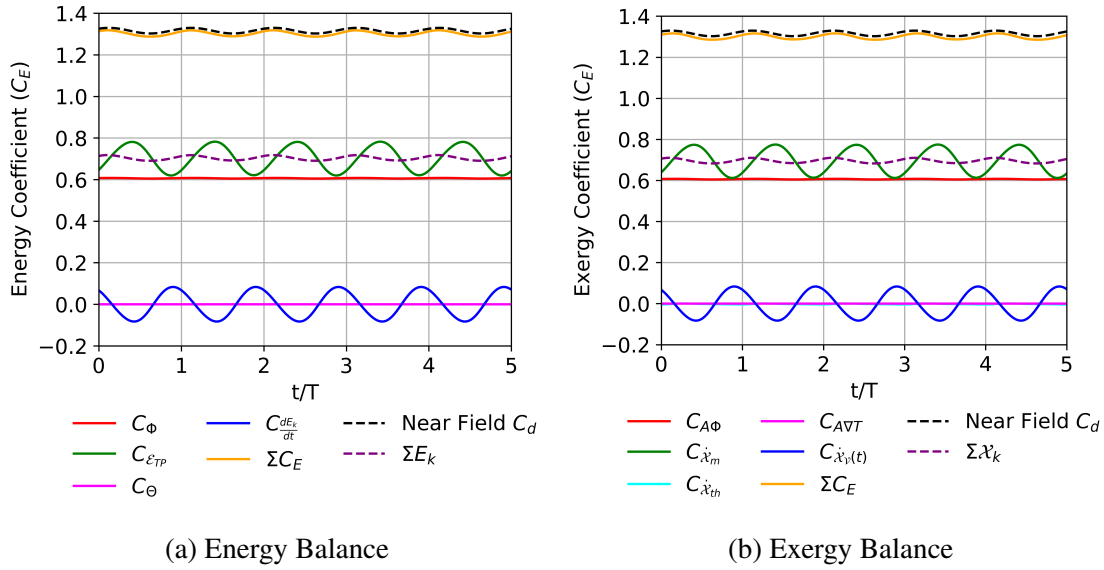
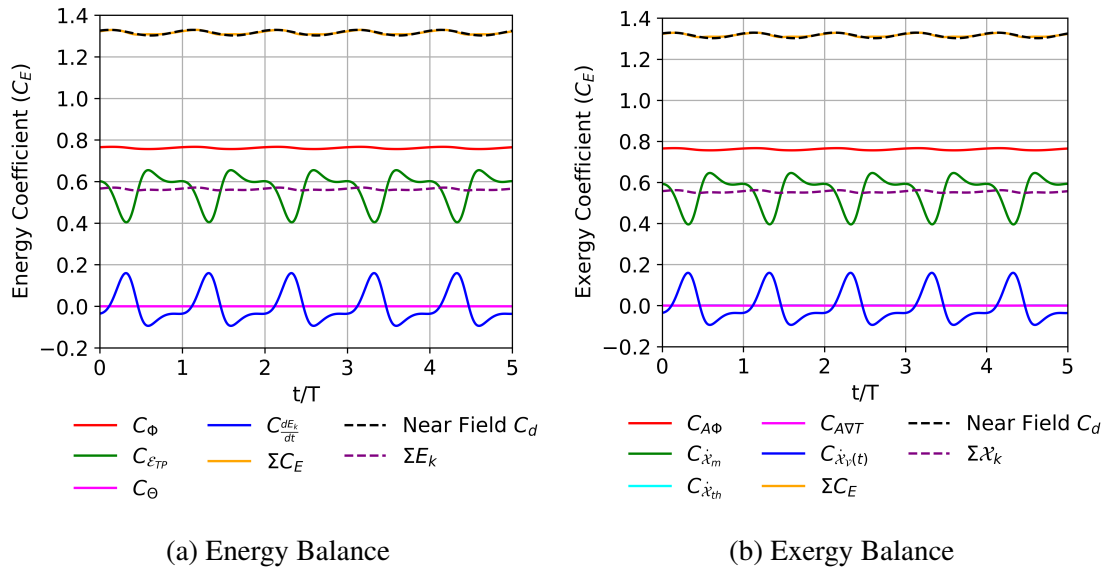
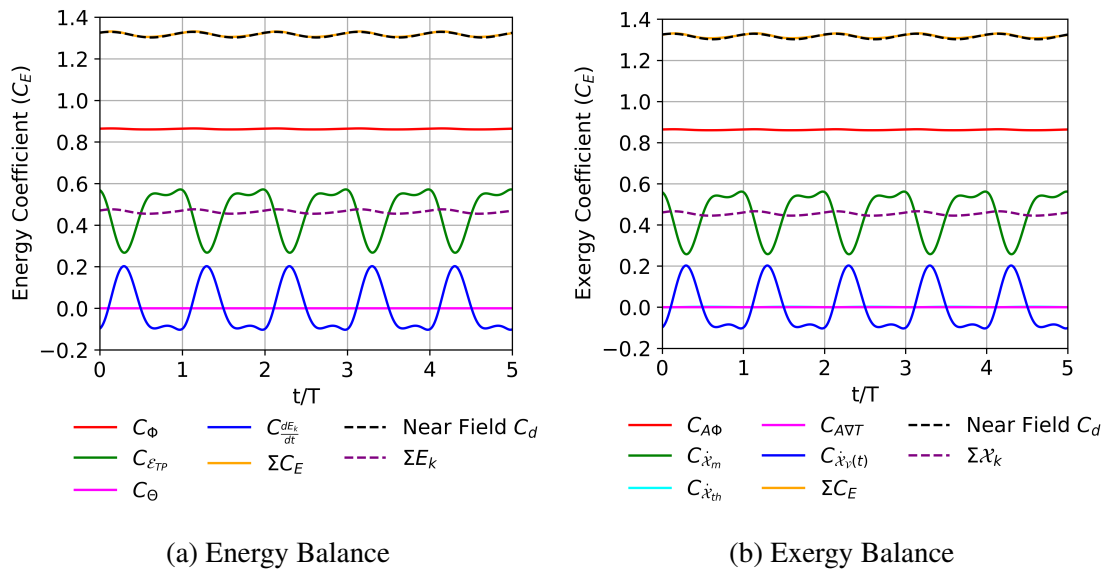


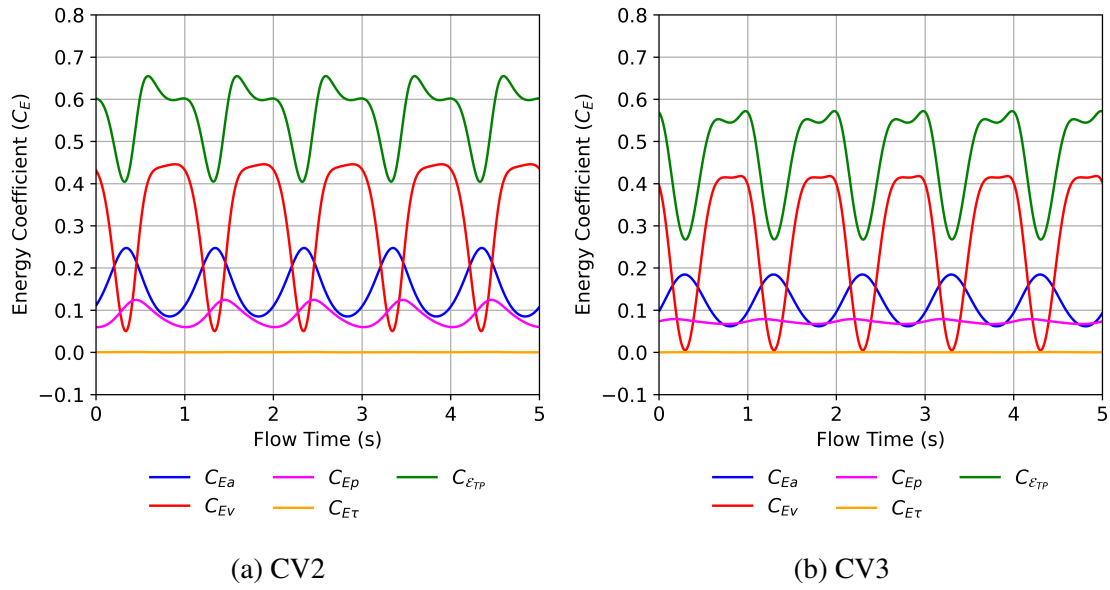
Figure 3.10: Far-Field Balance for CV1

The decomposition results reveal that drag power in this control volume is primarily driven by a combination of viscous dissipation (Φ and A_ϕ) and mechanical energy transport terms ($\mathcal{E}_{TP}$ and $\mathcal{X}m$). Notably, the gradients of kinetic energy and exergy (dE_k/dt and $\mathcal{X}v(t)$) exhibit a 180° phase shift relative to the mechanical energy terms, indicating that these gradients act to oppose the net mechanical energy flux across the wake plane. While both groups of terms are individually out of phase with the near-field drag signal, their combined effect (ΣE_k) results in a drag power that is in phase with the near-field result.

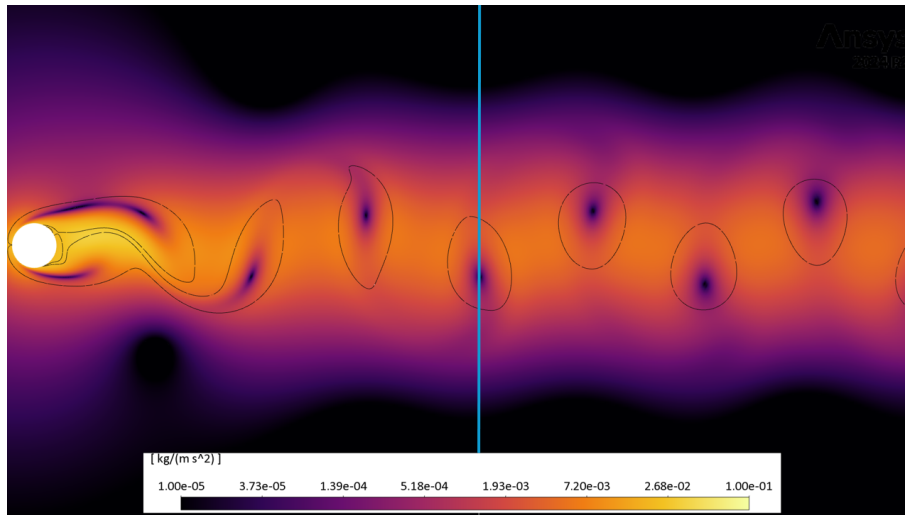
The energy and exergy decomposition results for CV2 and CV3 are shown in Figure 3.11 and 3.12 respectively. The results for CV2 and CV3 show the majority of the drag power being composed of viscous dissipation and mechanical energy. A large difference though when compared to CV1 is that the mechanical energy terms no longer have smooth sinusoidal oscillations, and again this is mirrored in the gradient of the kinetic energy. It is also noted how as the CV distance increases, the viscous dissipation term becomes more dominant in the drag power, with the mechanical energy term becoming less significant.



21

Figure 3.13: $\mathcal{E}_{TP}$ Decomposition at CV2 and CV3

Although the decomposition was able to identify the transverse energy as the cause of the non-sinusoidal oscillations, it is still important to understand the physical reasons as to why this is occurring. To do this the flow field was visualised at different time steps, with the kinetic energy contours and isoclips of the Q criterion plotted at CV3. Figure 3.14 shows the flow field at the trough of the $\mathcal{E}_{TP}$ oscillation ($t/T = 1.35$).

Figure 3.14: Kinetic Energy Contour at $t/T=1.35$ with Isoclips of Q Criterion
Blue line shows wake plane at CV3

At this particular instance in time it can be seen that a region over very little kinetic energy is crossing through the wake plane at CV3, this corresponds with the drop in mechanical energy at this time step (Figure 3.13b).

To understand the peak of the transverse energy oscillation, the flow field was visualised at the peak of the $\mathcal{E}_{TP}$ oscillation ($t/T = 1.75$). This is shown in Figure 3.15.

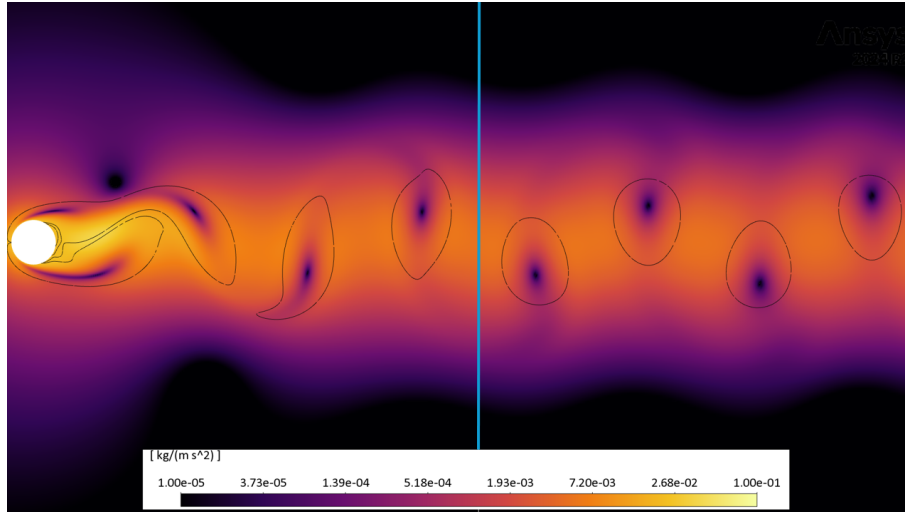


Figure 3.15: Kinetic Energy Contour at $t/T=1.75$ with Isoclips of Q Criterion
Blue line shows wake plane at CV3

It is seen at this point in time, the wake plane is located between two vortices, in a region of high kinetic energy. This accounts for the increased amount of measures mechanical energy at this time step, but still does not explain why the transverse energy is plateauing. To further understand this, the transverse energy was visualised at the same time step, as shown in Figure 3.16.

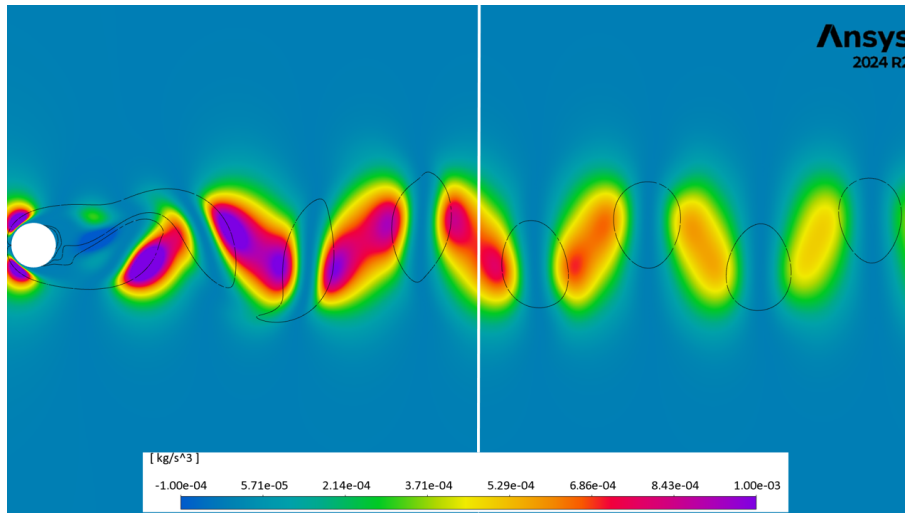


Figure 3.16: Transverse Energy Contour at $t/T=1.75$ with Isoclips of Q Criterion
White line shows wake plane at CV3

It can be seen that at $t/T = 1.75$, the wake plane is located in a region of high transverse energy, it can also be seen that on either side of the wake plane there is a hot spot of transverse energy. This position of the hot spots on either side of the wake plane is ultimately what causes the plateauing of the transverse energy oscillation, as the wake plane is always located in a region of high transverse energy when not within a vortex. It can be seen on the right hand side of the Figure that the Transverse energy begins to drop off,

suggesting that the energy is being lost to viscous dissipation. This is also supported when looking at the energy and exergy decomposition results for CV6, shown in Figure 3.17, where the mechanical energy term has returned to a sinusoidal oscillation and the viscous dissipation term has become the dominant term in the drag power.

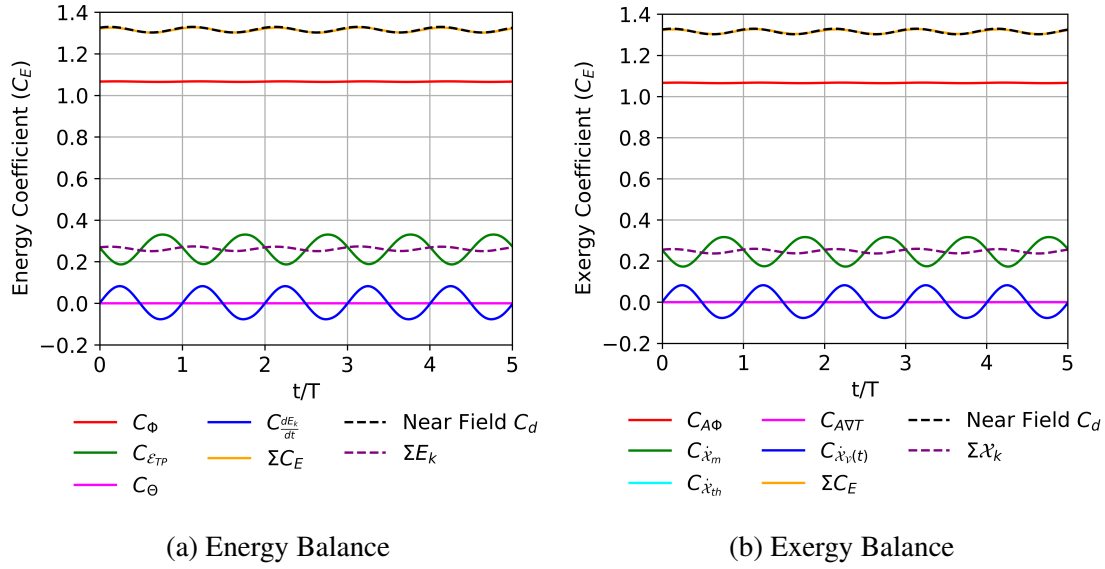


Figure 3.17: Far-Field Balance for CV6

The mathematical decomposition of the energy and exergy balances has allowed for a detailed understanding of the flow physics and the mechanisms dictating the drag power and the effects of the vortex street. Although the decomposition can be useful at individual control volumes, it is also important to understand how the energy and components change as the distance from the cylinder increases. To do this, the decomposed mechanical energy terms were plotted against distance as shown in Figure 3.18.

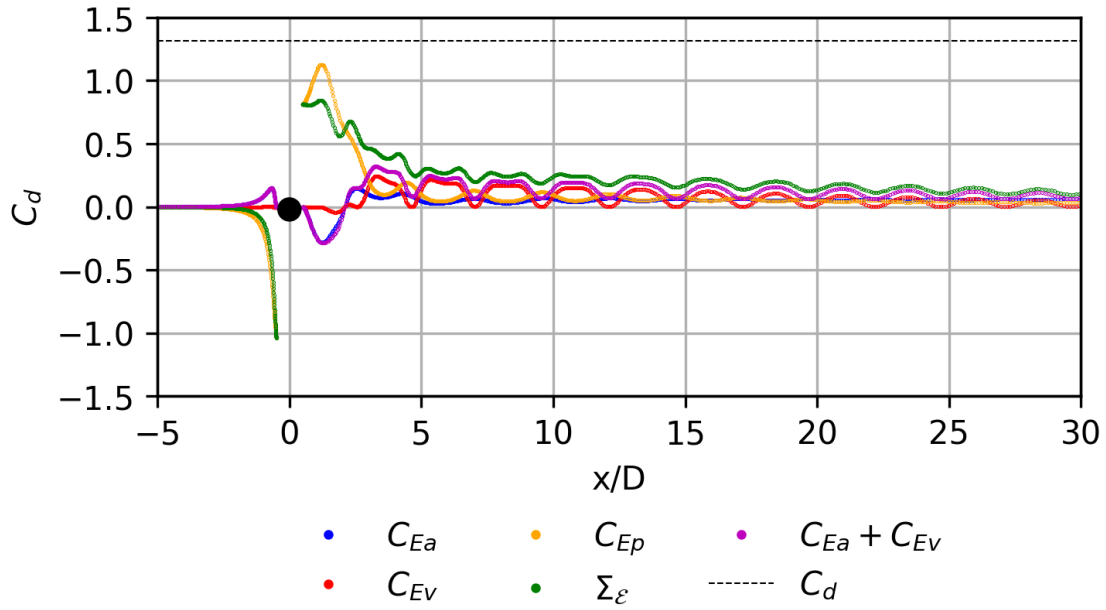


Figure 3.18: Mechanical Energy Wake Line Visualisation

The wake line visualisation in Figure 3.18 shows that right at the cylinder the majority of the drag power is composed of boundary pressure work ($\mathcal{E}_p$). This large value rapidly drops off and already contributes less than the other components by a distance of 5 diameters downstream of the cylinder. The non-sinusoidal oscillations of transverse energy (E_v) are clearly visible and only become smooth oscillations at a distance of roughly 30 diameters downstream. It is visible that all of the mechanical energy terms keep decreasing as the distance from the cylinder increases and theoretically would continue to do so until the energy is fully lost to viscous dissipation.

As the mechanical energy and kinetic energy terms are considered to be the recoverable portion of the energy, the potential for energy recovery in the wake can be evaluated as shown in Figures 3.19a and 3.19b.

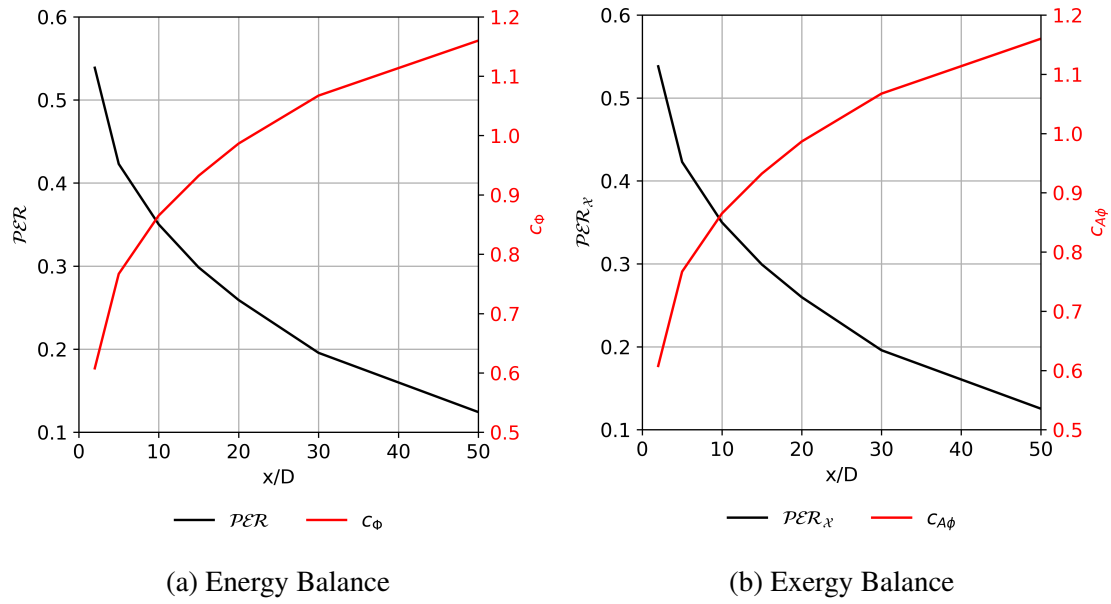


Figure 3.19: Potential for Energy Recovery in the Wake

As mentioned in the earlier results, the energy and exergy balances are near identical in this metric. Overall the results show that close to the cylinder there is a large amount of recoverable energy that is being wasted in the wake region. In this case the energy is being lost as kinetic energy wastage, as was discussed in the earlier decomposition results, and as such would require a mechanical energy recovery system such as a turbine to recover the energy. The results also show that the usability of the energy rapidly declines with distance with the majority of the recoverable energy being lost to viscous dissipation.

3.4 Test Case Conclusions

In summary, the vortex shedding test case demonstrated the capability of all three far-field methods to accurately capture both the drag coefficient and the oscillatory nature of the flow. The momentum-based method, while effective, showed a clear trend of increasing error with control volume size, particularly at larger distances from the cylinder. In contrast, the energy and exergy methods exhibited greater accuracy across the wake, with lower RMS errors and reduced sensitivity to truncation effects.

The decomposition analysis revealed that the drag power in this case is primarily driven by viscous dissipation and mechanical energy flux across the wake plane. Additionally, the unsteady kinetic energy gradients were shown to act in opposition to the net mechanical energy transport, resulting in a total drag power signal that remains in phase with the near-field response. Another important outcome of the decomposition was the ability of the energy and exergy terms to indicate the instantaneous positions of vortex cores within the vortex street, identified through irregularities in the mechanical energy oscillations.

Finally, the Wake and Potential for Energy Recovery ($\mathcal{P}\mathcal{E}\mathcal{R}$) analysis indicated that the flow is dominated by reversible losses near the cylinder as a result of kinetic energy deposition into the flow, which would potentially be recovered with a turbine.

Chapter 4

Oscillating Cylinder Test Case

4.1 Test Case Definition

This test case involves a two-dimensional circular cylinder oscillating sinusoidally about its center within a laminar, incompressible and, initially static fluid domain. That is, there is no imposed freestream velocity, and the flow field is entirely induced by the prescribed motion of the cylinder.

Originally developed by Józsa [25] as an internal flow solution, the case provides an analytical solution to the unsteady incompressible Navier-Stokes equations. It was later extended to an external flow configuration by Germaneau [26], enabling its application as a benchmark problem for numerical solvers. Furthermore, Germaneau's work was able to apply the energy balance to the case, thus, the test case not only allows for direct comparison of computed velocity and pressure fields against reference data, but it also allows it to serve as a form of validation to test the correct implementation of the energy balance into the CFD solver.

In this setup, all fluid motion arises solely due to the no-slip boundary condition imposed on the moving cylinder wall. Since there is no freestream velocity, the obtained flow-field is entirely generated by the oscillatory motion of the cylinder. Notably, at all points along the surface of the cylinder, the induced fluid velocity remains tangential, that is, it is oriented at a 90° angle to the local wall-normal vector. This results in a symmetric pressure and viscous stress distribution around the cylinder at any given instant in time, as shown in Figure 4.1.

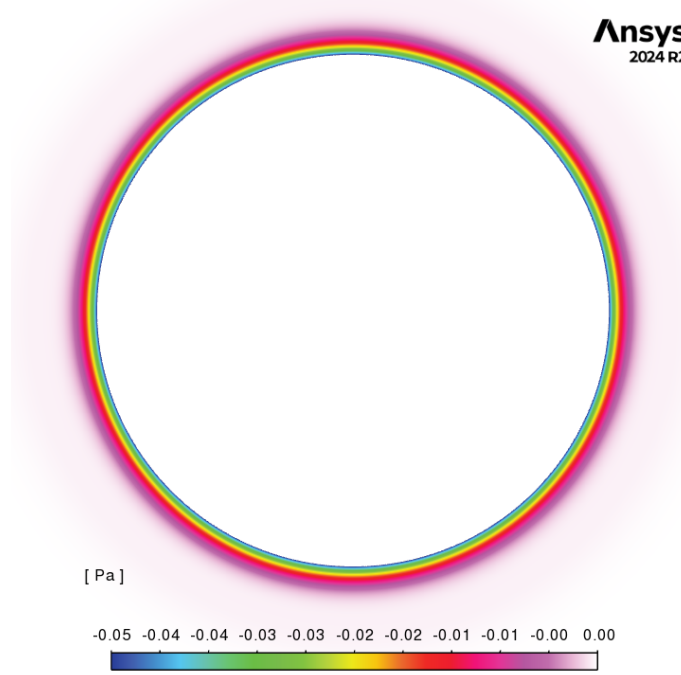


Figure 4.1: Guage Pressure around Cylinder

As a consequence, the net aerodynamic force acting on the cylinder is expected to be zero throughout the oscillation cycle. Therefore, in contrast to the previous test case, where the energy deposited in flow manifests as drag power, the energy transfer mechanisms in this case are not related to drag. Instead, the energy imparted to the fluid is balanced by a propulsive input term, directly linked to the cylinder's prescribed motion. This makes the test case particularly valuable for exploring the behaviour of energy and exergy balances in the absence of drag force.

In this test case, the energy and exergy balance are not determined through the drag power, but rather through the propulsive input term ($\mathcal{P}_s$). As such the far-field balances are defined as shown in equations 4.1 and 4.2 for the energy and exergy balance respectively.

$$\mathcal{P}_s = \frac{dEk}{dt} + \mathcal{E}_{TP} + \Theta + \Phi \quad (4.1)$$

$$\mathcal{P}_s = \dot{\mathcal{X}}_v(t) + \dot{\mathcal{X}}_{th} + \dot{\mathcal{X}}_m + \dot{\mathcal{A}}_\Phi + \dot{\mathcal{A}}_{\nabla T} \quad (4.2)$$

4.1.1 Numerical Setup

ANSYS Fluent 2024R2 was used to solve the incompressible Navier-Stokes equations. The motion of the cylinder is introduced through a moving wall boundary condition. The motion was prescribed as shown in equation 4.3, where A is the maximum velocity (1m/s in this case), and ω was calculated using the strouhal number of the oscillations (0.010186).

$$\text{Wall Rotational Velocity} = A \cos(\omega t) \quad (4.3)$$

The strouhal number was calculated using the womersly number, $Wo = 8$ from the analytical solution of Józsa [25], hence the strouhal number was obtained as shown in equation 4.4, where the Reynolds number for this case is $Re = 1000$.

$$Str = \frac{Wo^2}{2\pi Re} \quad (4.4)$$

The solver setup is slightly modified from the previous test case, with the main difference being that pressure velocity coupling method has been changed to a coupled scheme, and the time advancement method has been changed to an iterative time advancement method as a result. This was done as this test case experiences very small shifts in the pressure field which are directly introduced by the changes in the velocity. Using a segregated scheme would potentially result in phase delays between the pressure and velocity fields, which would result in amplification of any errors in comparison to the analytical solution.

The solver setup is shown in Table 4.1 with laminar flow selected as a viscous model to match the analytical solution and a time step size of $1 \times 10^{-3}s$.

Table 4.1: Numerical Solver Inputs

Solver Parameter	Input
Solver Type	Pressure Based
Time Advancement	Iterative Time Advancement
Pressure Velocity Coupling Scheme	Coupled
Transient Formulation	Second Order Implicit
Gradient	Least Squares Cell Based
Pressure Discretisation	Second Order
Density Discretisation	Second Order Upwind
Momentum Discretisation	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind
Energy	Second Order Upwind

The computational domain is defined as a large circle of radius $= 10R$, where R is the radius of the cylinder in the study ($R = 1m$). The cylinder is placed at the centre of the domain, and the oscillations occur rotationally about the origin. The boundary condition of the domain edges are set as a velocity inlet on the upper half of the domain, and the pressure outlet on the lower half of the domain, whilst the wall is a moving wall boundary condition, with no slip condition. The computational domain is shown in Figure 4.2.

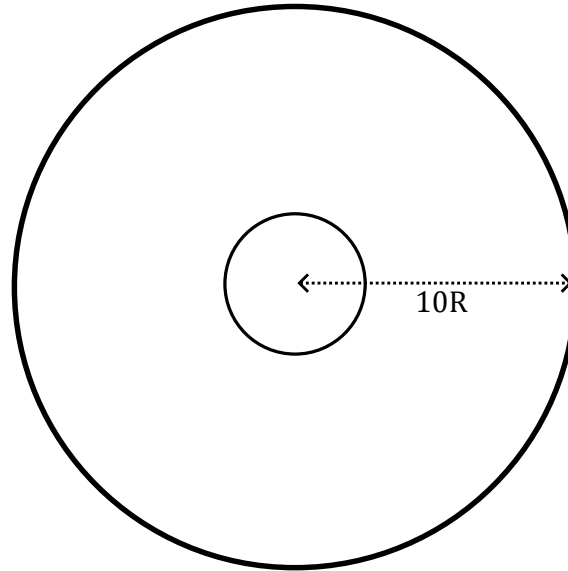


Figure 4.2: Computational Domain Dimensions
Diagram not to Scale

4.1.2 Control Volume Definition

Similarly as done in the Vortex Shedding Test Case, multiple control volumes were defined around the cylinder to apply the far-field balances. For this test case, the control volumes are defined as concentric circles around the cylinder, as shown in Figure 4.3.

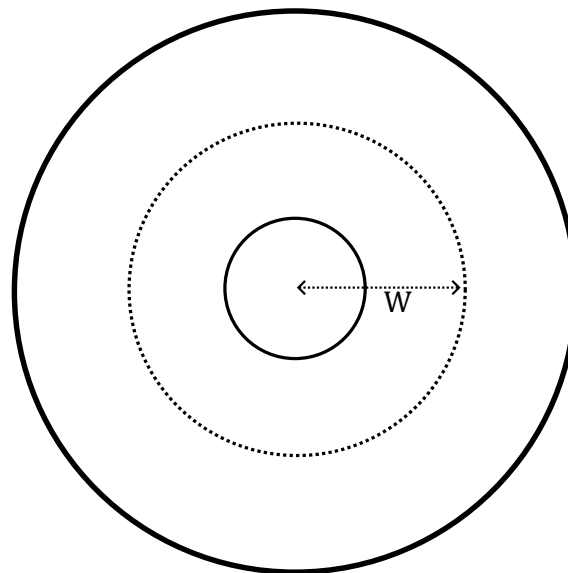


Figure 4.3: Control Volumes Diagram
Diagram not to Scale

Three control volumes were defined for the analysis, with all three being fairly close to the cylinder surface as the flow in this case is entirely a boundary layer solution. The radii of the control volumes are defined in Table 4.2, where it can be seen that CV1 has

been located well within the boundary layer, well CV2 and CV3 move further outward to investigate the dissipation of the energy emitted radially outward from the boundary layer.

Table 4.2: Control Volume Radii

Control Volume Name	W
CV1	1.1R
CV2	1.2R
CV3	2.0R

4.2 Near-Field Results Validation

To validate the numerical setup and mesh, a direct comparison with the analytical solution is performed. The results were validated at distinct time points in the cylinder's oscillatory period. The wall speed with respect to time is shown in Figure 4.4, where it can be seen that the wall speed oscillates sinusoidally. The time period (T) for the oscillations is $98.1747s$, all the results for the test case are plotted against the non-dimensionalised time using this set period.

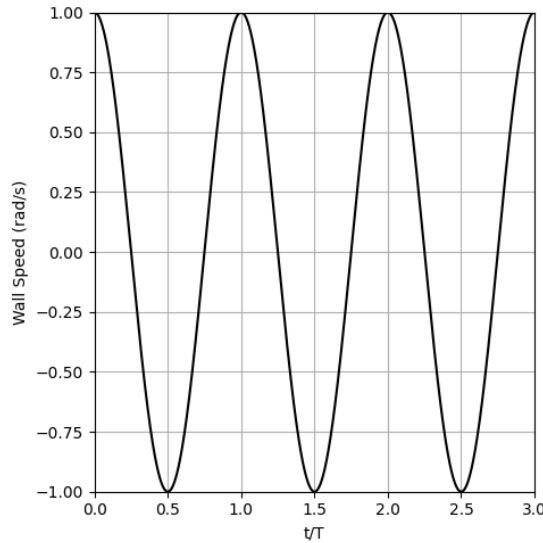


Figure 4.4: Wall Radial Velocity vs Time

To validate against the analytical solution, the velocity and pressure values were compared along a line extending radially outward from the edge of the cylinder. The comparisons were done at 4 distinct time points in the cylinder's oscillatory period, $t/T = 1, 1.25, 1.5, 1.75$. These time points were selected as they represent when the cylinder is at its maximum rotational speed in both rotational directions, and when the cylinder rotational speed = 0.

The comparison of tangential velocity with the analytical results along the radial line is shown in Figure 4.5. In the results it can be seen that at all the time points the maximum velocity on the wall ($r/R = 1.00$) matches the rotational speed of the cylinder.

Moving away from the wall, the CFD results maintain excellent agreement with the analytical solution across most of the boundary layer. Small deviations can be observed in the outer regions of the boundary layer, which are attributed to minor numerical errors arising from discretisation and time integration. The close agreement in velocity profiles indicates that the velocity gradients in the boundary layer are being directly resolved, rather than approximated through wall functions.

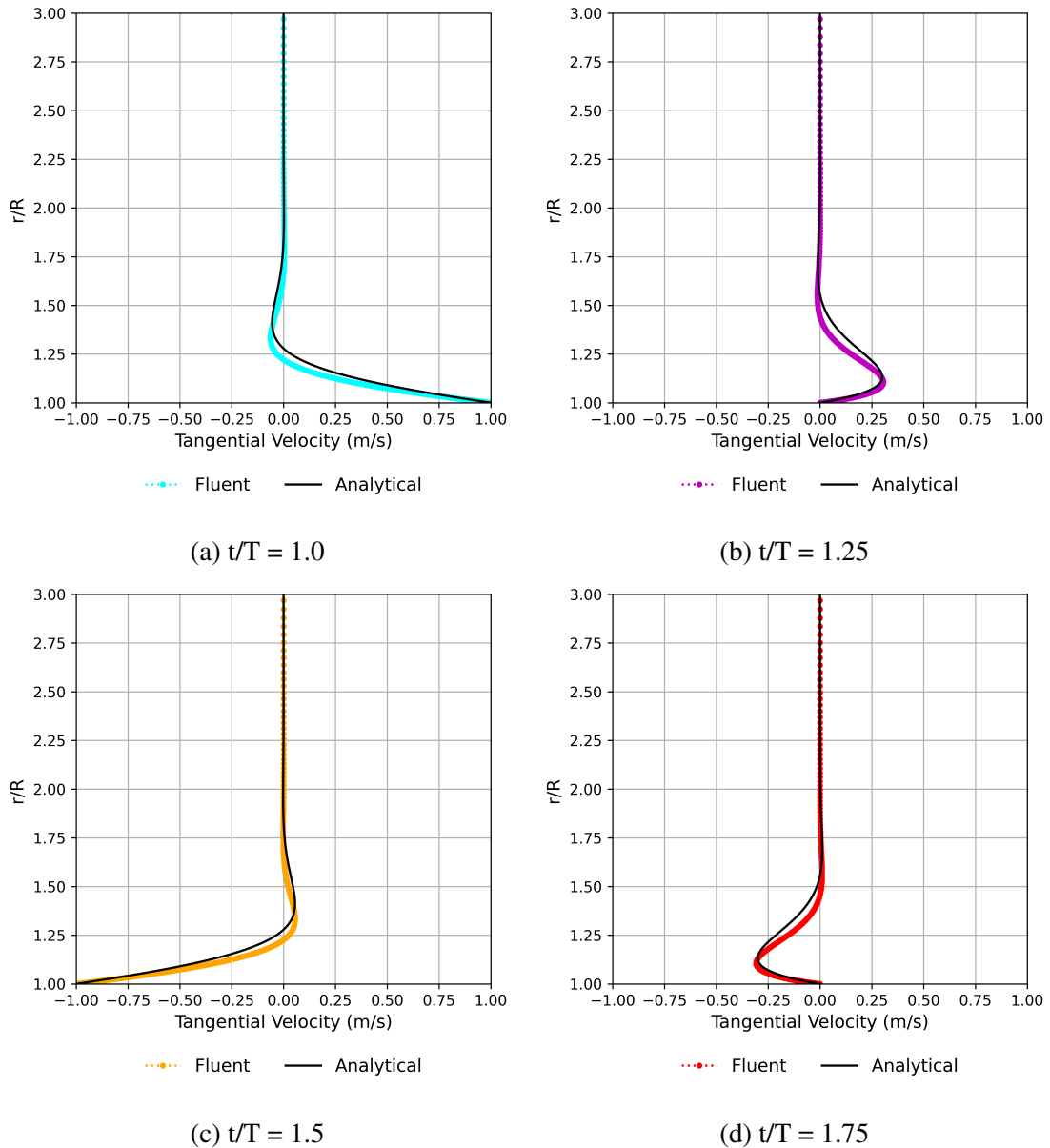


Figure 4.5: Velocity Comparison with Analytical Solution
Analytical Results Provided by Germaneau [26]

Further verification is conducted by plotting the gauge pressures at the same time points along the radial line as shown in Figure 4.6. It can be seen that the pressure values exhibit very similar trends to the analytical solution. The largest amount of error in this case occurs at $t/T = 1$ and $t/T = 1.5$, where it can be seen the max gauge pressure on the wall for the CFD has roughly $0.011Pa$ lower gauge pressure than the analytical solution. This error is reduced at $t/T = 1.25$ and $t/T = 1.75$, where the difference in the maximum gauge pressure on the wall is less than $0.002Pa$.

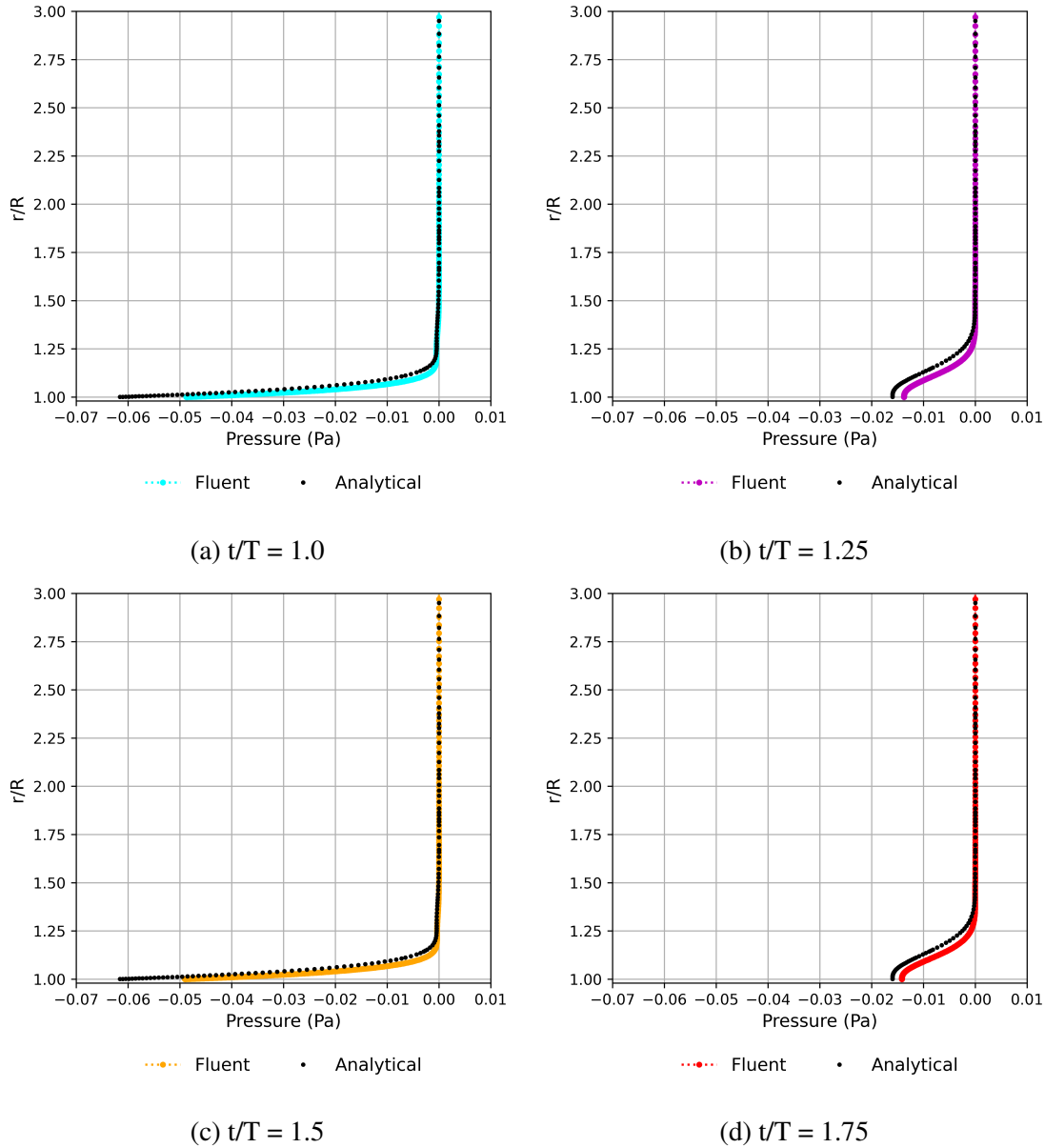


Figure 4.6: Gauge Pressure Comparison with Analytical Solution
Analytical Results Provided by Germaneau [26]

The velocity and pressure distributions obtained from the simulation demonstrate smooth, physically consistent profiles with no indications of major error sources. The behaviour of the velocity and pressure fields indicates that the numerical scheme is effectively cap-

turing the underlying physics of the test case, and is adequately resolving the gradients within the boundary layer. This consistency provides confidence that the present numerical setup and mesh combination is sufficient to conduct the far-field analysis.

4.3 Far-field Balances

To evaluate the test case, the far-field balances were applied. As was discussed in the test case description, the net drag force for this case is $0N$ at all points in time. This is reflected in the momentum balance results shown in Figure 4.7, where it can be seen that the near-field and far-field C_d results are constantly oscillating about zero.

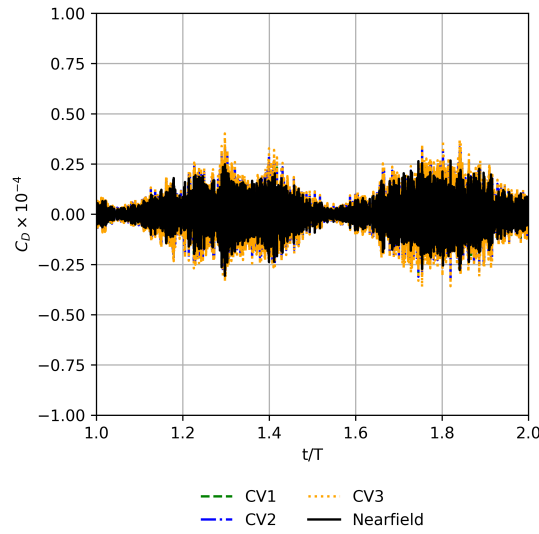


Figure 4.7: C_d comparison with Momentum Balance

Although the results are not perfectly zero, the oscillations are very small, and are most likely attributed to small amounts of numerical noise from the pressure or viscous forces.

The energy and exergy balances provide a far more comprehensive picture of the flow field behaviour. However, validating these balances requires first determining the near-field value $\mathcal{P}_S$. In ANSYS Fluent, obtaining this value directly is problematic because the software internally reconstructs the wall shear stress components using an undocumented method. According to the Fluent User's Guide and Technical Communication [18, 27], these wall shear stress values are only correct on the wall, implying that some form of boundary-condition correction is applied during their calculation. While these corrected values may be suitable for post-processing directly at the wall, they do not include all tensor components required for the present formulation. Conversely, manually reconstructing the stresses from velocity gradients at the wall yields inaccurate results, further complicating the direct calculation of $\mathcal{P}_S$ within Fluent. To demonstrate this, the $\mathcal{P}_S$ value obtained when using Boussinesq's approximation to obtain $\bar{\tau}$ is compared against the value obtained from the analytical solution in Figure 4.8.

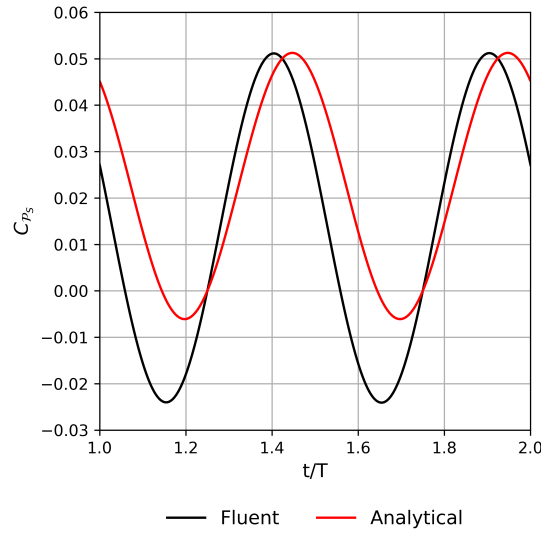


Figure 4.8: $\mathcal{P}_S$ Comparison with Analytical Solution
Analytical Data from Germaneau [26]

The results show that the numerical solution in Fluent has a larger amplitude compared to the analytical solution, furthermore it oscillates about a lower mean value, and the results from Fluent exhibits a phase shift when compared with the analytical solution. It is for this reason that for the far-field comparisons, the $\mathcal{P}_S$ values from the analytical solution were used to represent the left hand side of the balance.

The energy and exergy balances are applied at all three of the aforementioned control volumes to compare how increasing radial distance affects the flow field. Figure 4.9 shows the results of the energy and exergy balances for each control volume and compares them with the analytical solution for $\mathcal{P}_S$.

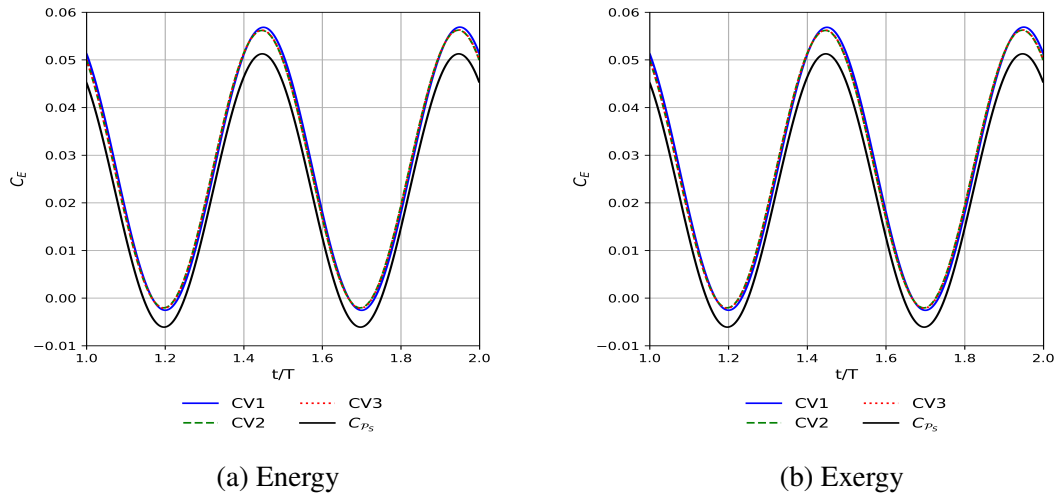


Figure 4.9: Far-Field Comparison with Analytical $\mathcal{P}_S$ [26]

Just as what was observed in the vortex shedding test case, it is seen that the energy and exergy balance yielded near identical results, as a result of very little thermal effects or

heat transfer in the flow field. It is also seen that all of the CV's yield near identical results to each other. Overlaying the analytical $\mathcal{P}_S$ solution onto the numerical results shows that the results are in phase and have extremely similar amplitudes. Although the phase and amplitude are in agreement, it is seen that Fluent's results are all shifted slightly higher than the analytical solution, suggesting Fluent is slightly overestimating the amount of work being done by the cylinder.

4.4 Far-Field Decompositions

To better understand the underlying mechanisms behind the energy transfer to the far-field, the energy and exergy values are mathematically decomposed and plotted for each CV. Figure 4.10 shows the decomposition for CV1, wherein it is shown that at this close of a control volume the energy from the cylinder is already largely lost to viscous dissipation. It is also noted that the mechanical energy flux across the wake plane and the kinetic energy gradient's have similar amplitudes, but unlike in the vortex shedding test case, the terms do not have a 90° phase shift to cancel each other's oscillations.

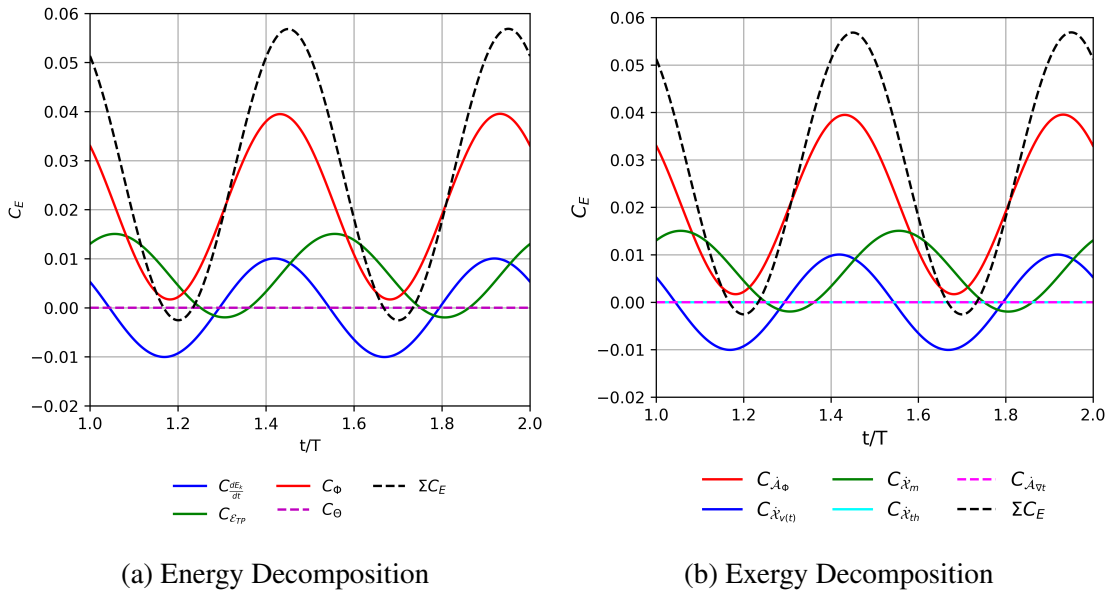


Figure 4.10: Far-Field Decomposition for CV1

The effects of distance on the loss of mechanical energy to viscous dissipation are shown in Figure 4.11 for CV2 and Figure 4.12 for CV3, Where it can be seen that outside of the boundary layer the work done from the cylinder is almost all lost to viscous dissipation. Moving from CV2 to CV3 shows no difference in the kinetic energy gradients. It is seen that outside the Boundary Layer there is little to no flux of energy across the wake planes radially outward.

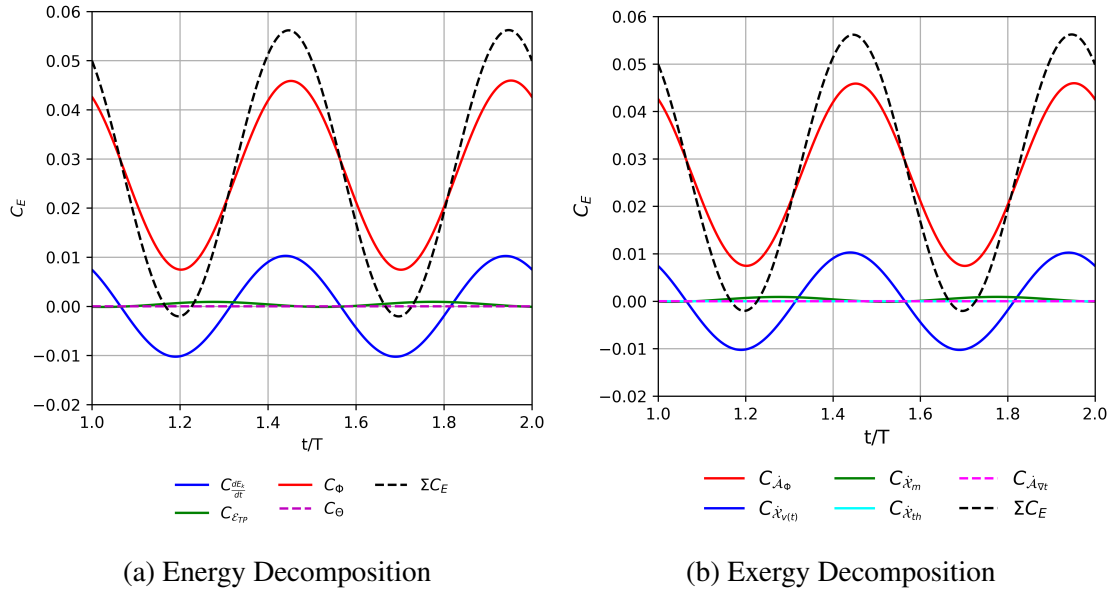


Figure 4.11: Far-Field Decomposition for CV2

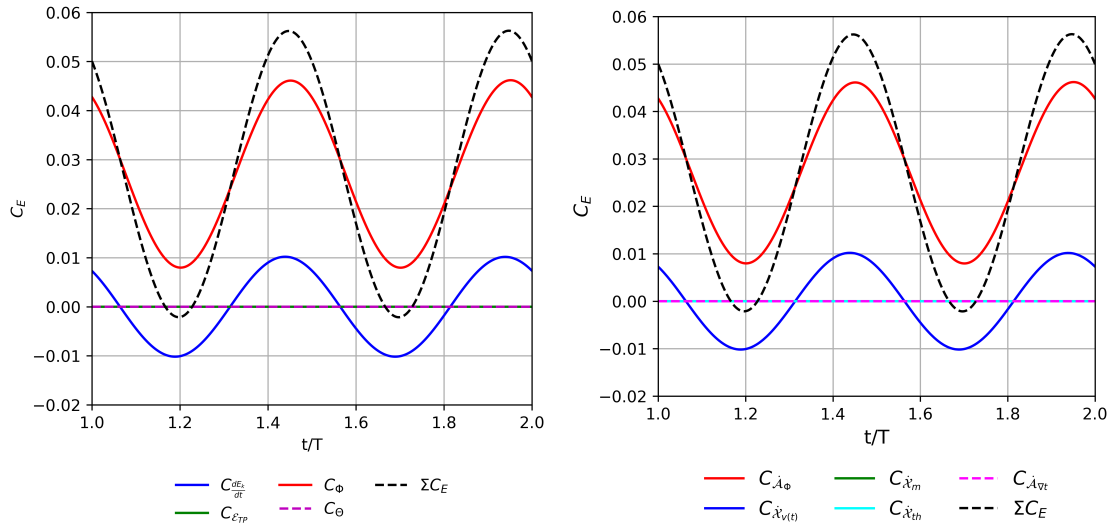


Figure 4.12: Far-Field Decomposition for CV3

To better understand why there is a notable amount of mechanical energy flux across the wake plan within the boundary layer but almost nothing outside of this region, the mechanical energy term was decomposed at each control volume into its respective sub-components.

The mechanical energy decomposition for CV1 is shown in Figure 4.13, where it can be seen that the mechanical energy flux across the wake plane is dominated by the viscous energy fluctuations. There are zero contributions from the other subcomponents suggesting that the viscous effects are the primary driver of the work done on the flow by the cylinder in the boundary layer.

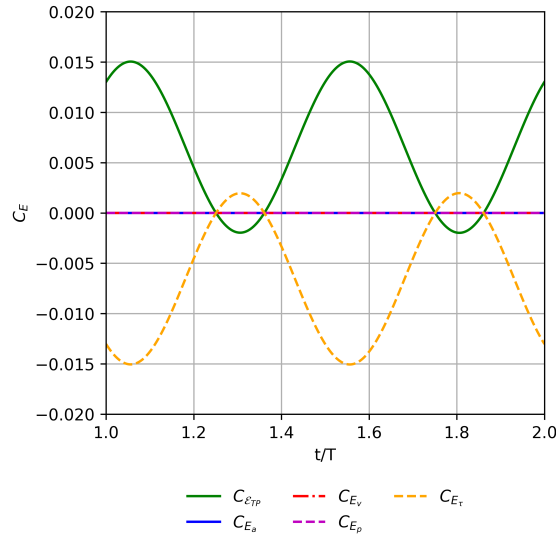


Figure 4.13: Mechanical Energy Decomposition for CV1

CV2 and CV3's mechanical energy decomposition is shown in Figure 4.14. At the edge of the boundary layer (CV2), it is seen that the viscous energy fluctuations have reduced to be very minor. Looking further away from the cylinder (CV3), there are no fluctuations in the viscous energy across the wake plane, thus explaining the rapid drop off in the mechanical energy term seen in the decompositions earlier (Figures 4.10-4.12).

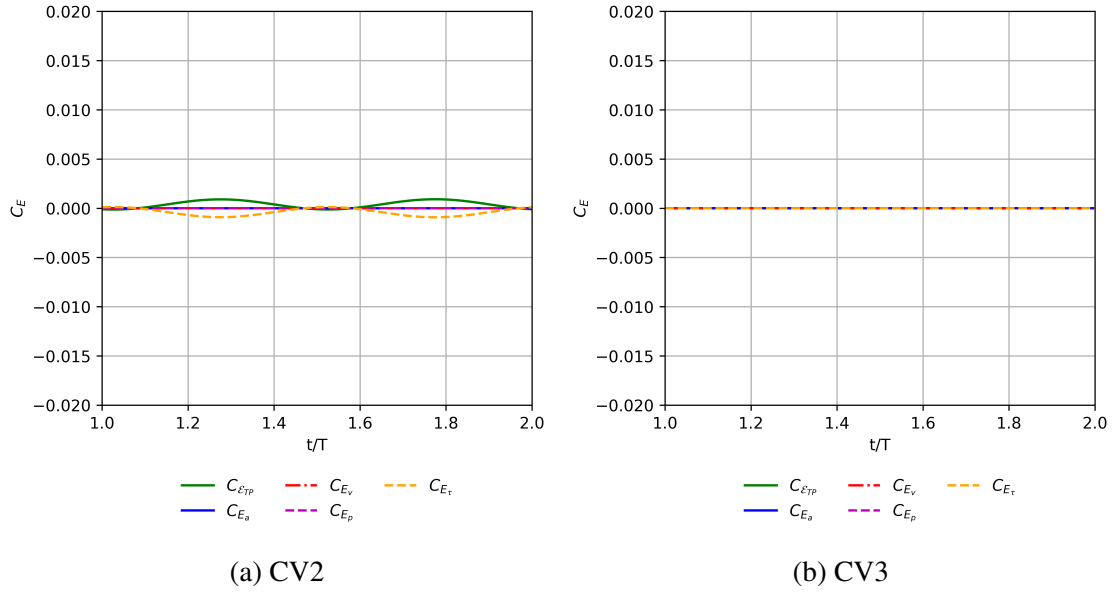


Figure 4.14: Mechanical Energy Decomposition at CV2 and CV3

4.5 Test Case Conclusions

In summary, the oscillating cylinder test case demonstrated the ability of the energy and exergy balances to accurately capture the energy transfer mechanisms in a flow field where drag is not present. The momentum-based method yielded a constant C_d of zero, consistent with the expected behaviour of the oscillating cylinder, whereas the energy and exergy balance were able to quantify the energy outputs and losses from the cylinder doing work on the flow.

Although the right hand side of the energy and exergy balance's gave reasonable results that lined up well with the analytical solution, the $\mathcal{P}_S$ term did not line up correctly with the right hand side of the balances or the analytical solution. The inability to correctly resolve the $\mathcal{P}_S$ term, is believed to be related to the reconstruction of the viscous stress tensor ($\bar{\tau}$) on the wall, needing adjustments due to the boundary condition. This highlights further work being needed on this aspect to understand how Fluent reconstructs the stress tensor on the wall boundary and to apply this in the implementation of the $\mathcal{P}_S$ term.

Overall the far-field results showed that the flow in this case is dominated by viscous effects as the transfer mechanism to the far-field. Although there is a consistent measurement of the kinetic energy gradient ($\frac{dEk}{dt}$), this movement in kinetic energy is always in a tangential direction around the cylinder and does not contribute to the radial energy flux across the wake plane. The mechanical energy flux across the wake plane is dominated by perceivably recoverable viscous energy across the wake planes, and irreversibly lost viscous dissipation.

Chapter 5

Combined Test Case

5.1 Test Case Definition

The final test case combines the features of the two previous configurations, incorporating both the vortex shedding flow regime of Test Case 1 ($Re = 140$) and the azimuthal oscillation of the cylinder wall from Test Case 2. This results in a flow field where naturally unsteady wake dynamics interact directly with motion-induced boundary layer disturbances. The oscillation is implemented onto the baseline vortex-shedding condition, producing a more complex unsteady flow environment.

The primary motivation for this case is to evaluate the performance of the energy and exergy balances in a scenario where both classical drag-producing mechanisms and body-induced motion are present simultaneously. The coupling of these two effects is expected to modify the near-wall shear layers, potentially intensifying viscous effects and altering the balance between reversible and irreversible energy transfer processes in the flow.

In particular, this configuration aims to determine whether the oscillatory motion changes the partitioning of energy between recoverable mechanical forms and irreversibly lost viscous dissipation. By introducing additional shear and unsteady acceleration near the wall, the motion may enhance entropy generation rates, thereby affecting the exergy balance. Furthermore, the test case investigates whether these increases in viscous effects influence the way energy is transported and dissipated along the wake, especially with regards to the potential for energy recovery ($\mathcal{P}\mathcal{E}\mathcal{R}$).

As the test case is again a drag-dominated flow, the energy and exergy balances in the case are achieved through comparing to the drag power ($F_B \cdot U_\infty$). This is shown in equations 5.1 and 5.2 for the energy and exergy balance respectively. This balance allows for direct comparison with the Drag force as the right hand side can be divided by V_∞ .

$$F_B \cdot U_\infty = \frac{dEk}{dt} + \dot{\mathcal{E}}_{TP} + \Theta + \Phi \quad (5.1)$$

$$F_B \cdot U_\infty = \dot{\mathcal{X}}_v(t) + \dot{\mathcal{X}}_{th} + \dot{\mathcal{X}}_m + \dot{\mathcal{A}}_\Phi + \dot{\mathcal{A}}_{VT} \quad (5.2)$$

5.1.1 Numerical Setup

For conducting the simulation, the same numerical setup as the vortex shedding test case was used, with the only difference being that the cylinder wall is now oscillating sinusoidally about its center using a moving wall boundary condition. The simulation was conducted in ANSYS Fluent 2024R2, with the same solver setup and mesh as utilised for the vortex shedding test case. The solver setup is summarised in table 5.1, while further details on the domain and mesh can be found in Chapter 3 (Vortex Shedding Test Case).

Table 5.1: Numerical Solver Inputs

Solver Parameter	Input
Solver Type	Pressure Based
Time Advancement	Non-Iterative Time Advancement
Pressure Velocity Coupling Scheme	Fractional Step
Transient Formulation	Second Order Implicit
Gradient	Least Squares Cell Based
Pressure Discretisation	Second Order
Density Discretisation	Second Order Upwind
Momentum Discretisation	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind
Energy	Second Order Upwind
Time Step Size (s)	1.4×10^{-4}

The oscillations are determined by the the sinusoidal expression shown in equation 5.3.

$$\text{Rotation Speed} = A \cos(2\pi ft) \quad (5.3)$$

The frequency used differs to what was previously implemented in the oscillating case, as the period in the oscilattng case was too slow to have a noticeable effect in the case of this study. Instead the frequency was set to match the frequency of the vortex shedding from the first test case, which was calculated using the strouhal number of 0.172.

5.1.2 Control Volume Definition

As this test case is intended to highlight the differences between the oscillating-cylinder configuration and the standard vortex shedding case, three control volumes were defined around the cylinder to apply the far-field balances. The control volume layout uses the same base dimensions and wake-plane positioning as in the vortex shedding case (Figure 5.1), ensuring that any observed differences can be directly attributed to the oscillatory motion rather than changes in integration regions.

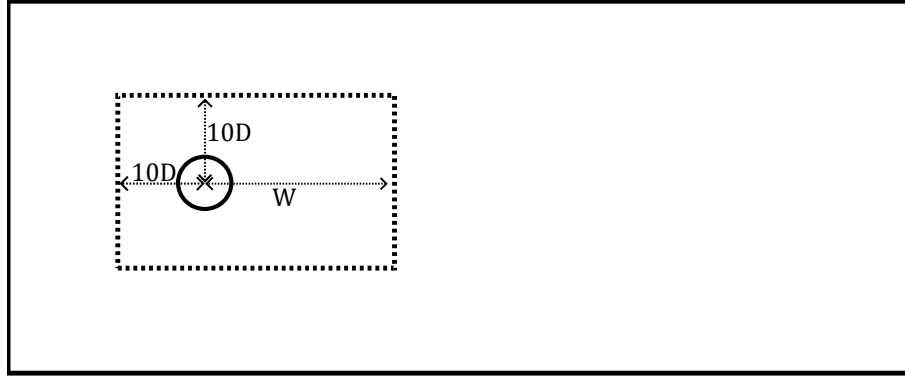


Figure 5.1: Control Volumes Diagram
Diagram not to Scale

The first control volume (CV1) is positioned close to the cylinder to capture the near-wake region immediately downstream of vortex formation. The second control volume (CV2) is located further downstream at a position where coherent vortices are well-developed and can be clearly detected in the mechanical energy flux term. The final control volume (CV7) is placed in the far-wake region, where most of the flow's recoverable mechanical energy has been lost to viscous dissipation, enabling evaluation of how the oscillatory motion influences the ultimate rate of irreversible energy losses. The wake distances are summarised in table 5.2.

Table 5.2: Control Volume Wake Lengths

Control Volume Name	W
CV1	2D
CV3	10D
CV7	50D

5.2 Results

The results for this test case are organised into four main subsections. The first subsection presents the near-field evaluation, directly comparing the drag characteristics of the oscillating cylinder with those of the standard vortex shedding case. The second subsection examines the far-field balances, showing the momentum, energy and exergy comparisons. For the decompositions subsection, as the previous test cases demonstrated, within this flow regime, the energy and exergy balances yield near-identical results, the far-field decomposition analysis therefore is presented only for the energy balance, with the corresponding exergy decomposition plots provided in the Appendix. Finally, the fourth subsection provides a detailed wake analysis to investigate how the oscillatory motion influences vortex dynamics and the distribution of energy dissipation along the wake.

5.2.1 Near-Field Evaluation

To understand the impact of oscillatory motion on the near-field flow characteristics, a detailed comparison of the drag coefficients for the oscillating and standard cases is conducted. Just as what was done in test case 1, the analysis is done from 14s to 15s of flow time, a significant amount of time after the flow has reached a periodically repeating state. Figure 5.2 shows the drag coefficient oscillations over time for the oscillating cylinder and the static cylinder.

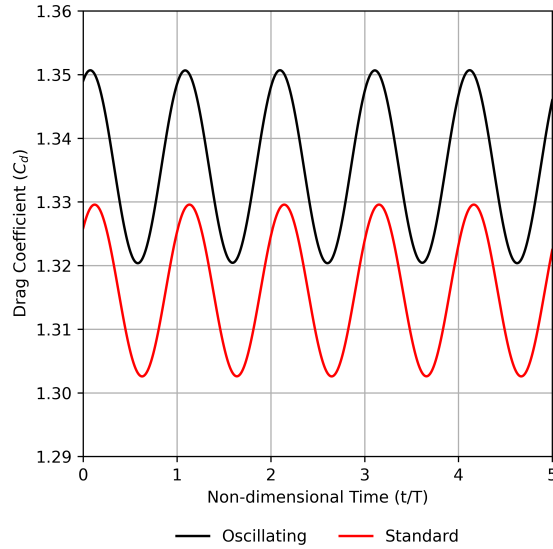


Figure 5.2: Near-Field C_d Comparison for Case 3 (Oscillating) and Case 1 (Standard)

From Figure 5.2, it is evident that the oscillating case exhibits an increase in the baseline value about which the drag coefficient oscillates, when compared to the standard case. This observation is confirmed in Table 5.3, where the mean C_d of the oscillating cylinder is slightly higher (1.3355) than that of the standard case (1.3160), a roughly 1.5% increase due to the oscillating motion. Despite this upward shift in the mean value, the amplitude of the drag fluctuations remains nearly unchanged. Similarly, the dominant frequency of the drag signal is consistent between the two cases at 7.2 Hz, indicating that the imposed oscillatory motion does not alter the natural shedding frequency of the wake. These results suggest that while the oscillatory wall motion increases the overall drag level experienced by the cylinder, the unsteady vortex dynamics governing the amplitude and frequency of drag oscillations remain largely unaffected.

Table 5.3: Near-Field Drag Comparison

Simulation Case	Mean C_d	Amplitude C_d	Frequency (hz)
Standard	1.3160	0.0268	7.2
Oscillating	1.3355	0.0302	7.2

5.2.2 Far-Field Balances

For evaluating how the far-field balances are affected by the oscillatory motion, the energy and exergy balances are plotted at CV1, CV3 and CV7. The plots compare the balance for the oscillating cylinder and compares with the results from the standard vortex shedding case. The plots for the energy and exergy balances are presented in Figures 5.3-5.5.

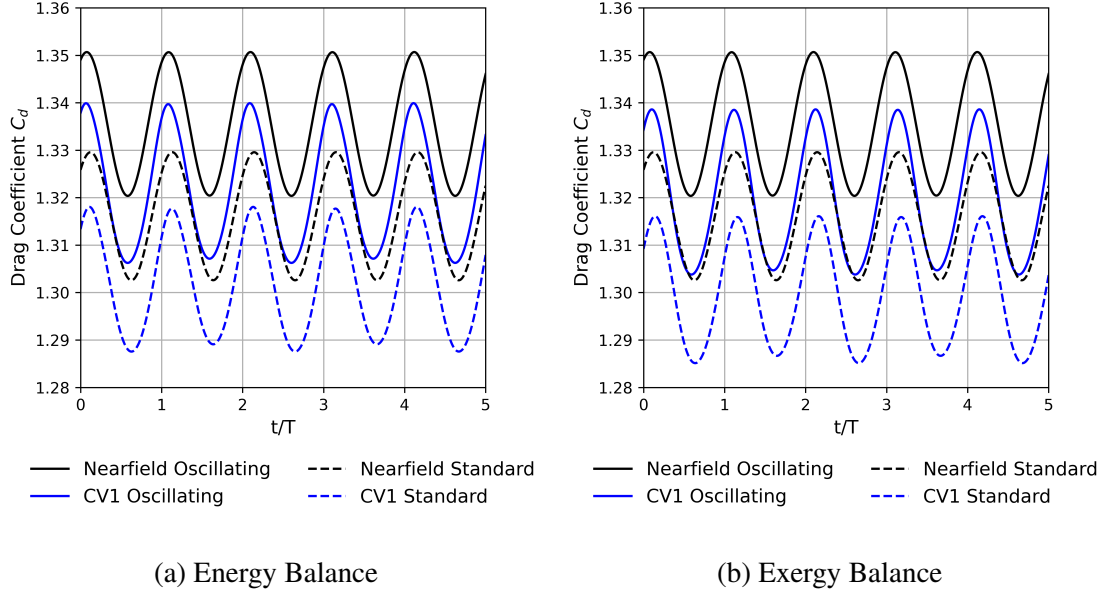


Figure 5.3: Energy and Exergy Balances for CV1

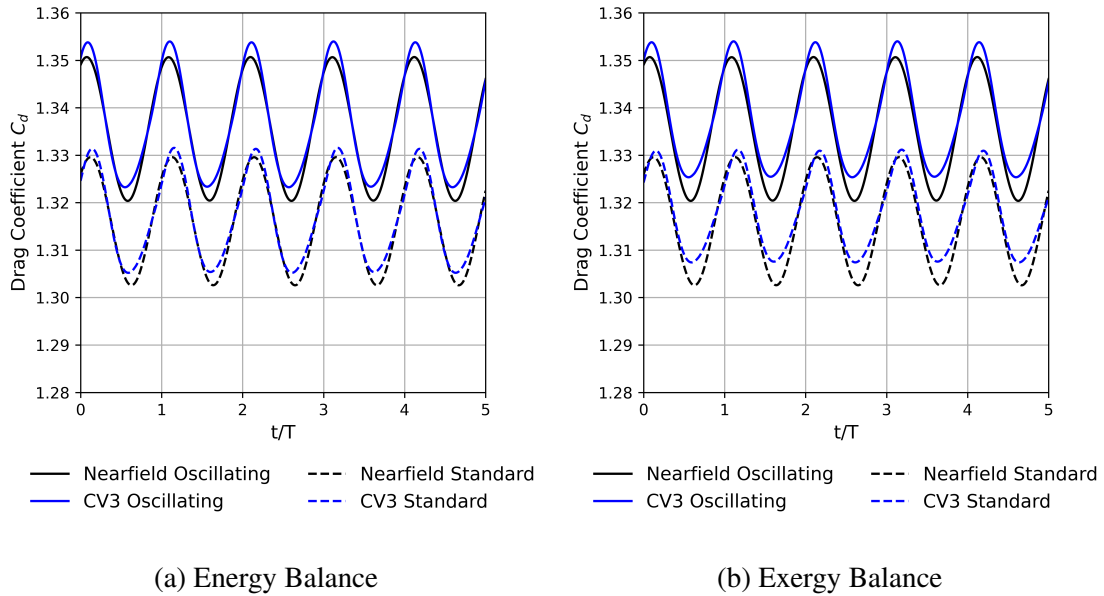


Figure 5.4: Energy and Exergy Balances for CV3

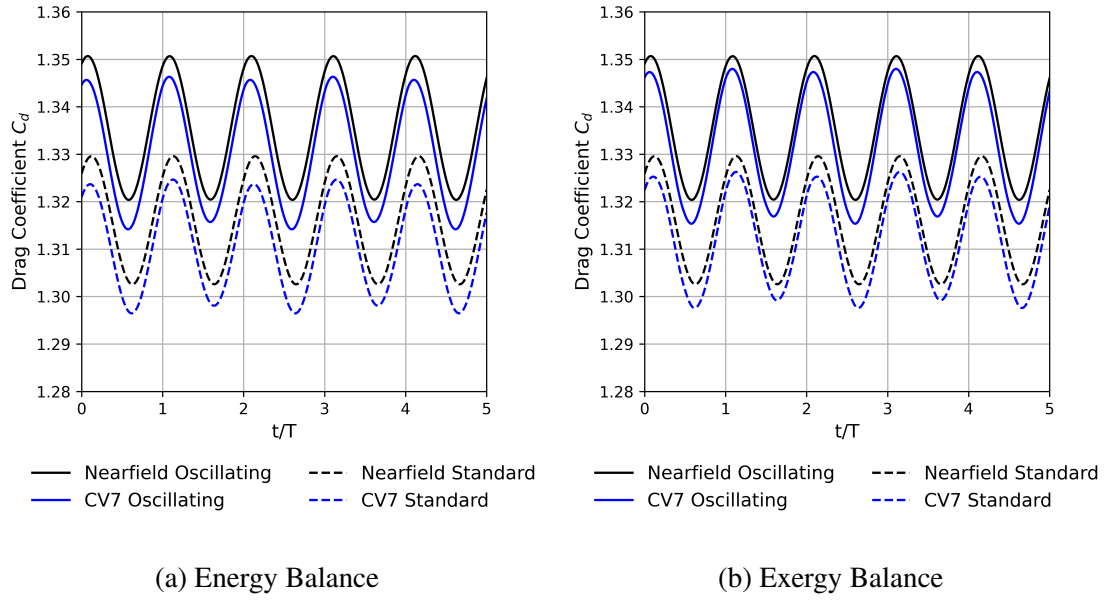


Figure 5.5: Energy and Exergy Balances for CV7

Looking at the energy and exergy balances it is seen that the energy and exergy balances again give extremely similar results across all control volumes. It is also noted that the trend across all three CV's is consistent, with the balances for the oscillating case shifting to a higher C_d value, but maintaining the same frequency and oscillation amplitude. The introduction of the oscillating motion has not affected the accuracies of the far-field balances, this is demonstrated in Tables 5.4 and 5.5 for the energy and exergy respectively.

Table 5.4: Energy Balance Time Averaged C_d

Case	Near-Field	CV1	CV3	CV7
Oscillating	1.3355	1.3221	1.3369	1.3303
Standard	1.3160	1.3018	1.3170	1.3105

Table 5.5: Exergy Balance Time Averaged C_d

Case	Near-Field	CV1	CV3	CV7
Oscillating	1.3355	1.3200	1.3379	1.3317
Standard	1.3160	1.2997	1.3180	1.3252

In the table it is consistently seen for all the control volumes and both the energy and exergy methods that the far-field balances increase the mean C_d value by roughly 1.5% when compared to the standard case. This is consistent with the changes that were experienced with the near-field results.

5.2.3 Far-Field Decompositions

To identify the source of the increased drag observed in the oscillating case, the drag power obtained from the energy balance is decomposed into its constituent subcomponents and directly compared against the standard case. This decomposition allows the contribution of viscous effects induced by the rotational motion to be distinguished from any general increase across all drag components. As outlined in the case description, only the energy balance results are presented here, since the exergy balance yields near-identical trends, the corresponding exergy decomposition plots are included in the Appendix for completeness.

Looking at the results for CV1 (Figure 5.6), it is seen that the results for the oscillating case are practically identical to those of the standard case, with only very small increases in drag power observed across all components.

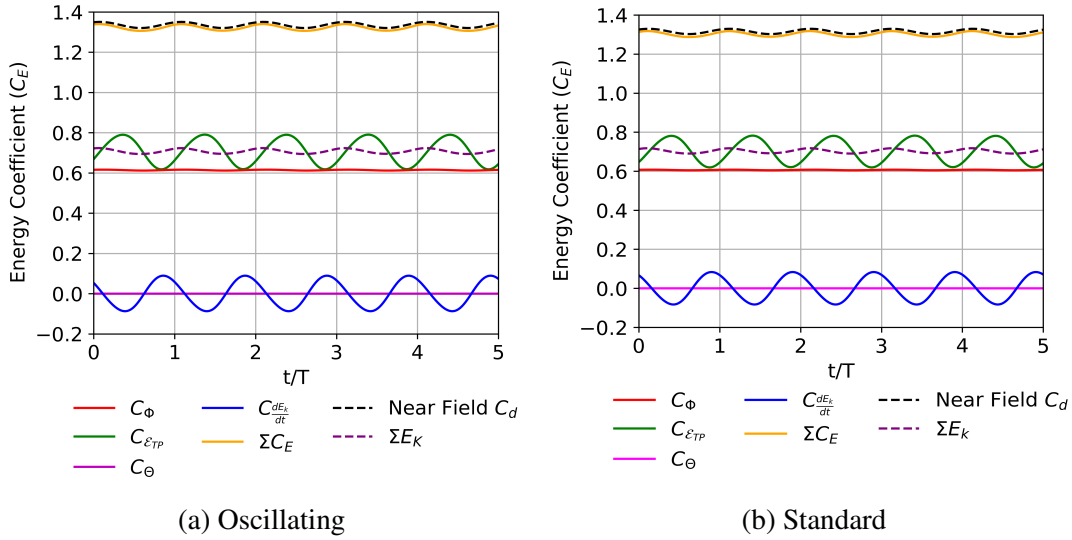


Figure 5.6: Energy Decomposition for CV1

Just as seen in the standard case, the change in kinetic energy with respect to time ($\frac{dE_k}{dt}$) and the mechanical energy ($\dot{\mathcal{E}}_{TP}$) are phase shifted by 180° and when summed together have a phase that matches the near-field C_d . It is noted that the mechanical energy term has a slightly larger amplitude in the oscillating case. To explore this further, the mechanical energy term was decomposed into its subcomponents, as shown in 5.7.

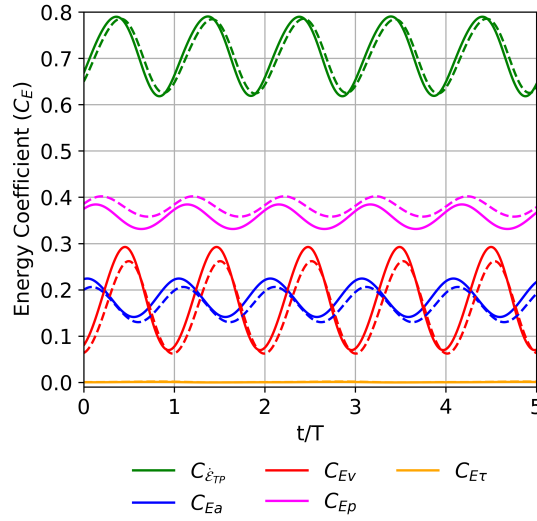


Figure 5.7: Mechanical Energy Decomposition for CV1
Solid Line: Oscillating Case, Dashed Line: Standard Case

The mechanical energy decomposition (Figure 5.7) shows small differences for the sub-components of the mechanical energy term in the oscillating case when compared against the standard case. The most notable difference is in the pressure boundary work term (C_{Ep}), which is lower in the oscillating case. This corresponds with visible increases in the oscillation amplitudes and increases in the time average mean values of the stream-wise (C_{Ea}) and transverse (C_{Ev}) energy flux's across the wake plane. The differences in the mean values of $\dot{E}_{TP}$, and the subcomponents are summarized in Table 5.6.

Table 5.6: Mechanical Energy Decomposition Time Averaged Mean Values

Case	$C_{\dot{E}_{TP}}$	C_{Ea}	C_{Ev}	C_{Ep}	$C_{E\tau}$
Oscillating	0.7072	0.1826	0.1775	0.3561	5.32×10^{-4}
Standard	0.7048	0.1679	0.1586	0.3775	9.32×10^{-4}

Although the total mechanical energy term shows only a very small change between the oscillating and standard cases, the results reveals a subtle shift in the mechanisms by which energy is deposited into the flow. Specifically, the oscillating motion leads to a reduction in the pressure-boundary contribution (C_{Ep}), while slightly increasing the viscous component ($C_{E\tau}$) and the kinetic energy fluxes (C_{Ea} and C_{Ev}). While these increases remain relatively small in magnitude, they suggest that the oscillatory motion redistributes the way energy is transferred across the wake plane, favouring viscous and kinetic mechanisms at the expense of pressure-driven mechanisms.

Looking at the results for CV3 (Figure 5.8), it is again seen that the oscillating case has extremely similar results to the standard case, again with only a very small increase in the terms making up the drag power. Just as what was discussed in the first test case, the position's of the vortex core's in the wake can be located in the mechanical energy and kinetic energy gradient terms, where the flattening of the peaks in the sinusoidal curves represents a region between two vortex cores.

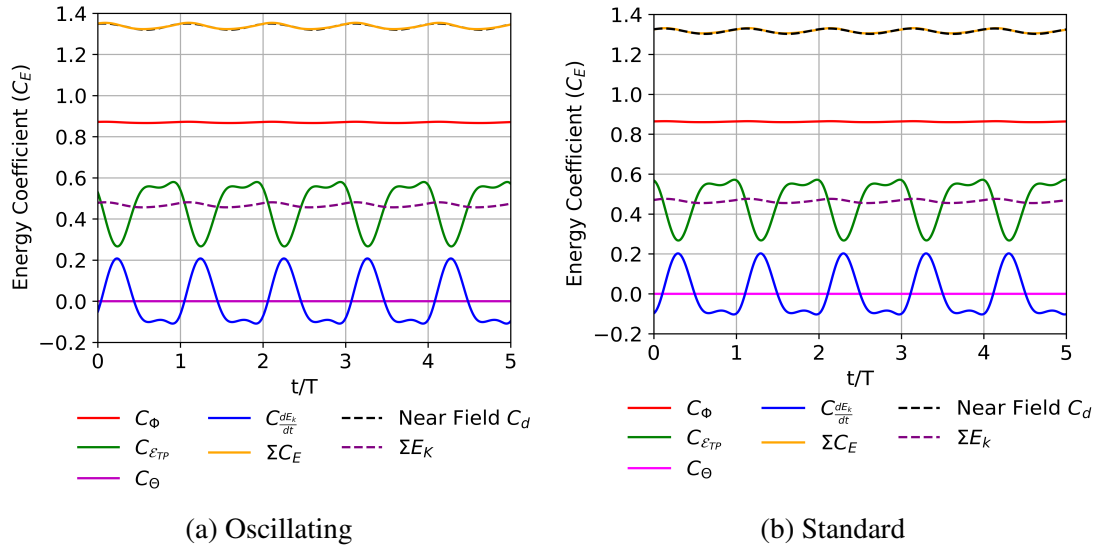


Figure 5.8: Energy Decomposition for CV3

It is also visible at CV3 that the amount of energy converted to irreversible losses (Φ) occurs at an extremely similar rate between the oscillating and standard cases. This trend continues to CV7 (Figure 5.9), where it is seen that the oscillating case again shows a similar pattern in the energy decomposition to the standard case and it is seen that the majority of the drag power now consists of irreversible viscous dissipation.

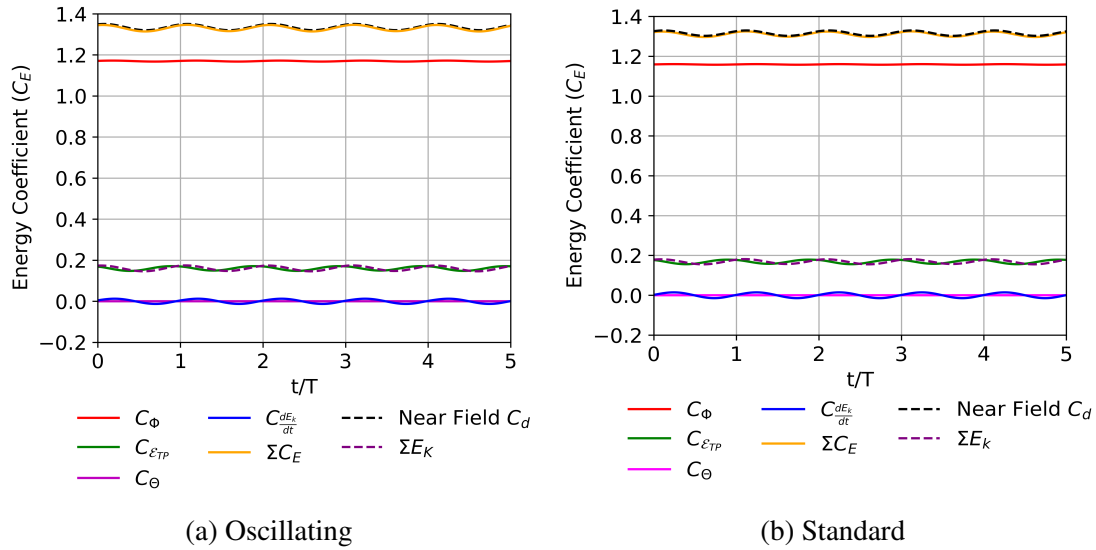


Figure 5.9: Energy Decomposition for CV7

5.2.4 Wake Analysis

The constant loss of mechanical energy to viscous dissipation across the wake planes is further analysed by looking at the mechanical energy components along the wake at a single time step as shown in Figure 5.10. Similar to what was seen in the standard test case, the majority of the mechanical energy deposited in the wake right by the cylinder is in the form of pressure work, which rapidly falls away becoming smaller than the transverse energy subcomponents by $x/D = 2$ already.

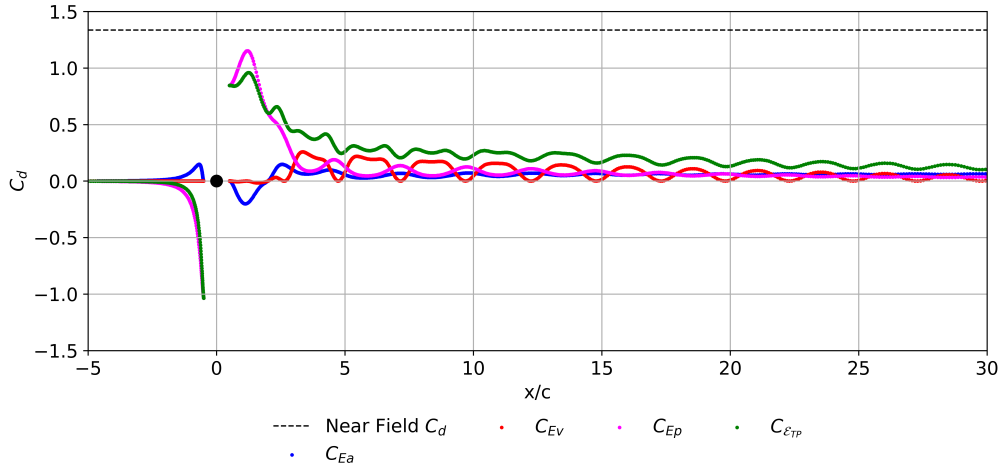


Figure 5.10: $\mathcal{E}_{TP}$ Wake Analysis

As was seen in the results for CV1-CV7, the mechanical energy keeps reducing as the distance along the wake increases. The losses due to viscous dissipation are quantified in Figure 5.11, where the $\mathcal{PER}$ term is shown for both the oscillating and standard cases.

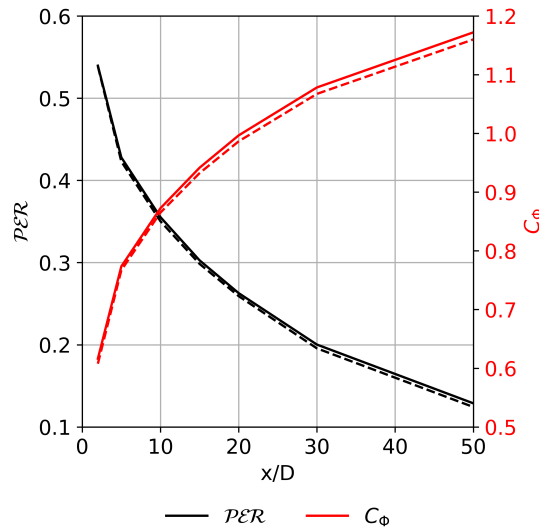


Figure 5.11: $\mathcal{PER}$
Solid Line: Oscillating Case, Dashed Line: Standard Case

Figure 5.11 shows that in the nearfield, that the oscillating case and the standard case have near identical amounts of usable energy and energy that has been irreversibly lost. The near identical $\mathcal{PER}$ values continues into the far-field as the mechanical energy is converted to viscous dissipation. It is seen that the oscillating case has a slightly higher viscous dissipation loss as you move further downstream, but this is balanced through the larger mechanical energy input as was discussed the decomposition section, resulting in the $\mathcal{PER}$ being consistent between the standard and oscillating cases in the far-field.

5.3 Test Case Conclusions

The combined test case has demonstrated the application of the energy and exergy balance to a configuration that merges naturally unsteady vortex shedding with additional body-induced motion. By introducing azimuthal oscillations into the baseline vortex-shedding regime, the case was designed to test whether the additional near-wall shear and unsteady acceleration mechanisms would fundamentally alter the balance between recoverable mechanical energy and irreversible viscous dissipation.

The results show that while the oscillatory motion does increase the overall mean drag force by approximately 1.5%, the unsteady wake dynamics remain characterised by the same characteristic frequency and oscillation amplitude observed in the standard case. This indicates that the imposed motion modifies the mean drag force without disrupting the dominant vortex-shedding mechanism responsible for unsteady drag fluctuations.

In the far-field balances, both the energy and exergy formulations once again yielded near-identical results, reinforcing the consistency of these two approaches. The oscillating case produced slightly elevated C_d values at all control volumes when compared against the static non-oscillating cylinder. Importantly, the decomposition analysis revealed that the increase in drag is not associated with a change in the energy transfer mechanisms, but rather with a subtle redistribution of contributions. Through the decompositions it is revealed that pressure-boundary work was reduced, while viscous and kinetic energy flux terms increased modestly. These changes reflect the increased shear-layer activity generated by the oscillatory wall motion, but their magnitudes were relatively small.

The wake analysis further confirmed this interpretation. The spatial development of the mechanical energy terms along the wake showed similar behaviour to the standard case, with pressure work dominant close to the cylinder but rapidly decaying downstream as kinetic and transverse fluxes became more prominent. The potential for energy recovery ($\mathcal{PER}$) was nearly identical between the two cases, suggesting that although oscillatory motion slightly increases viscous dissipation downstream, this is offset by a higher initial deposition of mechanical energy.

In summary, the combined case demonstrates that the addition of oscillatory wall motion primarily manifests as an incremental increase in mean drag and mean drag power, rather than as a fundamental change to the mechanisms of energy transfer and dissipation. The flow retains the same global dynamics as the baseline vortex-shedding case, with only minor shifts in how mechanical energy is partitioned between pressure-driven and viscous and kinetic contributions.

Chapter 6

Thesis Conclusions

The primary objective of this research was to investigate the applicability and accuracy of energy and exergy balance methods for evaluating drag in unsteady aerodynamic flows, and to compare their performance against the conventional momentum balance approach. While momentum-based methods remain widely used, their ability to provide insight into the mechanisms of energy transfer and irreversible dissipation is limited. By applying energy- and exergy-based formulations to a range of unsteady flow problems, this study aimed to assess their effectiveness in identifying the role of the different types of energy transfer mechanisms in the flow, and their robustness in transient flow conditions.

The first test case, involving a two-dimensional cylinder at $Re = 140$ undergoing periodic vortex shedding, provided a well-established benchmark for validation. The results confirmed that both the energy and exergy balances produced near-identical outcomes, closely matching the near-field drag values and the results obtained from the momentum balance. Importantly, the decompositions revealed how energy is progressively transferred into irreversible viscous losses downstream, while also demonstrating that the balances can successfully capture the periodic characteristics of the unsteady wake. Furthermore, the case demonstrated how the decompositions are able to identify the positions of phenomena such as vortices in the wake. This case therefore established the reliability of the balance formulations in a classical drag-producing flow.

The second test case examined an oscillating cylinder in a static fluid, where no net aerodynamic force or wake is generated. This posed a challenge for conventional near-field and momentum-based evaluations, but the energy and exergy balances provided valuable insight into how body-induced motion redistributes and dissipates energy within the flow. Although the $\mathcal{P}_S$ term could not be implemented into Fluent, the results showed good agreement with the analytical reference data. Moreover, the case demonstrated that even in the absence of net drag, the balances could still identify the mechanisms of energy transfer. This highlights their broader applicability to non-standard flow regimes where conventional force-based measures provide little insight.

The third and final test case combined the vortex shedding regime with an azimuthally oscillating cylinder wall, producing a more complex unsteady flow environment. The results demonstrated that while the oscillatory motion led to a small increase in mean drag

and drag power, the underlying trends in the energy and exergy balances remained consistent with those observed in the standard vortex shedding case. Decomposition analysis indicated subtle changes in the mechanisms of energy deposition, with reduced pressure contributions balanced by slightly increased kinetic and viscous components. Overall, however, the oscillatory motion did not fundamentally alter the energy transfer mechanisms within the flowfield.

While the present study has demonstrated the applicability of energy and exergy balances for drag evaluation in incompressible and laminar flow conditions, several avenues for further investigation remain.

First, future work should explore flow regimes where thermal and compressibility effects are more pronounced. The present incompressible cases limited the contribution of thermal effects, meaning that differences between the energy and exergy balances could not be identified. By examining high-speed or thermally driven configurations, the role of entropy generation, viscous heating, and compressibility on the balances could be better reviewed.

Second, the methodology should be extended to cases involving actual moving geometries, rather than moving wall boundary conditions. Although the oscillating wall condition used in this study served as a useful proxy, it does not fully capture the kinematics or grid deformation associated with real body motion. Implementing the balances in scenarios with deforming meshes or rigid-body motion would provide a more robust assessment of their accuracy under realistic motion-induced unsteadiness.

A further recommendation relates to the treatment of the surface pressure work term ($\mathcal{P}_S$). As noted in Test Case 2, the current Fluent implementation does not provide a direct means of obtaining this quantity, and attempts to reconstruct the stress tensor on the wall yielded inconsistent results. Fluent's documentation indicates that wall shear and stress terms are internally corrected to satisfy boundary conditions, but these reconstructions are not directly transferable to far-field formulations. Future work should focus on developing a reliable method for calculating $\mathcal{P}_S$ in Fluent.

Finally, while this study implemented the balances through named expressions, this approach was hindered by a lack of flexibility and the need for several workarounds to bypass Fluent's limitations. Issues included inconsistent behaviour of volume selections for integration and restrictions in report definition functionality. To streamline the process and improve reproducibility, future work should develop a set of modular and customisable User-Defined Functions (UDFs) capable of evaluating each balance directly. UDFs would provide greater control over data extraction, allow automation across multiple cases, and remove the reliance on named-expression workarounds, making the methodology more practical for widespread use.

In summary, this research has shown that energy and exergy balances provide a powerful framework for analysing aerodynamic drag and its underlying mechanisms, offering additional physical insight beyond that of the conventional momentum method. Although the

test cases considered were limited to laminar and incompressible flow regimes, the results highlight the capabilities of the far-field balances and their potential for wider application in more complex flow regimes. With further refinement of the implementation and extension to thermally and compressibly dominated flows, the approach developed here has the potential to become a valuable tool for advancing our understanding of aerodynamic losses and energy recovery in next-generation aerospace systems.

References

- [1] Toubin H. Prediction and Phenomenological Breakdown of Drag for Unsteady Flows; 2015.
- [2] Schlatter P, Li Q, Brethouwer G, Johansson A, Henningson D. In: Structure of a turbulent boundary layer studied by DNS. vol. 15; 2011. p. 9-14.
- [3] Smithsonian. Shock Waves; 2025. Accessed 02 August 2025. Available from: <https://howthingsfly.si.edu/media/shock-waves>.
- [4] Aeronautics N, Administration S. Airplane Vortex; 1990. Accessed 02 August 2025. Online. Available from: https://en.wikipedia.org/wiki/File:Airplane_vortex_edit.jpg.
- [5] Reist TA, Zingg DW. Aerodynamically optimal regional aircraft concepts: Conventional and blended wing-body designs. In: 52nd Aerospace Sciences Meeting. American Institute of Aeronautics and Astronautics Inc.; 2014. .
- [6] Zhao W, Zhang Y, Wu J. Energy-Based Aerodynamic Analysis on the Blended-Wing-Body Aircraft with Boundary Layer Ingestion. International Journal of Aerospace Engineering. 2022;2022.
- [7] Diamantidou DE, Hosain ML, Kyprianidis KG. Recent Advances in Boundary Layer Ingestion Technology of Evolving Powertrain Systems. Sustainability (Switzerland). 2022 2;14.
- [8] Doll U, Migliorini M, Baikie J, Zachos P, Ingo R, Melnikov S, et al. Non-intrusive flow diagnostics for unsteady inlet flow distortion measurements in novel aircraft architectures. Progress in Aerospace Sciences. 2022 04;130:100810.
- [9] Moirou NGM, Sanders DS, Laskaridis P. Advancements and prospects of boundary layer ingestion propulsion concepts. Progress in Aerospace Sciences. 2023 4;138.
- [10] Drela M, Kohler TJ. Power balance in aerodynamic flows. AIAA Journal. 2009 7;47:1761-71.
- [11] Destarac D, Vooren JVD. Drag/thrust analysis of jet-propelled transonic transport aircraft; Definition of physical drag components. Aerospace Science and Technology. 2004;8:545-56.
- [12] Berhouni I, Bailly D, Petropoulos I. On the Definition of Exergy in the Field of Aerodynamics. AIAA Journal. 2023 10;61:4356-66.

- [13] Arntz A, Atinault O, Merlen A. Exergy-based formulation for aircraft aeropropulsive performance assessment: Theoretical development. *AIAA Journal*. 2015;53:1627-39.
- [14] Cengel AY, Boles AM, Kanoglu M. *Thermodynamics An Engineering Approach*. 10th ed. Mc Graw Hill; 2024. Chapter 9.
- [15] Ruscio JP, Duplaa S, Aguirre MA, Aero A, Binder N. Exergy analysis of unsteady flow around an adiabatic cylinder in vortex shedding condition. In: 56th International Symposium of Applied Aerodynamics. 3AF; 2022. Available from: <https://www.researchgate.net/publication/359981414>.
- [16] Lamprakis I, Sanders DS, Laskaridis P. Fundamental Concepts of Boundary-Layer Ingestion Propulsion. *Journal of Aircraft*. 2025 5:1-22.
- [17] Sanders DS, Laskaridis P. Full-aircraft energy-based force decomposition applied to boundary-layer ingestion. *AIAA Journal*. 2020;58:4357-73.
- [18] ANSYS. *Fluent Theory Guide*. ANSYS Inc.; 2021. Accessed 24 May 2025. Available from: https://dl.cfdexperts.net/cfd_resources/Ansys_Documentation/Fluent/Ansys_Fluent_Theory_Guide.pdf.
- [19] Chorin AJ. A Numerical Method for Solving Incompressible Viscous Flow Problems. *Journal of Computational Physics*. 1997:118-25.
- [20] Kim J, Moin P. Application of a Fractional-Step Method to Incompressible Navier-Stokes Equations. *Journal of Computational Physics*. 1985:308-23.
- [21] Roshko A. On the Development of Turbulent Wakes from Vortex Streets. California Institute of Technology; 1954. Available from: <https://ntrs.nasa.gov/citations/19930092207>.
- [22] Franke R, Rodi W, Schonung B. Numerical Calculation of Laminar Vortex-Shedding Flow Past Cylinders. *Journal of Wind Engineering and Industrial Aerodynamics*. 1990:237-57.
- [23] Lienhard JH. *Synopsis of Lift, Drag and Vortex Frequency Data for Rigid Circular Cylinders*. Washington State University; 1966. Available from: engines.egr.uh.edu/sites/engines/files/talks/vortexcylinders.pdf.
- [24] Surmas R, Santos LOED, Philippi PC. Lattice Boltzmann simulation of the flow interference in bluff body wakes. In: *Future Generation Computer Systems*. vol. 20. Elsevier; 2004. p. 951-8.
- [25] Józsa TI. Analytical solutions of incompressible laminar channel and pipe flows driven by in-plane wall oscillations. *Physics of Fluids*. 2019 8;31.
- [26] Germaneau A. *Analytical Power Balance of Incompressible External Flow Driven by In-Plane Azimuthal Oscillations of a Cylinder*. Cranfield University; 2025.

-
- [27] ANSYS. How to calculate the shear stress of the fluid in ANSYS Fluent? ANSYS Inc.; 2016. Available from: <https://ansys13.ansys.com/KnowledgeArticles/Phase-1/2045478/2045478.pdf>.

Appendix A

Ethical Approval Letter



17 June 2024

Dear Mr Justin Du Plessis,

Reference: CURES/25709/2025

Project ID: 29137

Title: MSc Thesis

Thank you for your application to the Cranfield University Research Ethics System (CURES).

We are pleased to inform you your CURES application, reference **CURES/24831/2025** has been reviewed.

You may now proceed with the research activities you have sought approval for.

If you have any queries, please contact CURES Support.

We wish you every success with your project.

Regards,
CURES Team

Appendix B

Laminar Vortex Shedding Grid Information

B.1 Mesh Images

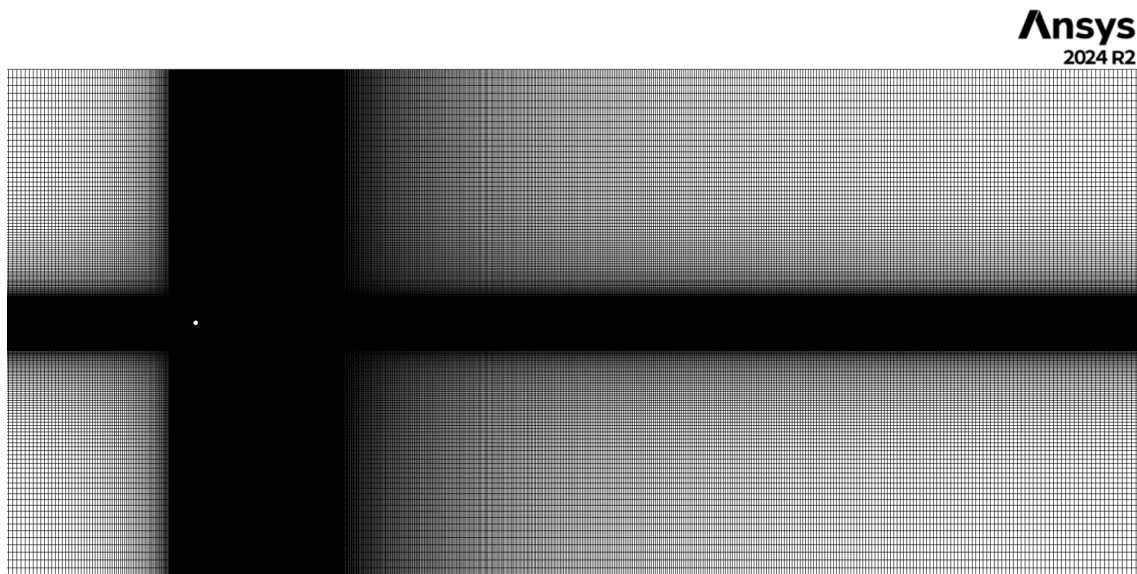


Figure B.1: Mesh Image for Laminar Vortex Shedding Test Case

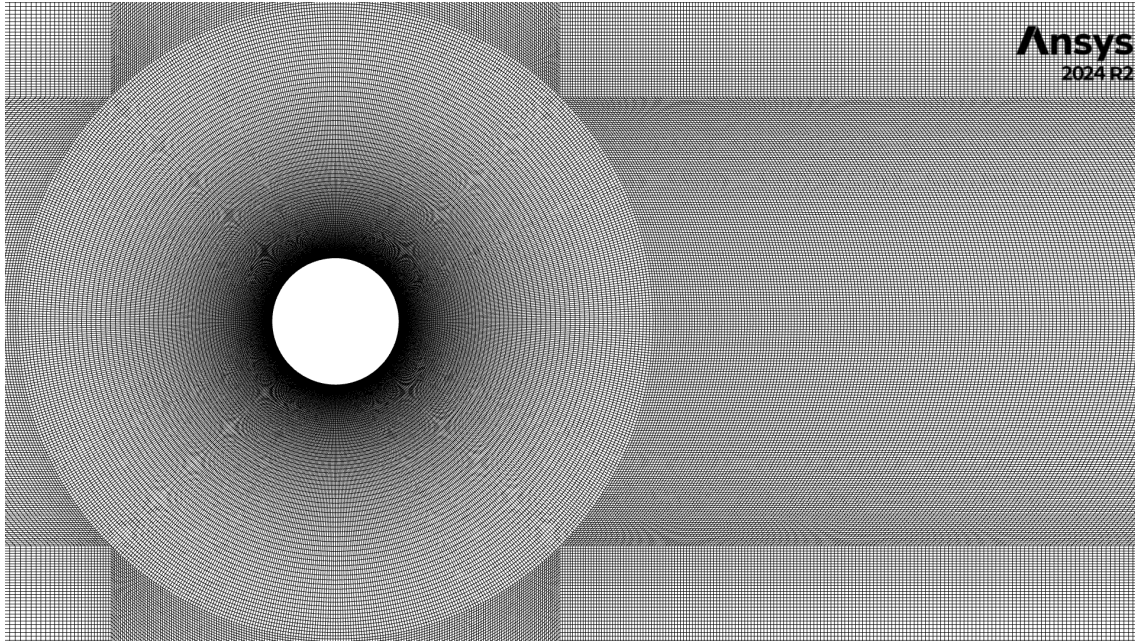


Figure B.2: Mesh Near-Field Region for Laminar Vortex Shedding Test Case

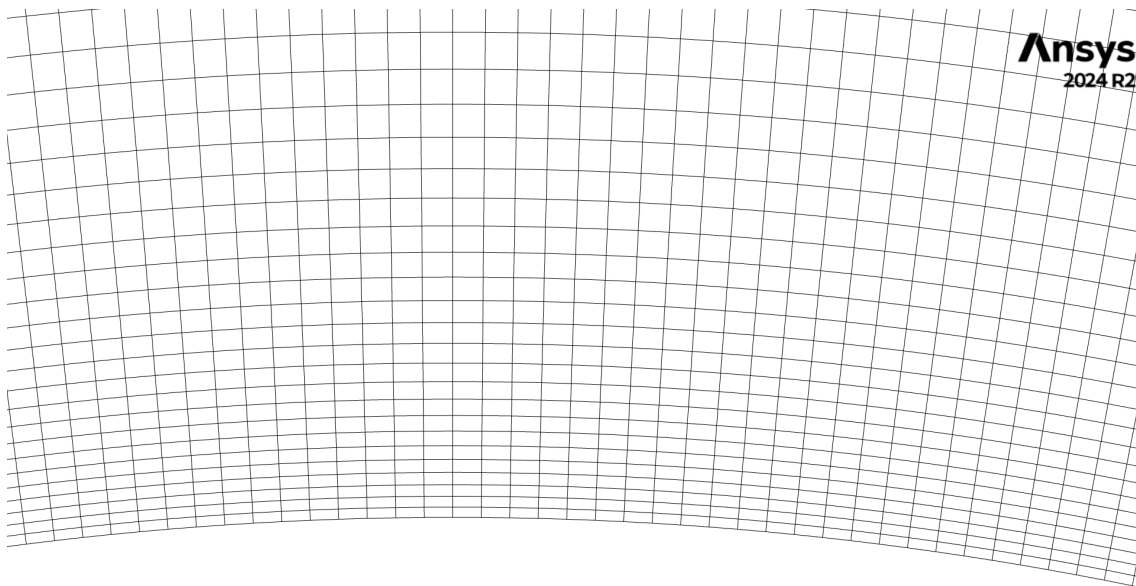


Figure B.3: Mesh Boundary Layer Region for Laminar Vortex Shedding Test Case

B.2 Mesh Quality Metrics

Table B.1: Mesh Quality Metrics

Orthogonal Quality			Equiangle Skewness		
Min	Avg	Max	Min	Avg	Max
0.71	0.97	1	1.36×10^{-6}	0.048	0.49

Appendix C

Oscillating Cylinder Mesh Information

C.1 Mesh Images

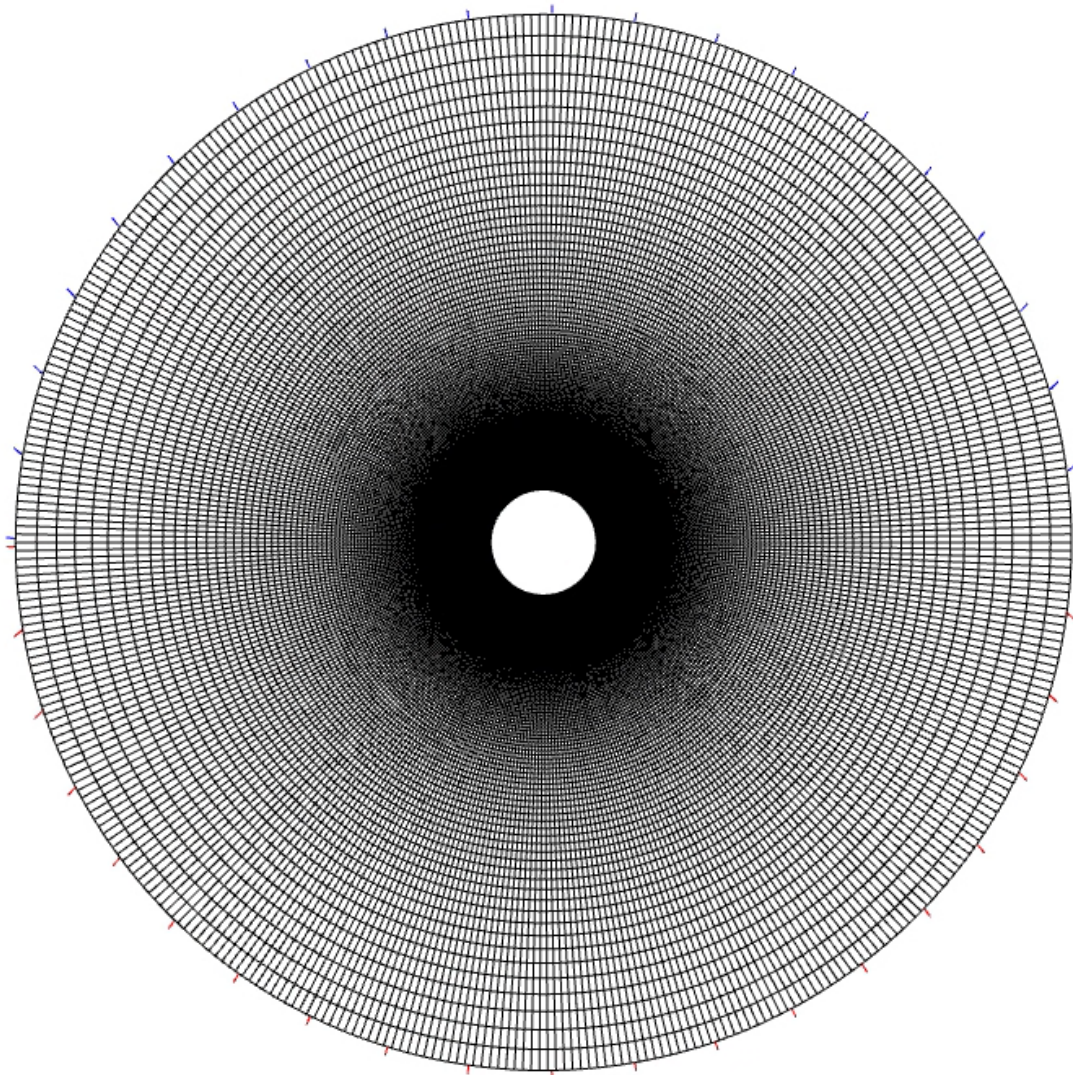


Figure C.1: Mesh Image for Oscillating Cylinder Test Case

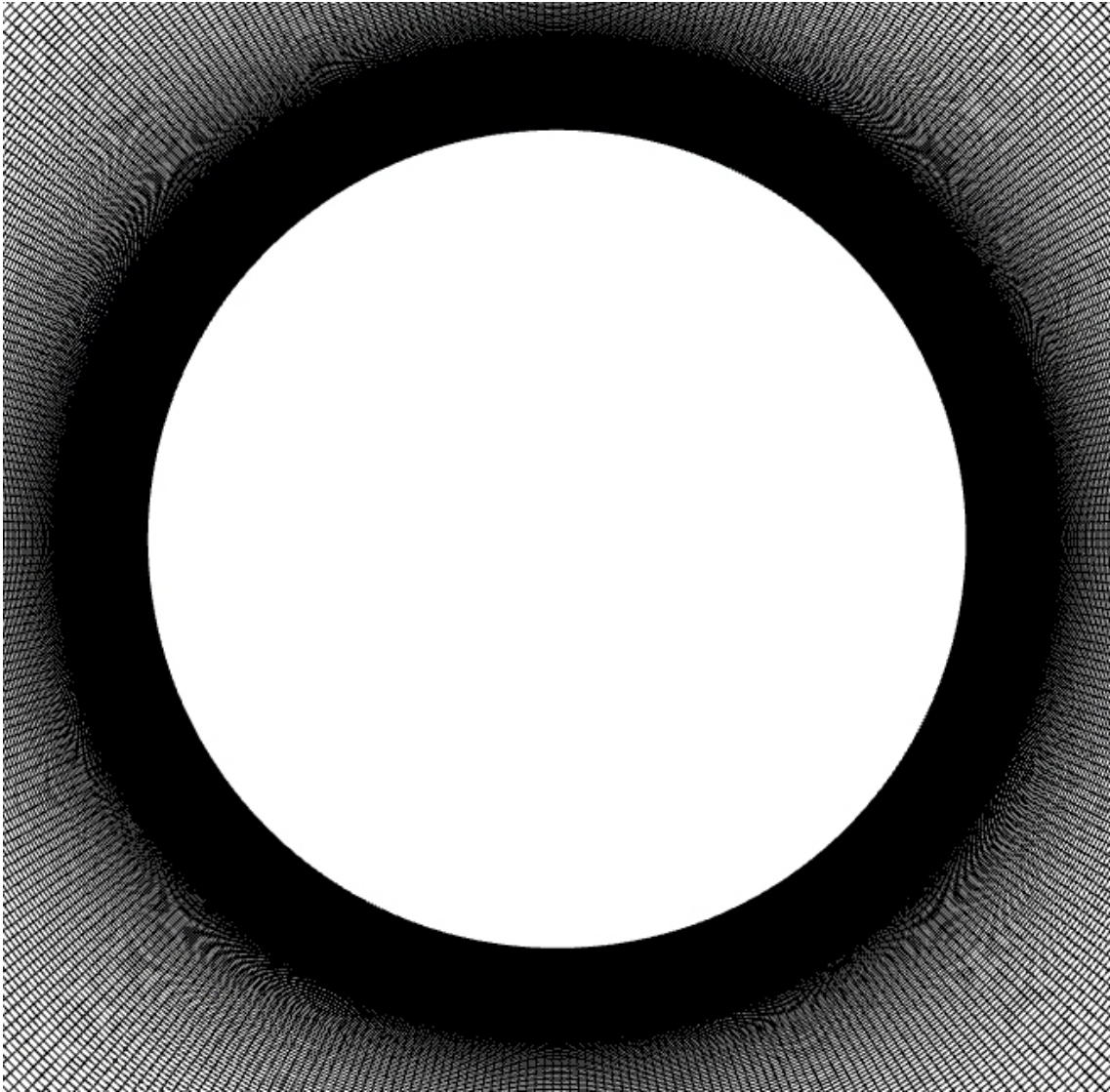


Figure C.2: Mesh Nearfield Region for Oscillating Cylinder Test Case

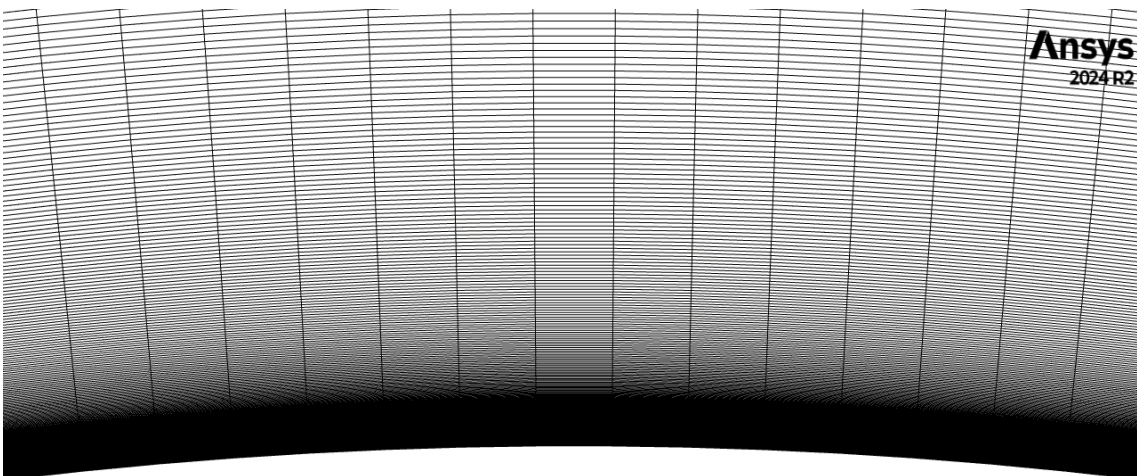


Figure C.3: Mesh Wall Region for Oscillating Cylinder Test Case

C.2 Mesh Quality Metrics

Table C.1: Mesh Quality Metrics

Orthogonal Quality			Equiangle Skewness		
Min	Avg	Max	Min	Avg	Max
0.98	0.99	1	0.005	0.005	0.005

Appendix D

Combined Test Case Exergy Plots

D.1 CV1 Decomposition

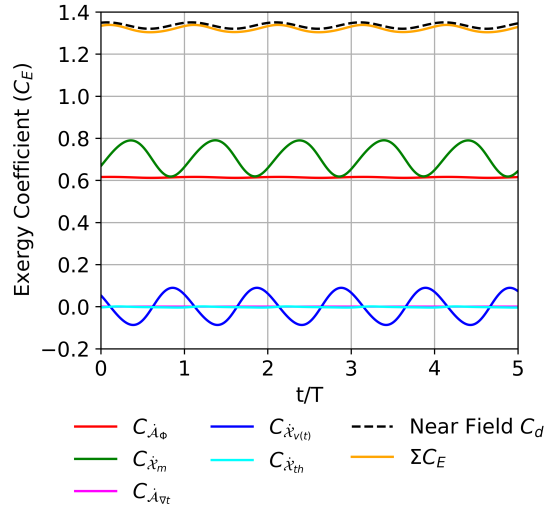


Figure D.1: Exergy Decomposition for CV1

D.2 CV3 Decomposition

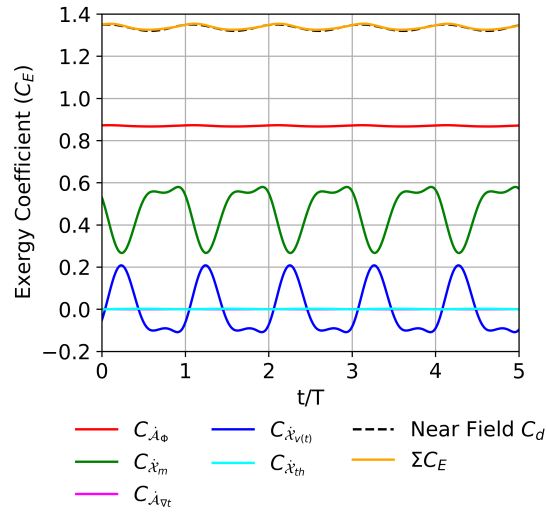


Figure D.2: Exergy Decomposition for CV3

D.3 CV7 Decomposition

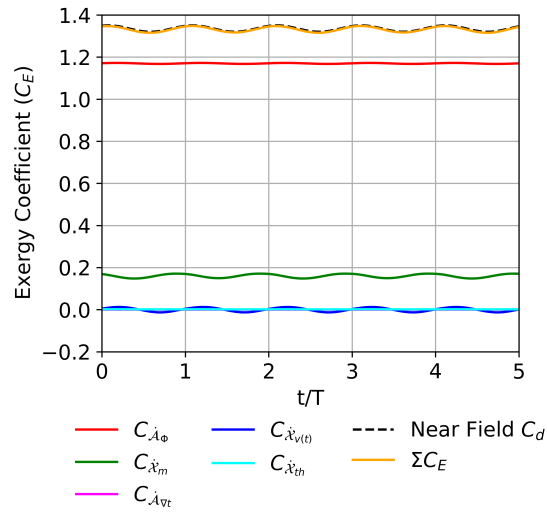


Figure D.3: Exergy Decomposition for CV7

Appendix E

Fluent Named Expressions

The following named expressions were used in the Fluent simulations to calculate the momentum, energy and exergy balances.

The components for $\bar{\tau}$ were obtained using Bouisinesq's Approximation.

Time derivatives were calculated externally in python.

The norm vector must be changed depending on the the direction for the integration surface.

The named expressions were then integrated when creating the report definition by choosing the correct integration type and the correct surface's.

E.1 Momentum Balance

E.1.1 Steady Term

$\text{Density} * (\text{Velocity} - V_{\text{inf}}) \cdot \text{dot}(\text{Velocity}, \text{Norm}) + \text{StaticPressure} * \text{Norm} - (\text{Tauxx} + \text{Tauxy})$

E.1.2 Unsteady Term

$\text{Density} * (\text{Velocity} - V_{\text{inf}})$

E.2 Energy Balance

E.2.1 $\mathcal{E}_{TP}$

$\text{dot}(0.5 * ((\text{Velocity.x} - V_{\text{infx}})^2 + (\text{Velocity.y} - V_{\text{infy}})^2) * \text{Density} * \text{Velocity} + \text{StaticPressure} * \text{Velocity} - (\text{vector}((\text{Tauxx} * (\text{Velocity.x} - V_{\text{infx}}) + \text{Tauxy} * (\text{Velocity.y} - V_{\text{infy}})), (\text{Tauyx} * (\text{Velocity.x} - V_{\text{infx}}) + \text{Tauyy} * (\text{Velocity.y} - V_{\text{infy}})), 0[\text{kg/s}^3])), \text{Norm})$

E.2.2 $\frac{dEk}{dt}$

$((\text{Velocity.x} - V_{\text{infx}})^2 + (\text{Velocity.y} - V_{\text{infy}})^2) / 2 * \text{Density}$

E.2.3 Θ

`-StaticPressure*VelocityDivergence*1[s^-1]`

E.2.4 Φ

`EffectiveViscosity*(StrainRate*StrainRate-
2/3*VelocityDivergence*VelocityDivergence*1[s^-2])-
(2/3*Density*TurbulentKineticEnergyk*VelocityDivergence*1[s^-1])`

E.3 Exergy Balance**E.3.1 $\mathcal{X}_s$**

`(SpecificInternalEnergy-einf)-Tinf*(SpecificEntropy-Sinf)
+Pinf*(1/Density-1/Rhoinf)`

E.3.2 $\dot{\mathcal{X}}_{th}$

`dot(Density*Xs*Velocity, Norm)`

E.3.3 $\dot{\mathcal{A}}_{VT}$

`Tinf*ThermalConductivity*(1/StaticTemperature*((dTd.x)^2+(dTd.y)^2))`

E.3.4 $\dot{\mathcal{A}}_{\Phi}$

`Tinf/StaticTemperature*Phi`

E.3.5 $\dot{\mathcal{X}}_v(t)$ Term 2

`Density*Xs`