



MARMARA UNIVERSITY
INSTITUTE FOR GRADUATE STUDIES
IN PURE AND APPLIED SCIENCES



SOLUTION-ADAPTIVE MESHLESS CFD METHODS FOR COMPLEX FLOWS IN

ENGINEERING APPLICATIONS

ÇİNAR LALOĞLU

Ph.D. THESIS

Department of Mechanical Engineering

Thesis Supervisor

Prof. Dr. Emre ALPMAN

ISTANBUL, 2024



MARMARA UNIVERSITY
INSTITUTE FOR GRADUATE STUDIES
IN PURE AND APPLIED SCIENCES



SOLUTION-ADAPTIVE MESHLESS CFD METHODS FOR COMPLEX FLOWS IN

ENGINEERING APPLICATIONS

ÇİNAR LALOĞLU

.....

Ph.D. THESIS

Department of Mechanical Engineering

Thesis Supervisor

Prof. Dr. Emre ALPMAN

ISTANBUL, 2024

ACKNOWLEDGEMENTS

I wish to seize this moment to convey my profound appreciation and thanks to those individuals and institutions that have been pivotal in the triumphant completion of my doctoral journey.

To begin with, my deepest sense of gratitude is reserved for my mentor, Professor Emre ALPMAN. Your consistent guidance, wisdom, and patience throughout my academic pursuit have been nothing short of monumental. Collaborating with you has been a rewarding experience, and your unwavering support over these years has been a cornerstone of my achievements.

My earnest thanks also go to the Department of Mechanical Engineering at Marmara University. The enriching and invigorating academic atmosphere provided by the department has been pivotal in shaping my intellectual pursuits. The resources and avenues provided by the institution played an essential role in expanding my academic horizons and refining my investigative aptitude.

In addition, a special note of appreciation to Professor Ralf Deiterding, from Southampton University's Aerodynamics and Fluid Mechanics division. Your invaluable assistance in the realm of simulations has been a guiding beacon, and I'm deeply thankful for it.

Lastly, but certainly not least, I would like to say my deepest love and gratitude to my family. Their unwavering support, encouragement, and belief in my abilities have been a constant source of strength throughout my academic journey.

February, 2024

Çinar LALOGLU

TABLE OF CONTENTS

ACKNOWLEDGEMENTS	i
TABLE OF CONTENTS	ii
ÖZET	iv
ABSTRACT	v
CLAIM FOR ORIGINALITY	vi
LIST OF SYMBOLS	vii
ABBREVIATIONS	ix
LIST OF FIGURES	x
LIST OF TABLES	xiii
1. INTRODUCTION	1
1.1. Overview of Computational Fluid Dynamics (CFD)	4
2. MESHLESS CFD METHODS	6
2.1. Lattice Boltzmann Method	7
2.2. Large - Eddy Turbulence Models	12
2.2.1. Smagorinsky Model	14
2.2.2. Dynamic Models	15
2.2.3. Scale-Dependent Models	15
2.2.4. Wall-Adapting Models	15
2.3. Applications of Lattice Boltzmann Method (LBM) in Fluid Dynamics	16
2.4. Advantages and Limitations of LBM	17
2.5. Introduction to Solution-Adaptive Methods	18
2.6. Adaptive Mesh Refinement in LBM	19
3. BENCHMARKS FOR LBM AND SOLUTION-ADAPTIVE METHODS	22
3.1. Simulation of the Flow past an Cylinder at high Reynolds Number	22
3.2. XFlow Simulation Setup	23
3.3. Evaluation of Mean Stream Velocity Field in Wake Region Using Lattice Boltzmann Method	24

3.4. Simulation of the Flow around an Oscillating Cylinder with Adaptive Lattice Boltzmann Methods	26
3.5. Evaluating of mesh refinement and solution time	28
3.6. Effecting of varying V_R in laminar flows	32
3.7. Effecting of varying V_R in turbulence flows	35
3.8. Darrieus VAWT Numerical Model Meshfree analysis using with LBM.....	37
3.8.1. Applied Turbulence Models	37
3.8.1.1. Large Eddy Simulation (LES)	38
3.8.1.2. Reynolds-Averaged Navier-Stokes (RANS)	39
3.8.1.3. Virtual Wind Tunnel Analysis and Boundary Conditions.....	39
3.8.1.4. Results and Discussion	42
4. FLUID-STRUCTURE INTERACTION.....	55
4.1. Application LBM-FSI analysis on Darieus-Wind turbine.....	56
5. CONTRIBUTIONS	59
5.1. Significance of Solution-Adaptive Meshless Methods.....	59
5.2. Integration with Lattice Boltzmann Method.....	59
5.3. Enhanced Handling of Complex Geometries	60
5.4. Application to Engineering Problems.....	61
5.5. Performance in Turbulent Flow Simulation	62
6. CONTRIBUTION TO WIND ENERGY RESEARCH.....	65
7. ADVANCING SIMULATION TECHNIQUES.....	66
8. CONCLUSIONS	67
REFERENCES	69
ÖZGEÇMİŞ.....	76

ÖZET

MÜHENDİSLİK UYGULAMALARINDA KARMAŞIK AKIŞLAR İÇİN ÇÖZÜM-UYARLI AĞSIZ HAD YÖNTEMİ

Çözüm-uyumlu ağsız hesaplamalı akışkanlar dinamiği (CFD) yöntemleri, mühendislik uygulamalarında karmaşık akışların simülasyonunda umut vadeden araçlar olarak öne çıkmaktadır. Bu doktora tezi, çözüm-uyumlu ağsız CFD yöntemlerinin geliştirilmesi ve incelenmesine odaklanmaktadır. Lattice Boltzmann yöntemi (LBM), bu yöntemlerin temelini oluştururken, mikroskobik modeller ve mezoskopik kinetik denklemleri kullanarak akışkan akışlarının temel fiziksel özelliklerini yakalamaktadır. Ağsız yöntemler, ağ yapılandırmasına ihtiyaç duymadan sistem cebirsel denklemlerinin oluşturulmasına imkan vererek LBM'yi tamamlar. Önceden tanımlanmış ağlara dayanmak yerine, alan düğümleri olarak adlandırılan dağılmış düğümler, problem alanını ve sınırlarını tanımlar. Bu araştırmanın temel katkısı, LBM ve ağsız yöntemlerin avantajlarını birleştiren çözüm-uyumlu ağsız CFD yöntemlerinin geliştirilmesidir. Bu yöntemler, akış özelliklerine dayalı olarak hesaplama ağını dinamik olarak uyumlu hale getirerek hesaplama kaynaklarının ilgi alanlarına optimal olarak dağıtılmasını sağlar. Bu uyarlanabilirlik, hesaplama maliyetlerini azaltırken doğruluğu artırarak mühendislik uygulamalarında karşılaşılan karmaşık akış problemleri için uygundur. Kapsamlı sayısal deneyler ve referans problemlerle yapılan doğrulamalar, önerilen çözüm-uyumlu ağsız CFD yöntemlerinin etkinliğini ve performansını göstermektedir. Elde edilen sonuçlar, bu yöntemlerin karmaşık akışları doğru bir şekilde simüle etme yeteneklerini ve hesaplama verimliliğini koruduğunu vurgulamaktadır. Araştırma bulguları, mühendislik uygulamalarında karmaşık akışların simülasyonunda güncel çalışmalara katkıda bulunarak geniş bir alan için pratik çözümler sunmaktadır. Anahtar kelimeler: çözüm-uyumlu ağsız CFD yöntemleri, lattice Boltzmann yöntemi, karmaşık akışlar, mühendislik uygulamaları, sayısal simülasyonlar, ağsız yöntemler, ağsız yöntemler, hesaplama verimliliği, doğruluk.

Anahtar kelimeler: Ağsız Hesaplamalı Akışkanlar Dinamiği Çözüm Yöntemi, Lattice-Boltzmann Method, Türbülans Modelleri

ABSTRACT

SOLUTION-ADAPTIVE MESHLESS CFD METHODS FOR COMPLEX FLOWS IN ENGINEERING APPLICATIONS

Solution-adaptive meshless computational fluid dynamics (CFD) methods have emerged as promising tools for simulating complex flows in various engineering applications. This doctoral thesis focuses on the development and investigation of solution-adaptive meshless CFD methods for accurately capturing the behaviour of complex flows. The lattice Boltzmann method (LBM) forms the basis of the proposed methods, employing microscopic models and mesoscopic kinetic equations to capture the essential physics of fluid flows. Meshless methods, such as meshfree or gridless methods, complement the LBM by enabling the establishment of system algebraic equations without relying on predefined meshes. Instead, scattered nodes, referred to as field nodes, define the problem domain and boundaries. The key contribution of this research is the development of solution-adaptive meshless CFD methods that combine the advantages of the LBM and meshless methods. These methods dynamically adapt the computational mesh based on flow features, ensuring optimal allocation of computational resources to areas of interest. This adaptive capability reduces computational costs and enhances accuracy, making them well-suited for handling complex flow problems encountered in engineering applications. Extensive numerical experiments and validation against benchmark problems demonstrate the effectiveness and performance of the proposed solution-adaptive meshless CFD methods. The results highlight their capability to accurately simulate complex flows while maintaining computational efficiency. The research findings contribute to advancing the state-of-the-art in simulating complex flows in engineering applications, offering practical solutions for a wide range of fields.

Keywords: Meshless CFD Methods, Lattice-Boltzmann Method, Turbulence models

CLAIM FOR ORIGINALITY

SOLUTION-ADAPTIVE MESHLESS CFD METHODS FOR COMPLEX FLOWS IN ENGINEERING APPLICATIONS

This doctoral thesis presents a unique contribution to solution-adaptive meshless computational fluid dynamics (CFD) methods for complex engineering flows. By combining the strengths of lattice Boltzmann method (LBM) and meshless techniques, this research proposes innovative solution-adaptive approaches. Integrating solution-adaptive meshless methods with LBM is a novel concept, overcoming limitations of traditional mesh-based approaches. The proposed methods dynamically adapt the computational mesh based on flow characteristics, improving accuracy and reducing computational costs. This adaptability is particularly beneficial for complex engineering flow problems. The research includes theoretical investigations, numerical experiments, and validations using reference problems. Results encounter the effectiveness and performance of the proposed methods in accurately simulating complex flows while maintaining computational efficiency. This original research significantly advances numerical simulations for engineering applications, providing practical solutions for complex flow phenomena. By pushing the boundaries of traditional CFD methods, this research opens new avenues for efficient and accurate simulations of complex flows. The combination of solution-adaptive meshless techniques and LBM offers promising prospects for the field. This doctoral thesis establishes innovative methodologies, contributing to the advancement of computational fluid dynamics in engineering.

LIST OF SYMBOLS

ω_i	: isotropy
C_x	: LES model dependent constant
c_s	: sound speed
e_{ia}	: discrete velocity
f^{eq}	: equilibrium distribution function
f_N	: natural frequency
f_t	: time frequency
ν_o	: molecular viscosity
$\bar{\omega}$	: <i>LES model operator</i>
Δx	: step size
A	: cross-sectional area
C_D	: drag coefficient
C_L	: lift coefficient
p	: pressure
Q	: torque
Re	: Reynolds number
t	: time
u	: macroscopic velocity
U_{avg}	: average velocity

v	: velocity
w	: collision frequency
x	: position
$y(t)$	: translational movement
ν	: kinematic viscosity
ρ	: density
τ	: characteristic time
Ω	: collision operator
D	: diameter
St	: Strouhal number
θ	: azimuthal angle
$\theta(t)$	: rotational movement

ABBREVIATIONS

AMROC	: Adaptive mesh refinement software system
BGK	: Bhatnagar, Gross and Krook
CFD	: Computational Fluid Dynamics
D2Q9	: Two dimensional discretization with nine distributions
D3Q19	: Three dimensional discretization with nineteen distributions
EFG	: Element-Free Galerkin method
FSI	: Fluid-Structure Interaction
LBM	: Lattice-Boltzmann Method
LES	: Large-Eddy Simulation
MPS	: Moving Particle Semi-implicit method
PIV	: Particle image velocimetry
RANs	: Reynolds-Averaged Navier-Stokes
SPH	: Smoothed Particle Hydrodynamics
TSR	: Tip-speed ratio
VAWT	: Vertical Axis Wind Turbine
VIV	: Vortex-induced vibrations
WALE	: Wall-Adaptive Large-Eddy

LIST OF FIGURES

Figure 2.1.	Discrete velocity directions for D2Q9 model in a computational cell.....	10
Figure 2.2.	The lattice structure of D3Q27 model.....	12
Figure 2.3.	Displaying the partial density distribution functions is crucial when understanding the data transfer at the junction between coarse and fine boundaries. The bold black demarcations symbolize a physical boundary. (a) Coarse data points extending into the fine grid; (b) incoming interpolated fine points in peripheral zones (top) and outgoing points in these zones post two fine-level transport maneuvers (bottom); (c) computed averages that supplant coarse data before another update cycle adjacent to the boundary.	20
Figure 3.1.	Computational area – dimensions	22
Figure 3.2.	XFlow Lattice Structure with using Adaptive Refinement.....	23
Figure 3.3.	Velocity X contours	25
Figure 3.4.	Velocity X in the far wake	25
Figure 3.5.	Motion imposed on cylinder	27
Figure 3.6.	Case 3: Evolution of dynamic mesh refinement (upper row) and distribution to four processors over time (lower row) in AMROC.	29
Figure 3.7.	Mesh refinement of Case 2 on a) AMROC b) XFlow for laminar flows and c) experimental contour (Nazarinia et al., 2011).....	29
Figure 3.8.	Mesh refinement of Case 2 on a) AMROC b) XFlow for laminar flows and c) experimental contour (Nazarinia et al., 2011).....	30
Figure 3.9.	Overall solution time depending on number of cores in laminar flow for $t=3.33s$ on AMROC	31
Figure 3.10.	Overall solution time depending on number of cores in turbulence flow for $t=1.66 s$ on AMROC	31
Figure 3.11.	Vorticity distributions for laminar flows at $U=0.0606 m/s$ and $Re= 1322$ a) XFlow and b) AMROC for Case 2, $V_R= 0.5$	33
Figure. 3.12.	Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 2, $V_R= 0.5$	33

Figure 3.13. Vorticity distributions for laminar flows at $U=0.0606$ m/s and $Re=1322$ a) XFlow and b) AMROC for Case 3, $V_R=1$	34
Figure 3.14. Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 3, $V_R=1$	34
Figure 3.15. Vorticity distributions for laminar flows at $U=0.303$ m/s and $Re=6310$ a) XFlow and b) AMROC for Case 2, $V_R=0.5$	35
Figure 3.16. Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 2, $V_R=0.5$	35
Figure 3.17. Vorticity distributions for laminar flows at $U=0.303$ m/s and $Re=6310$ a) XFlow and b) AMROC for Case 3, $V_R=1$	36
Figure 3.18. Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 3, $V_R=1$	36
Figure 3.19. CAD geometry of Darrieus Vertical Axis Wind Turbine	37
Figure 3.20. Cell Configuration of Darrieus Vertical Axis Wind Turbine (5 000 000 lattice cells).	40
Figure 3.21. Adaptive Wake Refinement for Darrieus Vertical Axis Wind Turbine	41
Figure 3.22. Variations in torque coefficients according to turbulence models and experimental results (Lee et al. 2015) for 2 500 000 lattices.	43
Figure 3.23. Variations in torque coefficients according to turbulence models and experimental results (Lee et al. 2015) for 5,000,000 lattices.	44
Figure 3.24. Variations in power coefficients according to turbulence models and experimental results (Lee et al. 2015) for 5,000,000 lattices	45
Figure 3.25. Velocity contours around VAWT in different turbulence models for TSR 1 and $\theta = 77^\circ$	46
Figure 3.26. Turbulence Intensity (%) around VAWT in different turbulence models for TSR 1 and $\theta = 77^\circ$	47
Figure 3.27. Velocity contours around VAWT in different turbulence models for TSR 1.2 and $\theta = 90^\circ$	48
Figure 3.28. Turbulence Intensity (%) around VAWT in different turbulence models for TSR 1.2 and $\theta = 90^\circ$	49

Figure 3.29. Velocity contours around VAWT in different turbulence models for TSR 1.6 and $\theta = 10^\circ$	50
Figure 3.30. Turbulence Intensity (%) around VAWT in different turbulence models for TSR 1.6 and $\theta = 180^\circ$	51
Figure 3.31. Vorticity coloured by field Velocity for LES-WALE turbulence model for TSR 1.2.....	52
Figure 3.32. Vorticity coloured by field Velocity for RANS-Spalart-Allmaras turbulence models for TSR 1.2.	53
Figure 3.33. Vorticity coloured by field Velocity for LES-Smagorinsky turbulence models for TSR 1.2.....	53
Figure 4.1. Power coefficient comparison between Experimental and LBM-FSI (Lee et al 2015).....	57
Figure 4.2. Contours velocity of Daireus wind turbine by using LBM- Xflow.	58
Figure 4.3. Angular velocity changes from self-starting position on Darieus wind turbine.....	58
Figure 5.1. AMROC implements the AMR method in C++ and can be used directly in C++ or with Fortran functions called from C++ interface objects.(Deiterding R.).....	64

LIST OF TABLES

Table 3.1.	Parameters of the calculations Lausova et. al.	23
Table 3.2.	Applied values of At and A_θ	27
Table 3.3.	Laminar and turbulence flows parameters for Case 2 and Case 3	31
Table 3.4.	Overall CPU time (s) with respect to number of lattices in different turbulence models	42



1. INTRODUCTION

The study of flow past circular cylinders has garnered significant attention in both experimental and numerical research due to the inherent symmetry of the cylinder geometry and its relevance to various industrial applications. Numerous investigations have been conducted to understand the fluid dynamics phenomena associated with the flow around cylinders, including separation, vortex shedding, and the transition to turbulence. These phenomena play a crucial role in determining flow-induced forces such as drag and lift forces, as well as pressure coefficients (Sarpkaya, 2004; William et al., 2004).

In recent years, there has been a growing focus on evaluating the effect of oscillations, both transverse and rotational, on the wakes generated by cylinders (Tokumaru et al., Filler et al., 1991). While the effect of translational oscillations on vortex-induced vibrations has been extensively studied, the investigation of rotational oscillations and their impact on wakes has gained attention more recently. Understanding the behavior of the oscillating cylinder, especially at higher Reynolds numbers, necessitates fine resolution and the use of a fine mesh in CFD analyses (Jiange et al., 2016).

LBM offers several advantages over traditional CFD methods, particularly in handling complex geometries and boundaries. Researchers have shown that LBM is highly suitable for simulating intricate geometries and complex boundary conditions (Huang et al., 2022; Zhang et al., 2023). The flexibility of the lattice-based grid structure in LBM allows for easy representation of irregular geometries and accurate capture of flow features.

Moreover, LBM excels in simulating multi-scale and multiphase flows. Recent studies have demonstrated the capability of LBM in microfluidics applications (Chen et al., 2022) and multiphase flows with phase change phenomena (Tang et al., 2023). The mesoscopic nature of LBM enables accurate modelling of flow behaviour at different length scales, leading to improved accuracy in numerical predictions. Efficient parallelization is another strength of LBM, enabling high-performance computing for large-scale simulations. This is particularly advantageous for turbulent flow simulations and aerodynamic analyses.

Researchers have successfully employed LBM in simulating turbulent flows with excellent scalability and computational efficiency (Li et al., 2022; Wang et al., 2023). Furthermore, LBM provides a natural framework for incorporating complex physics, such as thermal effects and fluid-structure interactions. Recent work has shown the capability of LBM in accurately capturing heat transfer phenomena in porous media (Liu et al., 2022) and simulating fluid-structure interactions (Wu et al., 2023). These advancements in modelling complex physics contribute to more realistic and comprehensive numerical results.

LBM has played a significant role for enhancing the quality of numerical results in CFD applications. Its ability to handle complex geometries, simulate multi-scale and multi-phase flows, efficient parallelization, and incorporation of complex physics makes it a promising choice for researchers.

In this doctoral research, we advocate for the integration of a solution-adaptive methodology using the LBM within meshless CFD frameworks. Leveraging particle-centric paradigms, like LBM, paves the way for distinctively discrete solutions in fluid flow dynamics. What sets LBM apart is its ability to seamlessly converge micro-scale and macro-scale understandings. Rooted deeply in microscopic modelling and mesoscopic kinetic theory, LBM facilitates the formulation of simplified yet profound kinetic representations. This, in turn, ensures that macroscopic outcomes align perfectly with anticipated macroscopic equations. (Jiange et al., 2016; Nazarinia et al., 2011).

The LBM, in combination with solution-adaptive meshless methods, enables the accurate simulation of complex flows around circular cylinders. Meshless methods, such as the meshfree or gridless approach, complement the LBM by eliminating the need for predefined meshes. Instead, scattered nodes, known as field nodes, define the problem domain and boundaries. By dynamically adapting the computational mesh based on flow features, the proposed solution adaptive meshless CFD methods allocate computational resources optimally to areas of interest. This adaptivity reduces computational costs while maintaining accuracy, making them well-suited for analyzing complex flow problems encountered in engineering applications (Park et al., 2011; Musavi et al., 2016).

Additionally, in a study conducted by Nazarinia et al. (2011), the physical mechanism behind the oscillating cylinder, involving both translational and rotational oscillations, was investigated through experimental analysis using particle image velocimetry (PIV). The purpose of their study was to characterize the wake of forced cylinder flow at a Reynolds number of 1322. To enhance the accuracy of their findings, a solution-adaptive method was employed, which included dynamic mesh adaptation to increase the local resolution near the cylinder. This method was implemented using the fully parallelized block-structured adaptive mesh refinement software system AMROC (Deiterding, 2011). In high Reynolds number situations, AMROC incorporated Smagorinsky's eddy viscosity approximation to model sub-grid scale turbulence.

The aerodynamic characteristics of a small-scale Archimedes spiral wind turbine blade are comprehensively analyzed through numerical simulations and experimental validation. The numerical analysis is conducted using XFlow, a software based on the lattice Boltzmann method (LBM), renowned for its ability to accurately simulate fluid dynamics phenomena. Specifically, the Wall-Adapting Local Eddy-viscosity (WALE) turbulence model is employed for its favorable properties in both laminar and turbulent flows, particularly near solid bodies and walls. To validate the numerical simulations, Particle Image Velocity (PIV) techniques are utilized, providing empirical evidence of the aerodynamic features of the spiral wind turbine. The velocity profiles obtained from XFlow simulations closely match the trajectory and magnitude of tip vortices observed in experimental results, confirming the accuracy of the numerical predictions. Furthermore, the researchers compare their findings with those obtained using XFlow software, highlighting the software's capability to accurately simulate complex flow phenomena. XFlow's unique particle-driven, Lagrangian methodology rooted in LBM principles enables the simulation of diverse problems, including aerodynamics, aero-acoustics, and fluid-structure interactions (Safdari et al., 2015).

This interdisciplinary study provides valuable insights into the physical behavior of the Archimedes spiral wind turbine blade and its wake characteristics. By employing advanced techniques such as PIV and solution-adaptive methods with dynamic mesh adaptation, the

researchers gain a deeper understanding of flow dynamics and associated phenomena. The integration of experimental validation and numerical simulations using XFlow software enhances the reliability and robustness of the findings, contributing to the existing knowledge base in fluid dynamics research (Safdari et al., 2015).

This doctoral thesis aims to contribute to the development and investigation of solution-adaptive meshless CFD methods based on LBM, specifically tailored to accurately simulate the behaviour of complex flows. The research findings will address the limitations of traditional mesh-based CFD methods and provide practical solutions for a wide range of engineering applications. Extensive numerical experiments and validation against benchmark problems will be conducted to lead the effectiveness and computational efficiency of the proposed solution-adaptive meshless CFD methods. Ultimately, this research endeavours to advance the state-of-the-art in simulating complex flows, providing valuable insights for various fields of study.

1.1. Overview of Computational Fluid Dynamics (CFD)

CFD is a powerful tool in the field of fluid mechanics, enabling the simulation and analysis of complex fluid flow phenomena. CFD involves the numerical solution of governing equations, typically the Navier-Stokes equations, which describe the fundamental principles governing fluid motion. These equations, along with additional equations for turbulence modelling or other specific physical effects, are discretized and solved numerically to obtain approximate solutions.

Traditional CFD methods rely on the use of computational grids or meshes to discretize the physical domain. However, in recent years, meshless and mesh-free methods have gained attention as alternative approaches. These methods eliminate the need for structured or unstructured grids, instead utilizing particle-based or function-based representations of the fluid domain. Examples of meshless methods include Smoothed Particle Hydrodynamics (SPH), Moving Particle Semi-implicit (MPS) method, and the Element-Free Galerkin (EFG) method. Mesh-free methods, on the other hand, utilize mathematical functions or

shape functions to represent the solution field without requiring a predefined mesh Barrios, et.al., 2016).

Meshless and mesh-free methods provide several advantages when compared to traditional mesh-based methods in certain scenarios. They provide greater flexibility in handling complex geometries and evolving interfaces, as they do not require explicit mesh generation or remeshing. These methods also exhibit better adaptivity to local features and can handle large deformations more effectively. Additionally, they offer efficient parallelization potential, enabling faster computations on parallel computing architectures.

The choice between mesh-based and meshless/mesh-free methods depends on the specific problem at hand and the desired accuracy and computational efficiency. Both approaches have their strengths and limitations, and researchers continue to explore and develop advanced techniques that combine the advantages of both methods.

2. MESHLESS CFD METHODS

Meshless Computational Fluid Dynamics (CFD) methods have emerged as an alternative approach to traditional mesh-based methods for solving fluid flow problems. Unlike mesh-based methods that require the generation and manipulation of a computational grid, meshless methods rely on a set of scattered data points or particles to discretize the flow domain. This unique characteristic of meshless methods offers several advantages in terms of flexibility and adaptability.

One of the key advantages of meshless methods is their ability to handle complex geometries and arbitrary mesh configurations. Traditional mesh-based methods often face challenges in accurately representing irregular or highly intricate geometries, requiring extensive grid generation and refinement efforts. In meshless methods, the need for a structured grid is eliminated, allowing for a more straightforward representation of complex geometries (Liu et al., 2018). This makes meshless methods particularly suitable for problems involving fluid flows in intricate domains or situations where the flow domain changes dynamically.

Additionally, meshless methods offer adaptivity in terms of spatial resolution. In traditional mesh-based methods, the accuracy of the solution is limited by the grid resolution, necessitating grid refinement in areas of interest. Meshless methods, on the other hand, do not rely on a fixed grid, but rather on scattered data points. This allows for the adaptation of spatial resolution based on the local flow features, resulting in efficient and accurate simulations (Liu et al., 2020). Furthermore, meshless methods can naturally handle dynamic remeshing, which is particularly advantageous for problems with evolving flow domains or moving boundaries.

Meshless methods exhibit excellent scalability and parallelizability, making them well-suited for high-performance computing. The absence of a structured grid simplifies the computational operations, enabling efficient parallel processing and distribution of computational tasks among multiple processors or computing nodes (Rabczuk et al., 2010).

This scalability is crucial for tackling large-scale and computationally demanding fluid flow problems.

It is worth noting that meshless methods also have some limitations. The accuracy and convergence behaviour of meshless methods depend on the choice of the radial basis functions or other interpolation schemes used to reconstruct the solution from the scattered data points. The accuracy can vary depending on the choice of interpolation functions and their parameters, requiring careful selection and calibration (Liu et al., 2018). Additionally, the computational cost of meshless methods can be higher compared to traditional mesh-based methods, especially for large-scale simulations.

Meshless CFD methods offer several advantages, including their ability to handle complex geometries, adaptivity in spatial resolution, and scalability in high-performance computing. These features make meshless methods a promising alternative for simulating fluid flows, particularly in situations involving intricate geometries or evolving flow domains.

2.1. Lattice Boltzmann Method

A recent advancement in computational fluid dynamics is the Lattice Boltzmann Method (LBM), which offers an alternative approach to solving fluid flow problems. Unlike traditional CFD methods that rely on structured or unstructured grids, LBM is a meshless method that operates on a lattice model. In LBM, the fluid is represented by distribution functions defined on a lattice grid, where each lattice node corresponds to a discrete fluid particle. The evolution of these distribution functions is governed by discrete Boltzmann equations, which describe the streaming and collision processes of the fluid particles. By simulating the distribution functions' dynamics, LBM captures the macroscopic behavior of the fluid flow. One of the key advantages of LBM is its ability to handle complex boundary conditions and geometries with ease. The lattice structure simplifies the implementation of various boundary conditions, such as no-slip walls or inflow/outflow conditions, without the need for complex mesh generation. Additionally, LBM exhibits excellent parallelizability, enabling efficient computations on parallel computing architectures.

LBM has demonstrated its effectiveness in simulating a wide range of fluid flow problems, including laminar and turbulent flows, multiphase flows, and flows in complex geometries. It has shown promise in simulating flows with complex physics, such as free-surface flows, porous media flows, and fluid-structure interactions. Moreover, LBM can be extended to incorporate additional physics, such as thermal effects or chemical reactions, making it suitable for studying various engineering applications (Guo et al., 2013).

The LBM adopts a mesoscopic perspective, simulating fluid dynamics based on interactions among particles arranged on a structured grid. The fluid flow is existed by the collective behaviour of these particles. The Boltzmann equation produce as single particle distribution function f takes the following form (Succi, 2001):

$$\frac{\partial f}{\partial t} + \vec{v} \cdot \nabla f = \Omega(f) \quad (2.1)$$

The function $f(\vec{x}, \vec{v}, t)$ corresponds to the likelihood of identifying particles possessing the granular, microscopic velocity $\vec{c}$ at the specific location x and moment t . The collision operator, $\Omega(f)$ governs the temporal alteration in the distribution function f during a particle collision. In a plethora of studies focused on the kinetic theory underlying fluid dynamics, the collision term Ω_i^{BGK} is typically symbolized as $\Omega(f)$. This term usually manifests as a convoluted integral expression. A simplification termed "BGK", conceived by Bhatnagar, Gross, and Krook, and later incorporated by Qian et al. in 1992, substitutes the intricate integral component with a more straightforward representation, as depicted in Equation 2.3 (Bhatnagar et al., 1954).

$$\Omega(f) = \frac{1}{\tau} (f_i^{eq} - f_i) \quad (2.2)$$

This mechanism simulates the collision's impact by letting the distribution function f relax towards a Maxwellian equilibrium state f_i^{eq} . Here, the parameter τ stands as an emblematic time, orchestrating the rhythm at which the distribution function gravitates towards equilibrium. A first step for discretization of the Boltzmann equation is the restriction of the continuous velocity space into a finite set of velocities. $\vec{V} = \{\vec{e}_0, \vec{e}_1, \dots, \vec{e}_{n-1}\}$ thus, the

distribution function $f(\vec{x}, \vec{v}, t)$ is reduced $f_i(\vec{x}, t)$ which describes the distribution along a finite lattice.

The Boltzmann equation under the BGK approximation becomes LBGK the discrete Boltzmann equation given by Qian et al., (1992)

$$\frac{df_i}{dt} + e_i \cdot \nabla f_i = \frac{1}{\tau} (f_i^{eq} - f_i) \quad (2.3)$$

The f^{eq} in Eq. 3 is the equilibrium distribution function for density. This equation is discretized with a step size Δx and a time step Δt which are linked by the following relation: $\frac{\Delta x}{\Delta t} = e_i$. This discretization ensures that the particles in a node x move in time Δt toward a neighbour node $x + e_i \Delta t$ along the vector $\vec{e}_i$ and equation is Qian et al., (1992) ;

$$f_i(\vec{x} + \vec{e}_i \Delta t, t + \Delta t) - f_i(\vec{x}, t) = \frac{1}{\tau} (f_i^{eq} - f_i(\vec{x}, t)) \quad (2.4)$$

1.Streaming: During each time step Δt , the particles undergo movement along the lattice bonds to adjacent lattice nodes. The distribution function $f_i^*(\vec{x}, t)$ propagates along the vector $\vec{e}_i$, and at the updated time $t + \Delta t$, the distribution function at a neighbouring $f_i(\vec{x} + \vec{e}_i \Delta t, t + \Delta t) = f_i^*(\vec{x}, t)$ Qian et al., (1992).

2. Collision: At time t , the particles located at a specific node x interact, leading to changes in their velocity directions. As a result of this collision, the distribution function transitions from $f_i(\vec{x}, t)$ to $f_i^*(\vec{x}, t) = \frac{1}{\tau} (f_i^{eq} - f_i(\vec{x}, t))$, where τ is a characteristic time representing the relaxation rate of the distribution function towards its equilibrium state f_i^{eq} .

LBM models are categorized based on the spatial dimension d and the count of distribution functions b , leading to the descriptor $DdQb$. For two-dimensional scenarios, a D2Q9 lattice model, encompassing nine distinct unit velocities, is employed to represent the flow field, as illustrated in Figure 1.

LBM schemes are classified as a function of the spatial dimension d and the number of distribution functions b , resulting in the notation $DdQb$. In the two-dimensional case, a model with nine discrete unit velocities is realized for simulating the flow field (D2Q9) lattice model as shown in Figure 1.1.

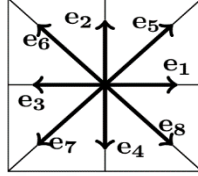


Figure 2.1. Discrete velocity directions for D2Q9 model in a computational cell (Deiterding,2011)

$$e_{\alpha} = \begin{cases} 0, \\ (\pm 1, 0)c, (0, \pm 1)c, \\ (\pm 1, \pm 1)c \end{cases} \alpha = 0$$

$$\alpha = 1, 2, 3, 4$$

$$\alpha = 5, 6, 7, \omega_i = \begin{cases} 4/9 \\ 1/9 \\ 1/36 \end{cases} i = 0$$

$$i = 1, 2, 3, 4$$

$$i = 5, 6, 7, 8$$

Usually the discretized equilibrium distribution function for D2Q9 adopts the following expression [5]:

$$f_i^{eq} = \rho \omega_i \left(1 + \frac{3}{c_s} (e_{ia} u_a) + \frac{9}{2c_s^2} (e_{ia} u_a)^2 - \frac{3}{2c_s^2} (u_a)^2 \right) \quad (2.5)$$

Where e_{ia} discrete velocity, $c_s = c/\sqrt{3}$ is the sound speed, u the macroscopic velocity, ω_i are built preserving the isotropy. The pressure was obtained through the equation of

state: $p = c_s^2 \rho$, here. It can be further demonstrated that there's a relationship between the kinematic viscosity ν and the collision frequency w , Chen et al., (1998).

$$w = \tau^{-1} = \frac{c_s^2 \Delta t}{\nu + \Delta t c_s^2 / 2} \quad (2.6)$$

Though the outlined model is readily applicable for simulating laminar flows, incorporating a turbulence model becomes indispensable in scenarios with high Reynolds numbers. Within the LBM framework, the large eddy simulation methodology is often the preferred choice.

In the three-dimensional realm of the LBM, the physical domain gets discretized into uniform Cartesian grids termed as lattices. Every lattice is structured with nodes, and these nodes communicate with adjacent ones via certain lattice velocities determined during the computation process. The classification of LBM frameworks depends on their spatial dimensionality (d) and their count of distribution functions (b), expressed as DdQb. For bi-dimensional configurations, standard models include D2Q7 and D2Q9. Meanwhile, in tri-dimensional contexts, models like D3Q13, D3Q15, D3Q19, and D3Q27 gain prominence. For instance, the XFlow computational tool harnesses the D3Q27 model, characterized by its twenty-seven distinct velocities, as illustrated in Figure 1.2.

By employing these lattice models, the LBM can accurately simulate fluid flow and capture complex flow phenomena in three-dimensional space. Each lattice node carries information about the fluid's local characteristics, such as velocity and density, allowing for the simulation of various fluid dynamics problems.

The discretized equilibrium distribution function for D3Q27 covers the following expression [3]:

$$f_i^{eq} = \rho \omega_i \left(1 + \frac{3}{c_s^2} (e_{ia} u_a) + \frac{9}{2c_s^4} (e_{ia} u_a)^2 - \frac{3}{2c_s^2} (u_a)^2 \right) \quad (2.7)$$

Where e_{ia} discrete velocity, c_s is the sound speed, u the macroscopic velocity, ω_i are built preserving the isotropy. For this purpose, the D3Q27 scheme has been implemented to the

analysis of VAWT and Wall-Adapting Local Eddy-viscosity (WALE), Spalart-Allmaras and Smagorinsky turbulence models have been selected to analysis static pressure, velocity and turbulence models. The WALE has good properties both near to and far from the wall turbulent flows (Weickert et al. 2010; Zhuo et al. 2013).

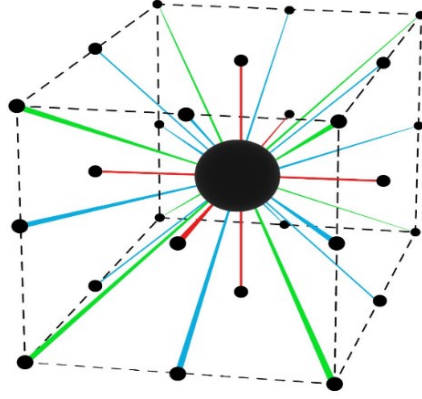


Figure 2.2. The lattice structure of D3Q27 model (Laloglu et al., 2023)

2.2. Large - Eddy Turbulence Models

Large Eddy Simulation (LES) is a computational method used to simulate turbulent flows by capturing the energy-containing large-scale structures while modelling the effects of the smaller-scale turbulent fluctuations. To accurately represent the subgrid-scale (SGS) turbulence, various turbulence models have been developed. In the context of LBM, it is common to adopt a large eddy simulation approach and assume that the partial density distribution functions used in the scheme represent the resolved scales. The sub-grid scale turbulence is then considered by adding a turbulent viscosity ν_t to the physical one Krafczyk et al (2003).

$$\nu^* = \nu + \nu_t = c_s^2 \left(\tau^* - \frac{\Delta t}{2} \right)$$

$$\text{where } \tau^* = \tau_L + \tau_t =: 1/\omega^*$$

and the effective relaxation frequency, ω^* , replaces ω in the collision operator throughout the scheme.

Although it is more computationally expensive to use the space-filtered NSE than the time-averaged Navier- Stokes Equation, the former produces accurate solutions that capture details of physical quantity fluctuations that are lost in time-averaging.

Large eddy simulation (LES) is the predominant form of the space-filtered NSE and is derived by introducing a space-filtered quantity, $\tilde{u}$.

$$\tilde{u}(x_i, t) = \iint \int \Delta x_i u(x_i, t) G(x_i, x_i') dx_i$$

where G is a spatial filter convolution kernel which has an associated cutoff length scale, l_s , and cut off time scale, τ_c . Scales smaller than these are eliminated from $\tilde{u}$. The resulting space-filtered Navier- Stokes Equation system is

$$\frac{\partial \tilde{u}_j}{\partial x_j} = 0$$

$$\frac{\partial \tilde{u}_i}{\partial t} + \frac{\partial \tilde{u}_i \tilde{u}_j}{\partial x_j} = F_i - \frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x_i} + \nu \frac{\partial^2 \tilde{u}_i}{\partial x_i \partial x_j} - \frac{\partial \tau_{ij}}{\partial x_j}$$

where τ_{ij} is the sub-grid-scale stress (SGS) that reflects the interaction of the unresolved scales with the resolved scales. It arises from the filtered advection term (Leonard, 1975)

$$\tilde{u}_i \tilde{u}_j = \tau_{ij} + \tilde{u}_i \tilde{u}_j$$

The Boussinesq assumption for turbulent stresses allows the sub-grid-scale stress to be represented with an SGS eddy viscosity, ν_t , as

$$\tau_{ij} = -\nu_t \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)$$

2.2.1. Smagorinsky Model

One of the most important tasks in an LES solution is to model the sub-grid scale stresses. In this study, Smagorinsky models (Smagorinsky, 1963) was used for this purpose.

Within LES simulations, an extra form of viscosity, termed the turbulent eddy viscosity v_t , is incorporated to account for turbulence effects. This specific viscosity is expressed as:

$$v_t = C_x^2 \Delta x^2 \bar{\omega} \quad (2.8)$$

Where C_x indicates the constant associated with the LES model, while, Δx represents the spacing in the lattice structure. The operator the lattice spacing, and $\bar{\omega}$ corresponds to the LES model. Within the framework of the lattice Boltzmann formulation, the connection between viscosity and the relaxation time τ is established by

$$v_{total} = v_0 + v_t = \frac{2\tau_0 - 1}{6} + \frac{\tau_t}{3} \quad (2.9)$$

where v_0 refers to the inherent molecular viscosity. Drawing parallels to the division of the viscosity parameter, the relaxation time τ is segmented into its molecular and turbulent components. The collective relaxation time can be expressed as:

$$\tau_{total} = 3v_{total} + 0.5 \quad (2.10)$$

By adding Eq. 2. 8)

$$\tau_{total} = 3(v_0 + C_x^2 \Delta x^2 \bar{\omega} + 0.5) \quad (2.11)$$

Derived the operator $\bar{\omega}$ exclusively from the shear stress tensor for Smagorinsky as follows:

$$\bar{\omega} = \sum_{i,j} S_{ij} S_{ij}$$

Smagorinsky, LES model constant was given the value of $C_x = 0.2$ in Equation 2.7.

2.2.2. Dynamic Models

Dynamic models are extensions of the Smagorinsky model that aim to improve the accuracy and adaptivity of the eddy viscosity coefficient. These models dynamically estimate the eddy viscosity coefficient based on the local flow characteristics. One example is the dynamic Smagorinsky model proposed by Germano et al. (1991), which incorporates a test filter to determine the optimal value of the eddy viscosity coefficient. The dynamic models account for the spatial and temporal variations in the flow and provide a more accurate representation of the turbulence.

2.2.3. Scale-Dependent Models

Scale-dependent models consider the scale dependency of turbulence and aim to capture the anisotropic behavior of turbulence at different length scales. These models utilize different eddy viscosity coefficients for different scales, allowing for a more accurate representation of the turbulent flow field. Examples of scale-dependent models include the dynamic mixed-scale model proposed by Lilly (1992) and the scale-similarity model developed by Vreman et al. (1997). These models consider the non-universal behaviour of turbulence and provide improved predictions for flows with complex geometries.

2.2.4. Wall-Adapting Models

Wall-adapting models are specifically designed to handle turbulent flows near solid walls, where the existence of boundary layers significantly affects the flow dynamics. These models incorporate wall functions or modifications to the eddy viscosity coefficient to account for the near-wall effects. Various wall-adapting models have been proposed to improve the accuracy of LES near walls. Examples include the wall-resolving model, which resolves the near-wall region with a fine grid, and the wall-modeled model, which uses wall functions to approximate the near-wall flow, Sagaut, (2006).

2.3. Applications of Lattice Boltzmann Method (LBM) in Fluid Dynamics

The LBM has gained significant attention in recent years due to its ability to simulate complex fluid flows with high accuracy and computational efficiency. LBM is a mesoscopic numerical approach that discretizes the fluid domain into a lattice structure, where fluid properties are represented by particle distribution functions. This method offers several advantages over traditional numerical methods, such as its inherent parallelism, easy handling of complex geometries, and efficient treatment of fluid-structure interactions.

LBM has been successfully applied to a wide range of fluid dynamics problems. One such area is in the study of multiphase flows, where the interactions between different fluid phases are of interest. The immiscible fluid interfaces and interfacial dynamics can be accurately captured using LBM, making it suitable for simulating phenomena such as droplet coalescence, bubble dynamics, and liquid-solid interactions (Liu et al., 2019). Additionally, LBM has been employed in modelling porous media flows, where the fluid flow through a porous structure is of interest. The ability of LBM to handle complex geometries and its inherent adaptivity make it suitable for simulating fluid flow through porous media (Liu et al., 2018).

Turbulent flows have also been extensively studied using LBM. The ability to model turbulence using large eddy simulation (LES) techniques with LBM has provided valuable insights into the complex behavior of turbulent flows. LES-LBM simulations have been employed in various applications, including the study of turbulent boundary layers, flow over complex geometries, and turbulent mixing processes (Chen et al., 2019). The combination of LBM and LES allows for a detailed analysis of the flow characteristics, such as vortex shedding, turbulence statistics, and energy spectra.

Furthermore, LBM has found applications in the field of microfluidics, where fluid flow at small scales is of interest. The ability of LBM to handle multiscale problems and its computational efficiency make it suitable for simulating fluid flows in microchannels and microdevices (Li et al., 2017). LBM has been used to investigate phenomena such as electrokinetic, droplet dynamics, and microscale heat transfer.

In summary, the LBM has proven to be a versatile and powerful tool in the field of fluid dynamics. Its applications span various areas, including multiphase flows, porous media flows, turbulent flows, and microfluidics. The accurate representation of complex fluid behaviour, computational efficiency, and flexibility in handling different flow regimes make LBM a valuable numerical approach in fluid dynamics research.

2.4. Advantages and Limitations of LBM

The Lattice Boltzmann Method (LBM) offers several advantages that make it an attractive choice for simulating fluid flows. One of the key advantages is its inherent parallelism, which allows for efficient utilization of modern high-performance computing architectures (Kruger et al., 2017). The lattice structure of LBM lends itself well to parallel processing, enabling the simulation of large-scale and complex fluid flows within reasonable computational timeframes. This makes LBM particularly useful for tackling problems that involve high Reynolds numbers or intricate geometries.

Another advantage of LBM is its ability to handle complex boundary conditions and interfaces with relative ease. The lattice-based nature of LBM allows for straightforward implementation of various boundary conditions, including no-slip, slip, and free-slip conditions, as well as the modeling of moving or deforming boundaries (Ginzburg et al., 2014). Additionally, LBM excels at capturing fluid interfaces and interfacial phenomena, such as phase separation and multiphase flow behavior, due to its mesoscopic nature (Shan et al., 2015). This makes LBM a valuable tool in studying problems involving fluid-structure interactions or complex fluid interfaces.

Moreover, LBM exhibits adaptivity in terms of spatial resolution, allowing for efficient and accurate simulations. Unlike traditional mesh-based methods, LBM does not require complex mesh generation procedures, making it particularly suitable for problems with irregular geometries or situations that involve frequent grid modifications (Kruger et al., 2017). This adaptivity enables the simulation of flow phenomena at different scales, ranging from macroscopic to microscale flows, using a unified numerical framework.

Despite its advantages, LBM also has some limitations that need to be considered. One limitation is the relatively higher memory requirement compared to other numerical methods. LBM requires storing particle distribution functions on the lattice, which can lead to significant memory usage, especially for large-scale simulations (Kruger et al., 2017). Additionally, the mesoscopic nature of LBM introduces statistical noise in the simulation results, which may require longer computational times or additional post-processing techniques for accurate analysis (Ginzburg et al., 2014).

In summary, the Lattice Boltzmann Method (LBM) offers several advantages, including its parallelism, ease of handling complex boundaries and interfaces, and adaptivity in spatial resolution. These characteristics make LBM well-suited for simulating complex fluid flows. However, it is important to consider its limitations, such as higher memory requirements and the presence of statistical noise, when applying LBM to fluid dynamics problems.

2.5. Introduction to Solution-Adaptive Methods

Solution-adaptive methods are a class of numerical techniques used in computational fluid dynamics (CFD) to dynamically refine or coarsen the computational grid based on the solution characteristics. These methods aim to enhance the accuracy and efficiency of CFD simulations by selectively refining regions with high solution gradients or important flow features while coarsening regions with low solution variations (Tumin & Erturk, 2016). By adaptively adjusting the grid resolution, solution-adaptive methods can provide more accurate solutions with reduced computational costs compared to fixed grid approaches.

The formulation of solution-adaptive methods involves several key steps. Firstly, an initial grid is generated to discretize the computational domain. This grid can be structured or unstructured, and its quality and resolution are chosen based on the problem requirements. The initial grid serves as the foundation for subsequent adaptivity procedures.

Next, the CFD solver is employed to solve the governing equations on the initial grid. The solution variables, such as velocity, pressure, and temperature, are computed iteratively using numerical algorithms. During the solution process, error indicators are employed to

assess the local accuracy of the solution and identify regions where grid refinement or coarsening is necessary (Bungartz & Zenger, 2002).

Various error indicators can be used to guide the adaptivity process. These indicators can be based on solution gradients, local truncation errors, or other flow characteristics of interest. Commonly used error estimation techniques include the Richardson extrapolation, adjoint-based error estimation, and residual-based error estimation (Iaccarino & Verzicco, 2003). These indicators provide information about the regions that require grid refinement or coarsening to improve the overall accuracy of the solution. Once the error indicators are computed, grid adaptation strategies are employed to refine or coarsen the grid accordingly. Grid refinement involves subdividing cells or adding new cells to increase the grid resolution, while grid coarsening involves merging cells or removing unnecessary cells to reduce the grid size. Various grid adaptation techniques, such as h-refinement, p-refinement, and adaptive mesh refinement (AMR), can be employed to adjust the grid structure based on the error indicators. The grid adaptation process is typically performed iteratively. After the grid is refined or coarsened, the CFD solver is applied again on the updated grid to obtain an improved solution. This iterative procedure continues until a desired level of accuracy is achieved or the solution reaches convergence. Overall, solution-adaptive methods provide an effective approach to enhance the accuracy and efficiency of CFD simulations. By dynamically adjusting the grid resolution based on the solution characteristics, these methods can capture flow features and resolve regions of interest with higher accuracy while minimizing computational costs in less significant areas.

2.6. Adaptive Mesh Refinement in LBM

For dynamic mesh adaptation at a local level, it has been integrated the block-structured AMR technique introduced by Berger & Collela. Originally, this method was conceptualized for time-explicit finite volume schemes pertaining to hyperbolic conservation laws. Nevertheless, given its iterative process and intrinsic design that considers time step refinement, it suits lattice Boltzmann schemes quite fittingly. This compatibility stems from the fact that the fundamental theory behind the transport step of

these schemes is rooted in a hyperbolic constant velocity advection equation. To ensure a seamless integration with the pre-existing parallelized finite volume AMR software, AMROC, it has been fashioned the LBM to be cell-centric. With the block-based AMR strategy, finite volume cells are systematically grouped into distinct, non-overlapping rectangular grids via a specialized algorithm. These grids are characterized by an appropriate layer of halo cells.

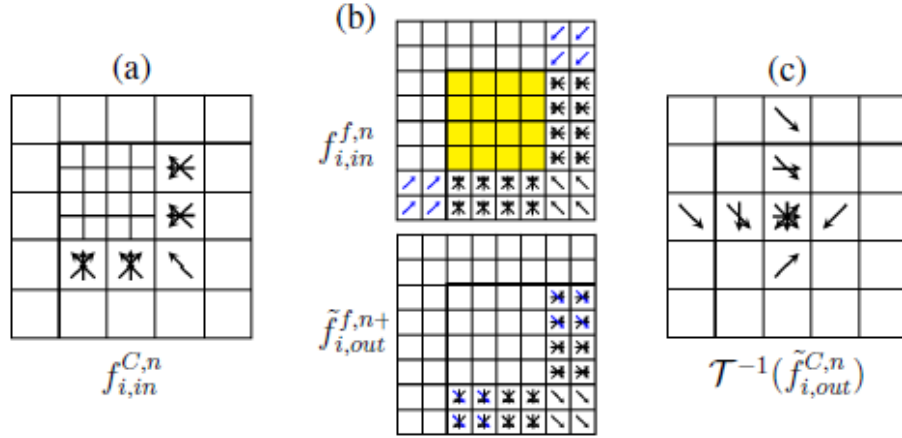


Figure 2.3. Displaying the partial density distribution functions is crucial when understanding the data transfer at the junction between coarse and fine boundaries. The bold black demarcations symbolize a physical boundary. (a) Coarse data points extending into the fine grid; (b) incoming interpolated fine points in peripheral zones (top) and outgoing points in these zones post two fine-level transport maneuvers (bottom); (c) computed averages that supplant coarse data before another update cycle adjacent to the boundary, Deiterding, (2002).

Grids are structured with particular emphasis on establishing conditions at inter-level junctions and physical boundaries. These refinement tiers are processed in a nested manner.

The spatial mesh width Δx and the time step Δt_l are refined by the same factor r_l , where we assume $r_l \geq 2$ for $l > 0$ and $r_0 = 0$. Note that in an adaptive LBM the collision frequency w_l is not a constant but needs to be adjusted.

Furthermore, the interface zone demands specific attention. In the given depiction and as illustrated in Figure 2.3, time intervals at the coarse layer C are marked by the superscript n , while the label F highlights the fine tier. The prefixes 'in' and 'out' demarcate distributions moving into and out of the fine grid at the junction with the coarse grid. Computationally, this algorithm aligns with the technique presented by Chen et al., 1998. It's precisely adapted to the Berger-Collela recursive pattern, which prioritizes updating the entire coarse grids before delving into fine grid calculations. This systematic full update of the associated coarse lattice followed by necessary adjustments is what makes the Berger-Collela method notably efficient. Yet, this strategy has been seldom employed in lattice Boltzmann techniques. Earlier adaptive LBM methodologies prioritized the fine grid's update before addressing the corresponding coarse tier, offering no clear path for introducing time-interpolated conditions at the fine-level interface.

3. BENCHMARKS FOR LBM AND SOLUTION-ADAPTIVE METHODS

3.1. Simulation of the Flow past an Cylinder at high Reynolds Number

The phenomenon of fluid flow around a circular cylinder has garnered considerable attention in the literature, undergoing extensive examination through both experimental and computational studies. Lausova et al. seek to identify an appropriate numerical model for simulating 2D flow past a smooth circular cylinder at a Reynolds number of 17,000.

Among the various approaches for modeling turbulent flows in engineering, RANS (Reynolds-Averaged Navier-Stokes) models, which rely on the time-averaged equations of motion, are predominantly utilized. Within this domain, the SST (Shear-Stress Transport) $k-\omega$ model is selected for this investigation due to its proven effectiveness in high Reynolds number scenarios. LBM analysis is compared with the study using XFlow and considered to implement close amounts of grids

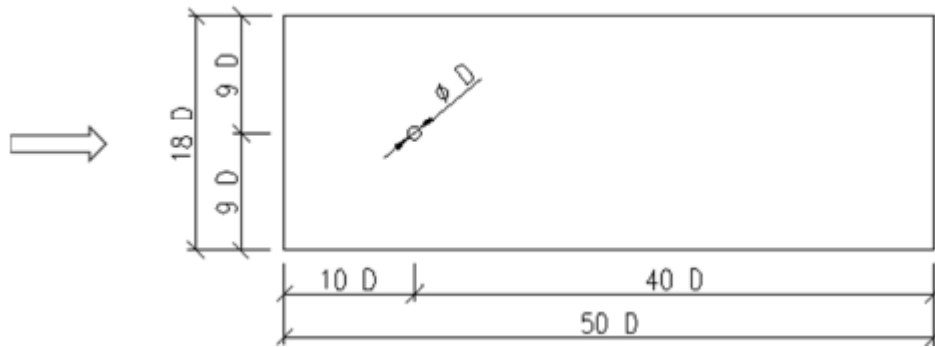


Figure 3.1. Computational area – dimensions Lausova et al., (2019).

Table 3.1. Parameters of the calculations Lausova et al., (2019).

Geometric dimensions	Values	Parameters of the calculations	Values
Cylinder diameter	$D = 0.1 \text{ m}$	Air velocity inlet	$u_0 = 3 \text{ m}\cdot\text{s}^{-1}$
Length of the area	$L = 5 \text{ m}$	Turbulence intensity	$i_u = 2 \%$
Width of the area	$B = 1.8 \text{ m}$	Kinematic viscosity	$\nu = 1.7 \cdot 10^{-5} \text{ m}^2 \text{ s}^{-1}$
Distance of cylinder axis from inlet	$l = 1.0 \text{ m}$	Density	$\rho = 1.225 \text{ kg}\cdot\text{m}^{-3}$

where u represents the flow velocity, ν is kinematical viscosity of the running fluid and D is a diameter of the cylinder. The value $Re = 1.7 \cdot 10^4$ represents a fully developed turbulence in the subcritical regime.

3.2. XFlow Simulation Setup

To conduct a comparable analysis to that performed the computational domain was discretized with a lattice arrangement. A total of 89,920 lattice points were strategically implemented across the computational area for this analysis. This meticulous lattice arrangement ensures the accurate representation of the flow dynamics surrounding the circular cylinder, facilitating a comprehensive comparison with the results.

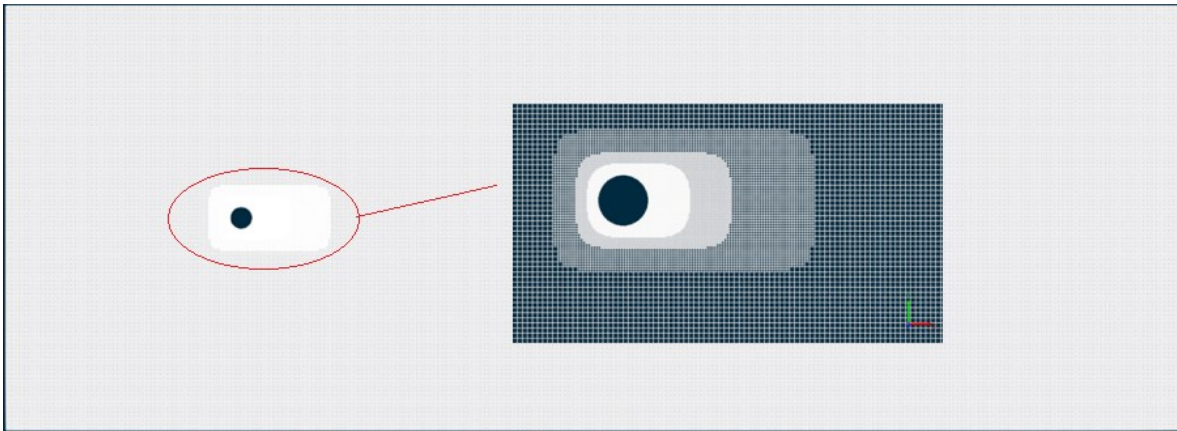


Figure 3.2. XFlow Lattice Structure with using Adaptive Refinement

3.3. Evaluation of Mean Stream Velocity Field in Wake Region Using Lattice Boltzmann Method

In this study, the focus lies on evaluating the mean stream velocity field within the wake region trailing the circular cylinder. To achieve this, the computational domain is discretized using the Lattice Boltzmann Method (LBM), a powerful numerical technique widely employed for simulating fluid flows.

The LBM approach is particularly suited for capturing complex flow phenomena due to its inherent ability to model fluid behavior at various scales. By employing LBM, the flow dynamics around the cylinder can be accurately simulated, providing insights into the characteristics of the wake region.

Furthermore, the Wale-Adaptive Large Eddy Simulation (WALE) turbulence model is employed specifically within the wake region. This model offers enhanced capabilities for capturing turbulent fluctuations, making it well-suited for accurately representing the turbulent flow characteristics prevalent in the wake of the cylinder. Illustrative figures are utilized to depict the mean stream velocity and velocity profiles across different grid configurations. These figures serve as visual aids for comparing and analyzing the computed results, providing valuable insights into the flow behavior and the effectiveness of the numerical approach employed.

Overall, the combination of the Lattice Boltzmann Method with the WALE LES turbulence model offers a robust framework for accurately evaluating the mean stream velocity field in the wake region behind the cylinder, facilitating a comprehensive understanding of the flow dynamics and contributing to advancements in fluid dynamics research.

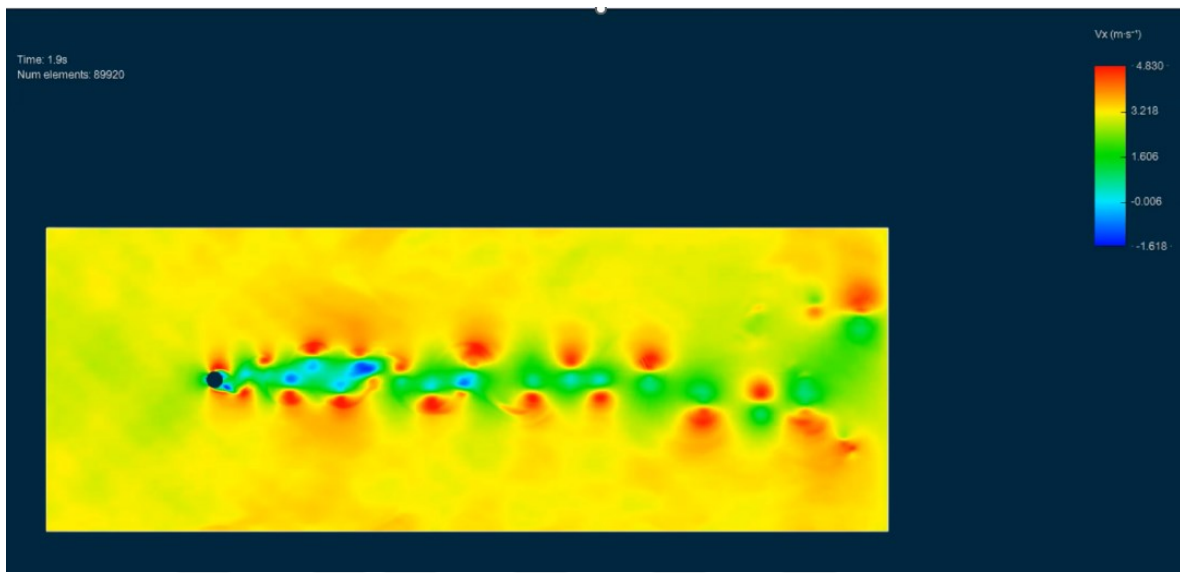


Figure 3.3. Horizontal velocity components

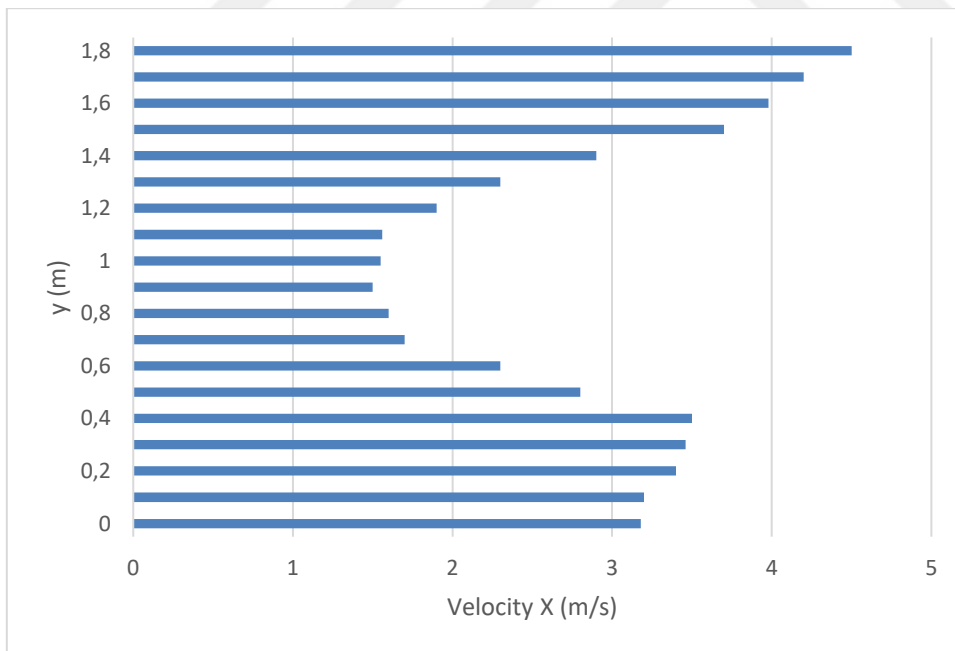


Figure 3.4. Horizontal Velocity Components – Vertical axis in the far wake

Figure 3.4 illustrates the mean stream velocity field in the far wake, providing valuable insights into the distribution of velocities downstream of the cylinder. This visualization

aids in understanding the evolution of flow structures and the extent of wake recovery, crucial for assessing the aerodynamic performance of the cylinder and its impact on downstream flow conditions.

3.4. Simulation of the Flow around an Oscillating Cylinder with Adaptive Lattice Boltzmann Methods

Many aerodynamic problems are characterised by an interaction between moving structures and fluid flow. As a canonical example for this problem class, we consider in here the subsonic flow around a cylindrical body undergoing an imposed regular oscillatory motion. This flow has been studied extensively experimentally as well as numerically because of its obvious importance as a fundamental problem occurring in a multitude of industrial applications. Studies have revealed that transverse or rotational motion of the cylinder can have a significant influence on the wake flow. Especially the influence of translational oscillations on vortex formation and the effect of rotation on wake behaviour have been evaluated Hou et al.; Howell et al. Flow-induced forces such as drag and lift are highly sensitive to the accurate representation of the motion as well as the mesh resolution, providing an ideal and challenging benchmark for modern fluid solvers that can handle large-scale fluid-structure interaction. In CFD analysis of the oscillating cylinder setup, a fine resolution is required to accurately represent the effect of the motion, which means that a high-resolution mesh must be used to analyse the flows around the cylinder, especially if the flow has a higher Reynolds number Laloglu et al., (2017).

To capture high frequency cylinder motion as well as small-scale flow structures, this study uses solution-adaptive mesh refinement. The underlying numerical discretisation adopted here is the LBM. LBM derives its foundation from the Boltzmann equation, which primarily dictates the behaviour of particle propagation. However, instead of focusing on singular particles, the LBM operates within the continuum domain by considering aggregates characterized by their probability densities Laloglu et al., (2017).

The lattice Boltzmann method can be understood as a mesoscale approach, designed for the intermediate scale regime between continuum (macro-scale) and individual particles

(micro-scale). The fundamental concept is to devise streamlined kinetic models at encapsulate the core physics of the microscopic interactions, ensuring that the aggregated values align with the targeted macroscopic equations. Simplified kinetic models, such as the BGK (Bhatnagar, Gross and Krook) collision operator, which is based on the assumption that only two particles can collide at a time, can be used if the macroscopic dynamics of a fluid is the result of the collective behaviour of many microscopic particles in the system. For instance, Jiange et al. (2016) have used a BGK-based incompressible lattice Boltzmann method to numerically investigate vortex-induced vibrations (VIV) phenomena around an oscillating square cylinder. In their study, results for stationary cylinder cases were in in good agreement with results of traditional CFD methods for Reynolds numbers $Re \leq 166$. When the Reynolds number was $Re > 166$, the trend of the Strouhal number variation over Reynolds number was still consistent with the results from conventional continuum mechanical methods, but differences increased with Reynolds number, which was attributed to differences in computational domain size and the inlet/outlet boundary conditions. Here, we reproduce by numerical simulation two laminar flow configurations that have been experimentally studied by Nazarinia et al.,(2011). They used particle image velocimetry (PIV) to characterise the wake of forced cylinder flow at Reynolds number 1322. We then extend these cases into the turbulent regime.

Table 3.2. Applied values of A_t and A_θ , Nazarinia et al.,(2011).

Case	A_t	A_θ	V_R
2	$D/4$	1.0	0.5
3	$D/2$	1.0	1.0

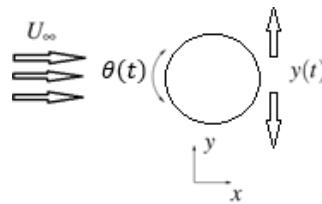


Figure 3.5. Motion imposed on cylinder, Nazarinia et al.,(2011).

The harmonic motions of the cylinder have translationally and rotationally movements. Equation (3.1) and (3.2) involve the rotational amplitude, frequency and time. The forced motion of the cylinder is specified by;

$$y(t) = A_t \sin(2\pi f_t t) \quad (3.1)$$

$$\theta(t) = A_\theta \sin(2\pi f_\theta t) \quad (3.2)$$

As can be seen Figure 3.5, the oscillating cylinder which is an outer diameter of 20 mm was moved on translational and rotational. The computational analysis in 2D was performed with a constant velocity of $U_{avg}=0.0606$ m/s for laminar flow and $U_{avg}=0.303$ m/s and $Re= 6610$ for turbulence flow. The frequencies of the motions are fixed close to that of the natural (von Karman) frequency ($T^{-1}=f_t=f_\theta=0.6$ s⁻¹). The natural frequency is $f_N \approx 0.6154$ s⁻¹. Thus, Strouhal number;

$$St_t = f_t D / U_\infty \approx 0.198 \text{ for frequency,}$$

$$St = f_N D / U_\infty \approx 0.203 \text{ for forcing (Nazarinia et al. 2011).}$$

3.5. Evaluating of mesh refinement and solution time

As comparing to experimental results, the selection of the lattice size can effectively change the CFD results owing to increasing of number of particles and relaxation time depending on probability distribution functions. When lattice size decreases in domain, the collision frequency will decrease and the results are more sensitive. Solution-adaptive approach provided the better accuracy and in good agreement with experimental data. The parameters of solvers information were tried to same level to correctly compare the results. There is no wall function that applied on boundary layer.

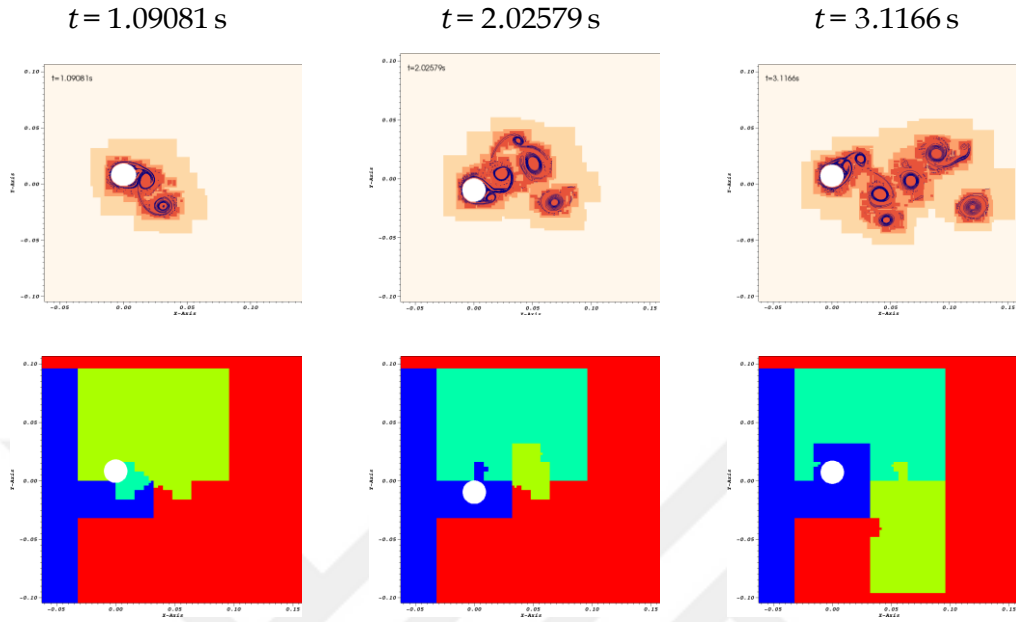


Figure 3.6. Case 3: Evolution of dynamic mesh refinement (upper row) and distribution to four processors over time (lower row) in AMROC. (Laloglu et al., 2017)

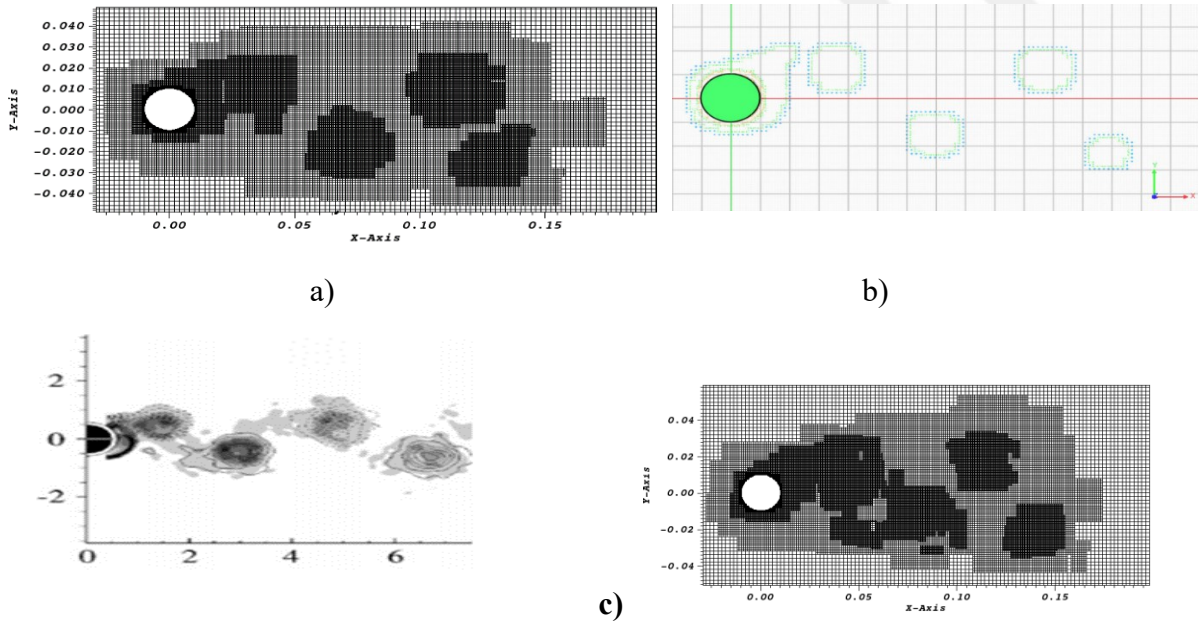


Figure 3.7. Mesh refinement of Case 2 on a) AMROC b) XFlow for laminar flows and c) experimental contour (Nazarinia et al., 2011)

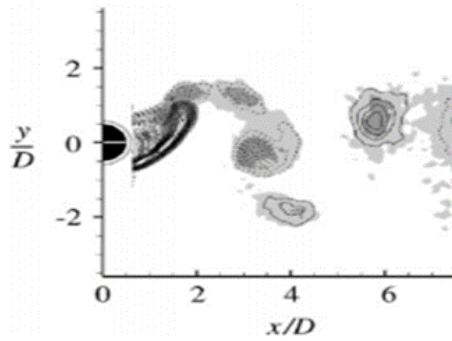
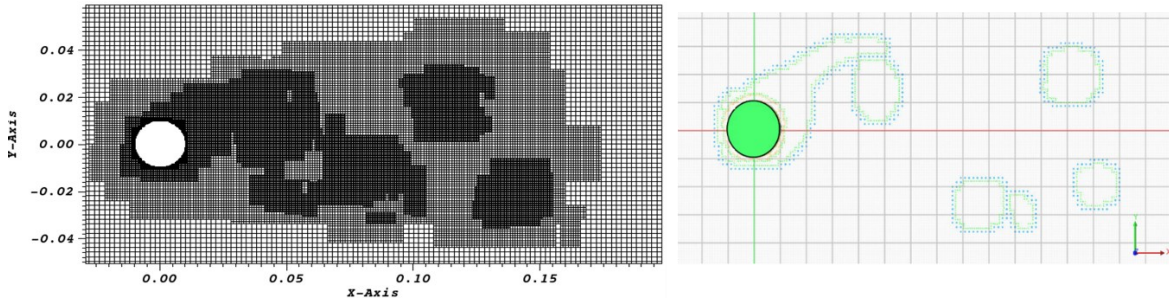


Figure 3.8. Mesh refinement of Case 2 on a) AMROC b) XFlow for laminar flows and c) experimental contour (Nazarinia et al., 2011)

An exemplary computation for Case 3 is visualised in Fig. 3.8. In the upper row, the domains of the three levels of successively finer block-structured meshes are shown with isosurfaces of the magnitude of the vorticity vector overlaid. It is apparent that the refinement follows the vortices generated by the cylinder motion. This computation was run on 4 processors and the lower row depicts in different colours the domains associated to the processors. The AMROC system uses a rigorous domain decomposition approach, in which higher level temporal and spatial refinement is considered in computing the actual workload, but only distribution entities in terms of the base level mesh are used for partitioning.

Table 3.3. Laminar and turbulence flows parameters for Case 2 and Case 3 (Nazarinia et al., 2011)

Flow type	Case	Δt_0 [s]	Total cells		Δt_e [s]	Re	y^+	CPU time [s]	
			AMROC	XFlow				AMROC	XFlow
Laminar	2	0.0015	85982	84778	3.33	1322	0	161.89	176
	3	0.0015	91774	90488	3.33	1322	0	165.97	183
Turbulent	2	0.00031	232840	216452	1.66	6310	2.4	635.8	887
	3	0.00031	255582	246366	1.66	6310	2.6	933.2	1325

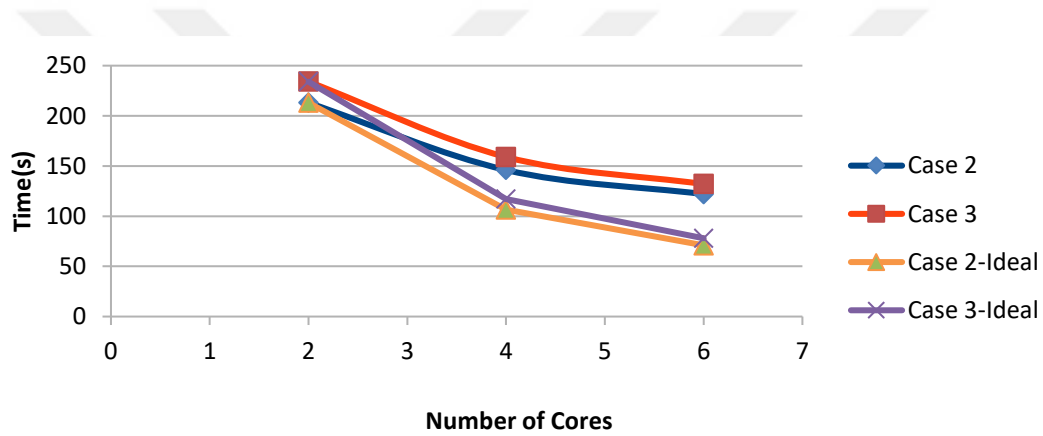


Figure 3.9. Overall solution time depending on number of cores in laminar flow for $t=3.33s$ on AMROC

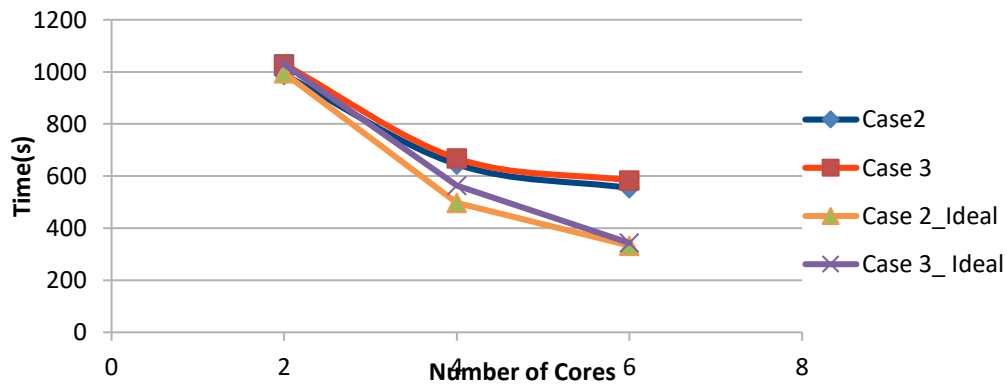


Figure 3.10. Overall solution time depending on number of cores in turbulence flow for $t=1.66 s$ on AMROC

The solution time can be decreased effectively with increasing of number of cores in (Message Passing Interface) MPI parallel solver. The code was operated on Ubuntu 14.04 with a workstation an Intel® Xeon® CPU Process @3.50 GHz, 16GB RAM, NVIDIA Quadro K4200 work station operating on 12 CPUs (6 cores, 6 threads). It is understood from Figure 3.9-3.10 that when the number of cores are increased, overall solution time is decreasing due to communications delaying among the processors.

3.6. Effecting of varying V_R in laminar flows

Obtaining computational analysis results are presented in vorticity distributions for AMROC code and XFlow commercial software data were compared with investigation of C_D (drag coefficient) and C_L (lift coefficient) in different velocity ratios. Utilizing by adaptive mesh refinement technique on AMROC, near-wake flow structures on downstream flow were obtained using with LBM method at laminar flows. The flow domain was arranged at same size for AMROC and XFlow. Note that the significant impact to solve the motion is required to observe the vortex shedding behind the cylinder. For this purpose, the vortex mode is classified by the Williamson and Roshko in 1988. From their analyses, they experimentally discerned the structures of the near-wake by examining vortex shedding, which revealed distinct vortex modes. The experimental review helped in recognizing specific vortex configurations close to the primary phase-lock-in zone, namely 2S, 2P, and P + S. The 2S pattern indicates that every half-cycle feeds a vortex into the downstream wake, reminiscent of the natural Karman vortex shedding. On the other hand, 2P depicts the creation of vortex duos that laterally drift away from the wake's central axis. The P + S mode stands as an unbalanced variant of the 2P configuration, where the cylinder releases both a vortex duo and a solitary vortex within a single cycle.

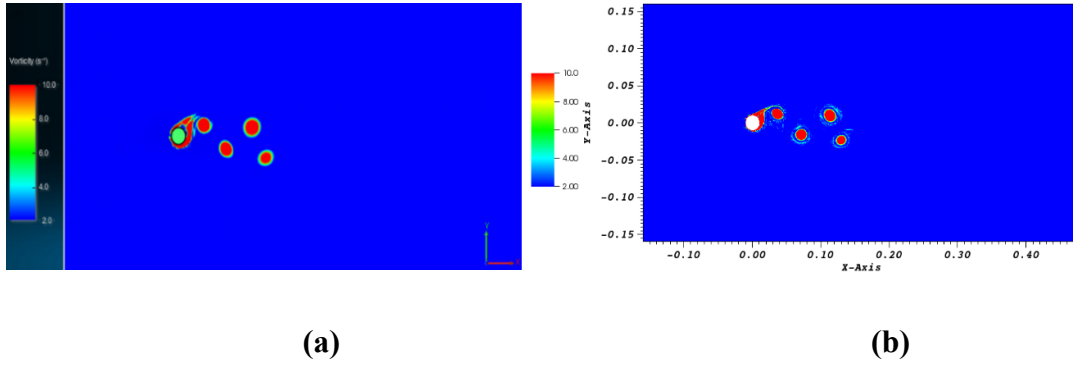


Figure 3.11. Vorticity distributions for laminar flows at $U=0.0606$ m/s and $Re= 1322$ a) XFlow and b) AMROC for Case 2, $V_R= 0.5$

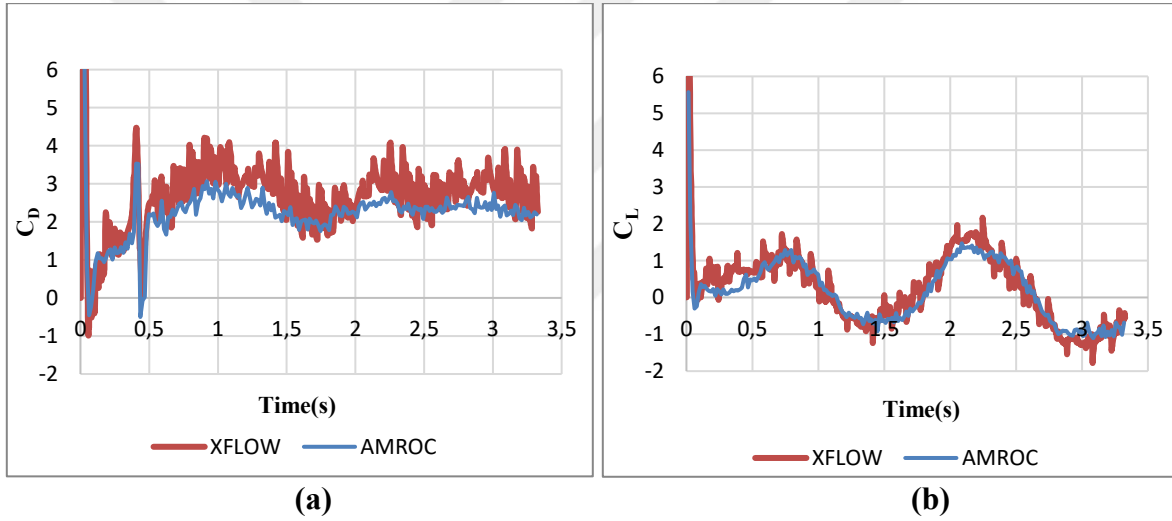


Figure. 3.12. Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 2, $V_R= 0.5$

Figure 3.11 gives us the 2S vortex mode and are coherent when compared to the referenced paper and XFlow commercial software for $V_R=0.5$. Furthermore, the near-wake vortices are remaining coherently in the near-wake and are synchronized with the translational motion. Besides, the graphs of drag and lift coefficient were demonstrated to get detailed comparison AMROC code with XFlow as shown Figure 3.12a and Figure 3.12b at V_R of 0.5. It showed the C_D and C_L values were presented an agreement even if they have some discrepancies. As can be seen from Figure 3.8, the same discrepancies were obtained at V_R of 0.5 during the analysis of Case 2.

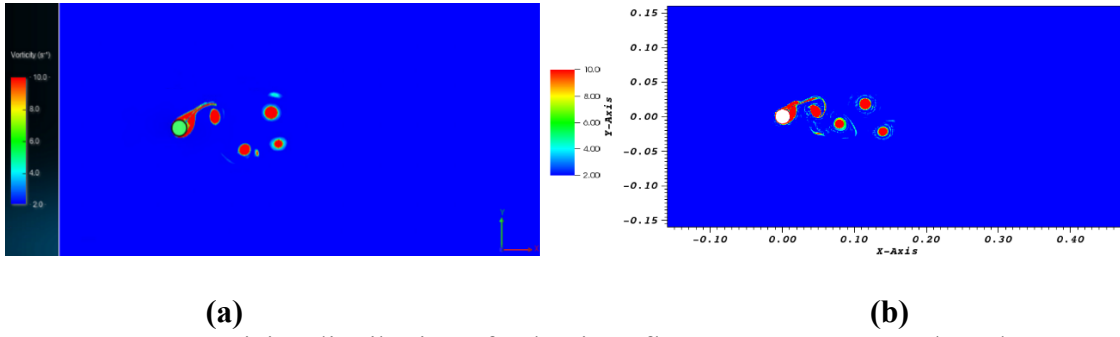


Figure 3.13. Vorticity distributions for laminar flows at $U=0.0606$ m/s and $Re= 1322$ a) XFlow and b) AMROC for Case 3, $V_R= 1$

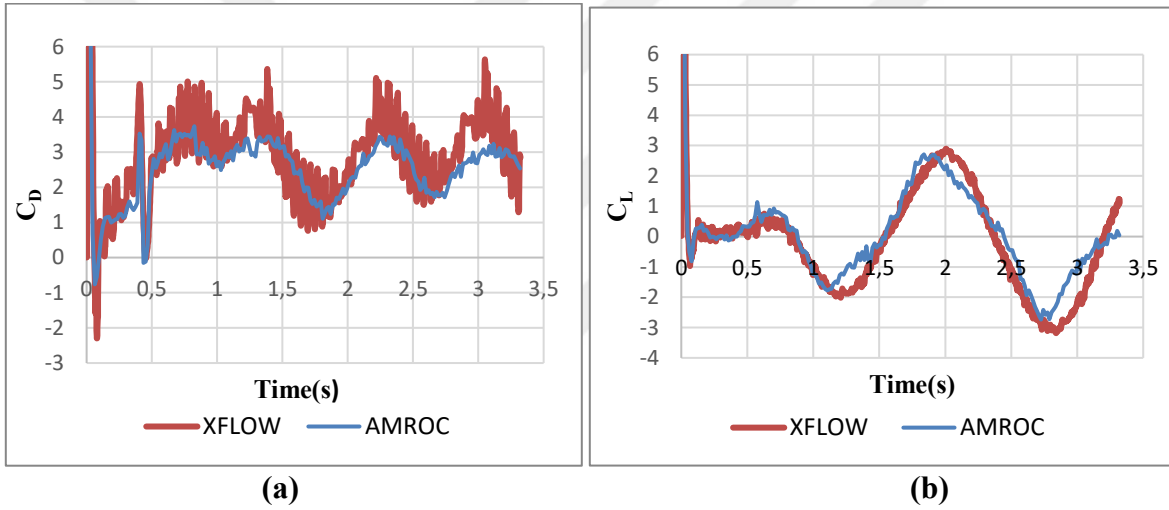


Figure 3.14. Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 3, $V_R= 1$

The parameters taking from referenced paper (Nazarinia et al., 2011) values were applied to for Case 2 and Case 3 as given Table 1. Obtained results were enabled to understand better the behaviour of the motion and mechanism behind of the oscillating cylinder movement with increasing of velocity ratios at phase-locked. The near-wakes structures are very agreement with XFlow commercial software as shown Figure 3.13. As the V_R increases to 1.0, observable from Figure 3.13, there's a notable transition in the vortex shedding mode. Instead of the previously observed 2S mode, a P+S mode emerges in the near-wake, characterized by the formation of one solitary vortex and a vortex pair during each cycle. The vortex shedding pattern identified for this scenario is distinctly P+S. Such vortex

formations are commonly recognized within the vortex street and have been studied for their synchronization with oscillatory movements. These vortex patterns contribute to a broader wake, predominantly because the translational motion in these situations outpaces the rotational. The C_D and C_L values were demonstrated still eligible discrepancies but they were occurred coherently as shown Figure 3.14.

3.7. Effecting of varying V_R in turbulence flows

In order to attain the more knowledge about the near-wake structures behind the cylinder, it was increased the average velocity further to $U=0.303 \text{ m/s}$ and $Re= 6310$. Smagorinsky sub-grid scale turbulence model was used to see the vorticity shedding at high Reynolds number. All cases as given Table 3.3 were applied at the same domain in 2D analysis.

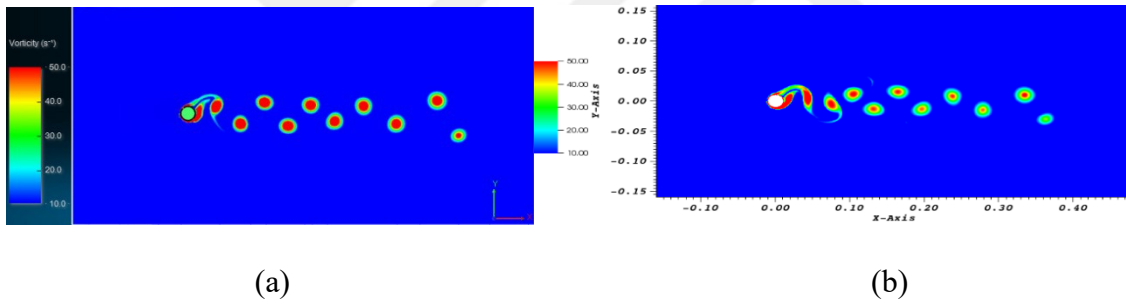


Figure 3.15. Vorticity distributions for laminar flows at $U=0.303 \text{ m/s}$ and $Re= 6310$ a) XFlow and b) AMROC for Case 2, $V_R= 0.5$

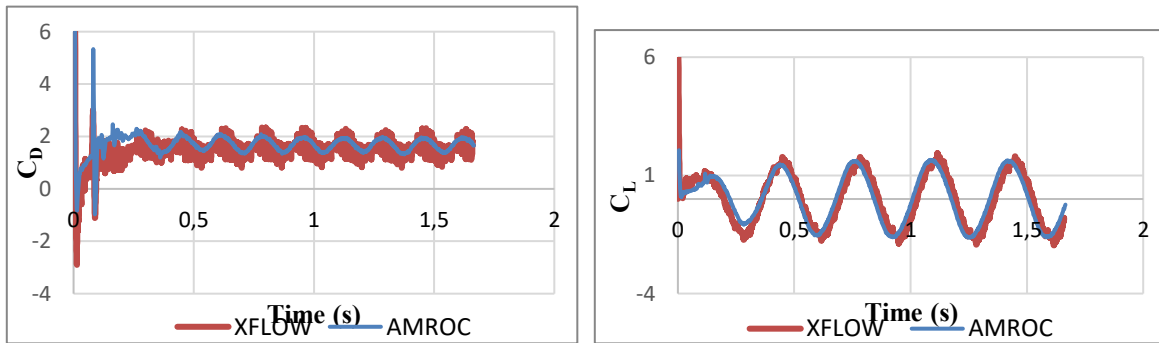


Figure 3.16. Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 2, $V_R= 0.5$

Figure 3.15 shows the 2S vortex mode obtaining from AMROC and are coherent when compared to the XFlow commercial software for $V_R=0.5$. In particular, C_D and C_L values of AMROC were presented very compatibility with XFlow results with slightly discrepancies as shown Figure 3.16.

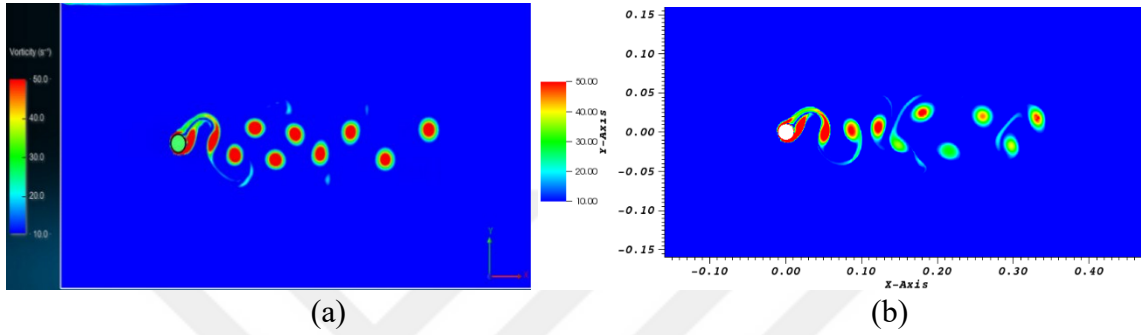


Figure 3.17. Vorticity distributions for laminar flows at $U=0.303$ m/s and $Re= 6310$ a) XFlow and b) AMROC for Case 3, $V_R= 1$

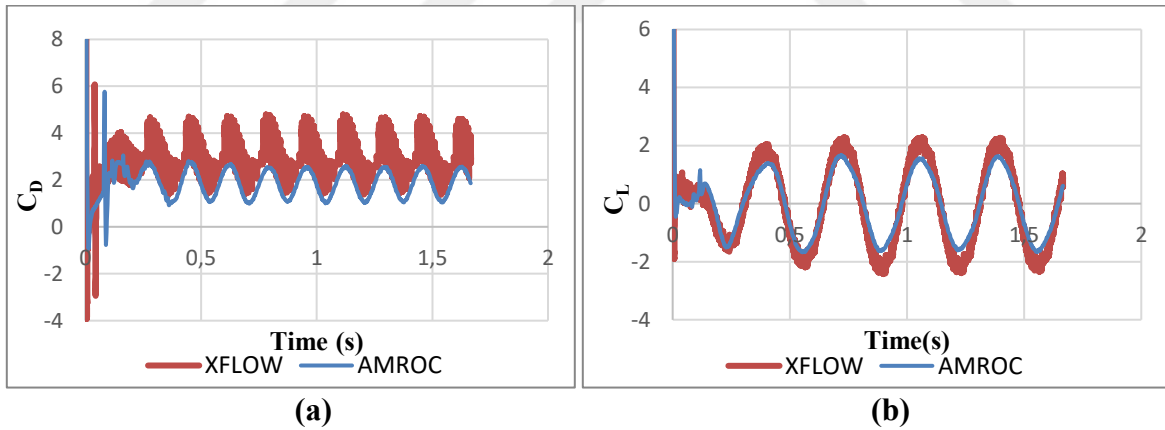


Figure 3.18. Comparison of a) drag coefficient (C_D) and b) lift coefficient (C_L) graph depending on time for Case 3, $V_R= 1$

Figure 3.17 gives vortex shedding mode to the signature of a P+S mode in the near-wakes structures is very agreement with XFlow commercial software. With increase of Re , it produced more defined a weak wake that proves the P vortex mode to outwards.

3.8. Darrieus VAWT Numerical Model Meshfree analysis using with LBM

Vertical axis wind turbines (VAWTs) are commonly categorized into two main types: drag-driven turbines, such as the Savonius design, and lift-driven turbines, like the Darrieus design. While drag-driven turbines tend to have better initial start-up performance, they exhibit lower power generation efficiency compared to lift-driven turbines. In this investigation, the focus is on evaluating the performance of a Darrieus-type wind turbine operating at a wind speed of 12 m/s. The straight blades of the turbine utilize the NACA 0015 airfoil profile. The specific characteristics of the VAWT employed in the virtual wind tunnel simulation conducted by (Lee Y. and Lim H. 2015) are as follows: the chord length is 150 mm, the rotor diameter (D) is 740 mm, the length (L) is 600 mm, and the aspect ratio (L/D) is 0.81. The turbine consists of three blades. Figure 3.19 illustrates a schematic CAD representation of the Darrieus VAWT employed in this study, with dimensions matching those of the experimental study conducted by (Lee Y. and Lim H. 2015). The Reynolds number for the given values of air velocity at 12 m/s is approximately 120,000.

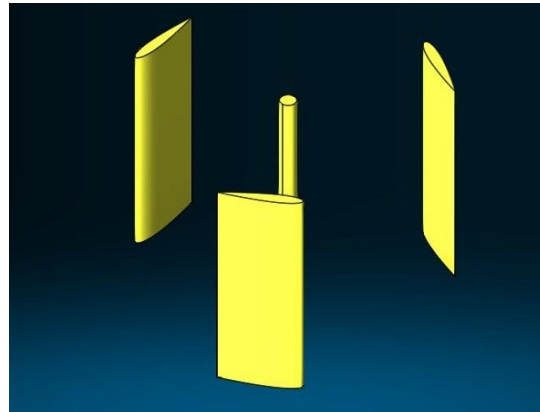


Figure 3.19. CAD geometry of Darrieus Vertical Axis Wind Turbine

3.8.1. Applied Turbulence Models

Selecting a turbulence model is an important factor to ensure that accurate numerical solutions can be obtained in a reasonable time. Two turbulence modelling approaches were used during the computations; LES and RANS.

3.8.1.1. Large Eddy Simulation (LES)

In Large Eddy Simulation (LES), an essential aspect is the representation of sub-grid scale stresses. This study employed two different models, namely WALE and Smagorinsky models, to address this task. During LES simulations, turbulent eddy viscosity ν_t is introduced as an additional viscosity to capture the characteristics of turbulence. The value of this eddy viscosity is determined by the chosen model equation 2.8 ;

Here, C_x represents a constant specific to the chosen LES model, Δx refers to the spacing between lattice points, and $\bar{\bar{\omega}}$ corresponds to the LES model operator. In the LBM model, the viscosity is connected to the relaxation time τ . This relationship governs the rate at which the distribution function approaches equilibrium. By appropriately adjusting the relaxation time, the desired viscosity can be achieved, influencing the flow behaviour in the simulation equation 2.9.

The relaxation time τ can be decomposed into molecular and turbulent components, analogous to the separation of the viscosity term. This allows for distinguishing the contributions of molecular viscosity ν_o and turbulent effects. The total relaxation time encompasses both these aspects, as expressed by the following equation 2.10.

By applying Eq. (2.8), we arrive at equation 2.11

Derived the operator $\bar{\bar{\omega}}$ exclusively from the shear stress tensor for Smagorinsky as follows equation 2.12

The WALE model considers the shear stress tensor including the rotation tensor. The operator $\bar{\bar{\omega}}$ is defined as follows ,Weickert et al., 2010 :

$$\bar{\bar{\omega}} = \frac{(G_{ij}^d G_{ij}^d)^{\frac{3}{2}}}{(S_{ij} S_{ij})^{\frac{5}{2}} + (G_{ij}^d G_{ij}^d)^{\frac{5}{4}}} \quad (3.3)$$

The difference between WALE and Smagorinsky, LES model constant was given the value of C_x , 0.325 and 0.12, respectively, in Equation 6.

3.8.1.2. Reynolds-Averaged Navier-Stokes (RANS)

In the context of RANS simulations, the Spalart-Allmaras turbulence model is a one-equation model solving a transport equation for a variable called $\tilde{v}$, which represents an approximation of viscosity. This variable is then used to adjust a turbulent viscosity ν_t . Within the framework of the lattice Boltzmann method (LBM), the turbulent viscosity can be employed to modify the relaxation time in the collision operator. In this investigation, a modified version of the model, presented in Equation 3.4, was implemented to enhance the numerical behaviour in the vicinity of the wall within the laminar sublayer. The transport equation and the complete set of parameters associated with the model are provided below (Laloglu et al., 2023):

$$\frac{\partial \bar{v}}{\partial t} + \vec{u} \cdot (\vec{\nabla} \bar{v}) = \frac{1}{\sigma} \left[\vec{\nabla} \cdot ((v + \tilde{v}) \vec{\nabla} \bar{v}) + c_{b2} (\vec{\nabla} \bar{v})^2 \right] + c_{b1} \tilde{S} \bar{v} - c_{w1} f_w \left(\frac{\bar{v}}{d} \right)^2 \quad (3.4)$$

$$\nu_t = \tilde{v} f_{v1}, \quad f_{v1} = \frac{X^3}{X^3 + c_{v1}^3}, \quad X = \frac{\tilde{v}}{v},$$

$$c_{b1} = 0.1355, \quad \sigma = \frac{2}{3}, \quad c_{b2} = 0.622, \quad \kappa = 0.41,$$

$$c_{w1} = \frac{c_{b1}^2}{\kappa} + \frac{1 + c_{b2}}{\sigma}, \quad c_{w2} = 0.3, \quad c_{w3} = 2, \quad c_{v1} = 0.1$$

3.8.1.3. Virtual Wind Tunnel Analysis and Boundary Conditions

The computational analysis consists of lattice structures shown Figure 3.20. The dimensions of the numerical wind tunnel were selected as $3\text{m} \times 2\text{m} \times 6\text{m}$ (width $\times$ height $\times$ length) to observe better analysis because it gives suitable environment depending on turbine size, investigating wake effect inside in virtual wind tunnel. In order to see the effect of number of cells on the solutions, two cases were chosen; lattice structures with element numbers about 2 500 000 and 5 000 000. In this simulation, inlet wind velocity of

12m/s is applied to predict the aerodynamic characteristics with various turbulence models in accordance with the reference (Lee et al. 2015). Adaptive wake refinement technique was used to observe detailed wake developing around the blades. The XFlow engine incorporates an automatic scale adaptation feature that adjusts the resolved scales based on user-defined criteria. This adaptive process enhances the solution accuracy in regions near the walls by refining the mesh, effectively capturing strong gradients within the flow. Additionally, as the flow progresses, the engine dynamically adapts to refine the wake region, ensuring an accurate representation of the evolving flow patterns. This behavior is illustrated in Figure 3.29, showcasing the adaptability and effectiveness of the XFlow engine in optimizing the solution quality throughout the computational domain.

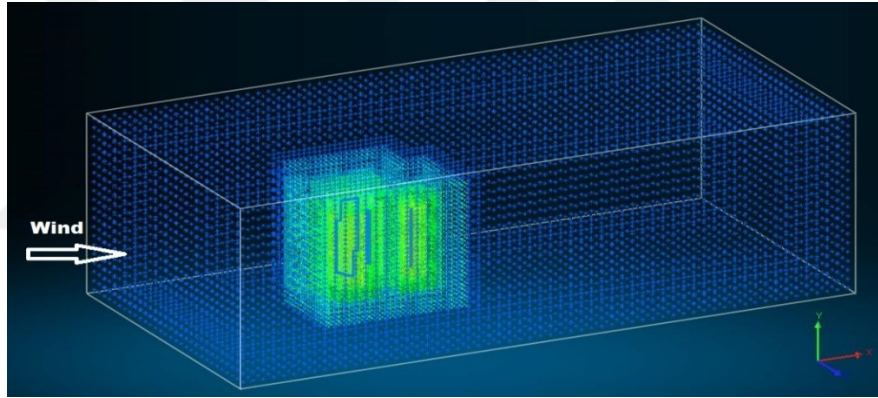


Figure 3.20. Cell Configuration of Darrieus Vertical Axis Wind Turbine (5 000 000 lattice cells).

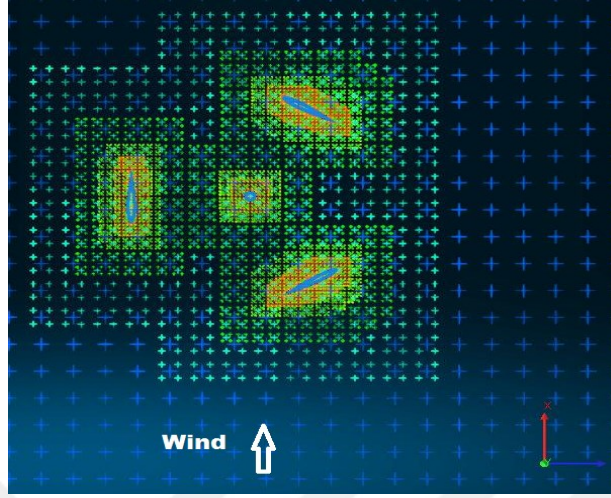


Figure 3.21. Adaptive Wake Refinement for Darrieus Vertical Axis Wind Turbine (5 000 000 lattice cells). The spatial resolution in this case varied within a range of 0.00125m to 0.05m.

The design parameters of turbine and boundary conditions were specified according to the reference (Lee et al. 2015). One of the performance parameters of wind turbines is the tip-speed-ratio (TSR), which is the blade-tip speed against wind speed and is defined in Equation 10. Typically, the power performance of the rotor can be presented by variation of power coefficient with TSR. In a wind turbine power obtains by multiplying the torque generated on the blades by the aerodynamic forces with the rotation speed. Torque coefficient (C_Q) and power coefficient (C_P) were calculated using Equations 3.11 and 3.12, respectively (Lee et al. 2015).

$$TSR = \frac{Tip\ Speed}{Wind\ Speed} = \frac{\omega \times R}{U_{\infty}} \quad (3.11)$$

$$C_Q = \frac{Q}{0.5 \rho A R U_{\infty}^2}$$

In Equation 11, Q is torque [Nm], ρ denotes the air density [kg/m^3], and A is the cross-sectional area of the rotor [m^2] (Lee et al. 2015).

$$C_p = \frac{Q\omega}{0.5\rho AU_\infty^3} \quad (3.12)$$

In order to investigate the performance characteristics, three tip-speed ratios (TSRs) of 1, 1.2 and 1.6 were chosen for the calculations. (Lee et al. 2015) obtained the maximum power coefficient at TSR of 1.2 in their experimental results.

3.8.1.4. Results and Discussion

Solution times with different turbulence modelling approaches

In this study, CFD solutions were performed on an Intel® Xeon® CPU Process @3.50 GHz, 16GB RAM, NVIDIA Quadro K4200 workstation operating on 12 CPUs. Mesh structures and their qualities may increase computational cost, and it is required that higher performance computer is established. However, LBM based on particle-based approach uses a uniform Cartesian grid that can be refined as it is required depending on geometry and flow field. In order to see the effect of the number of lattices on the solution time, CPU times of solutions with different turbulence modelling approaches were given in Table 3.3.

Table 3.4. Overall CPU time (s) with respect to number of lattices in different turbulence models

Number of Lattices	Overall CPU Time (s)		
	RANS	LES	LES
	Spalart Allmaras	WALE	Smagorinsky
2,500,000	1.13×10^4	1.16×10^4	1.18×10^4
5,000,000	1.64×10^4	1.77×10^4	1.87×10^5

The Courant Number plays a crucial role in governing the choice of time step scheme. By increasing the time step size, computational speed can be accelerated, making it desirable to set the Courant Number as high as feasible. On other hand smaller numbers mean a more stable solution but it will be slower since it is doing more steps. The relation between courant number and stable solution can be taken in consideration of stability parameter. The

stability parameter must be less than 1, so configuration of the simulation time was adjusted the given value of Courant number of 0.8.

When solution time displayed in Table 1 is compared, RANS solution with Spalart-Allmaras model had lower computational time compared to the others for both 2 500 000 and 5 000 000 lattices. Large- Eddy Simulation (LES) method, WALE and Smagorinsky turbulence models have greater values owing to the influence of time-dependent solution as shown Table 1. In other words, The RANS approach relies on ensemble-averaged equations, which limits its ability to predict local flow fluctuations. In contrast, the LES approach utilizes spatially filtered equations, allowing for the capture of large-scale flow structures based on the chosen filter size.

Torque and power coefficients

Torque coefficients were obtained using the XFlow software by time averaging the instantaneous torque predictions. Results obtained with 2500000 lattices are displayed and compared with experimental data in Figure 3.22.

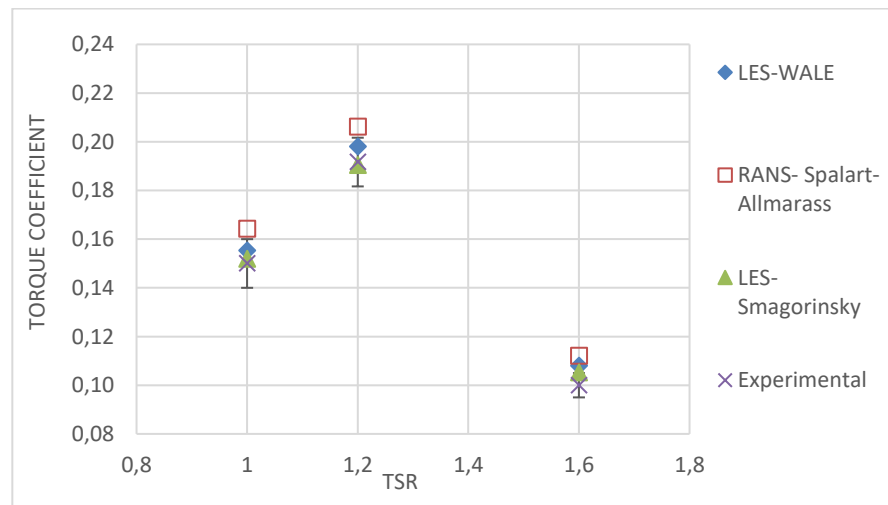


Figure 3.22. Variations in torque coefficients according to turbulence models and experimental results (Lee et al. 2015) for 2 500 000 lattices.

From Figure 3.22, it seems that results do not all agree well with the referenced experimental results. Here solutions with Spalart-Allmaras model showed the worst agreement while the other predictions remained within the error bars of the measurements for TSRs of 1 and 1.2. Among the solutions, LES predictions with Smagorinsky model yielded the best agreement.

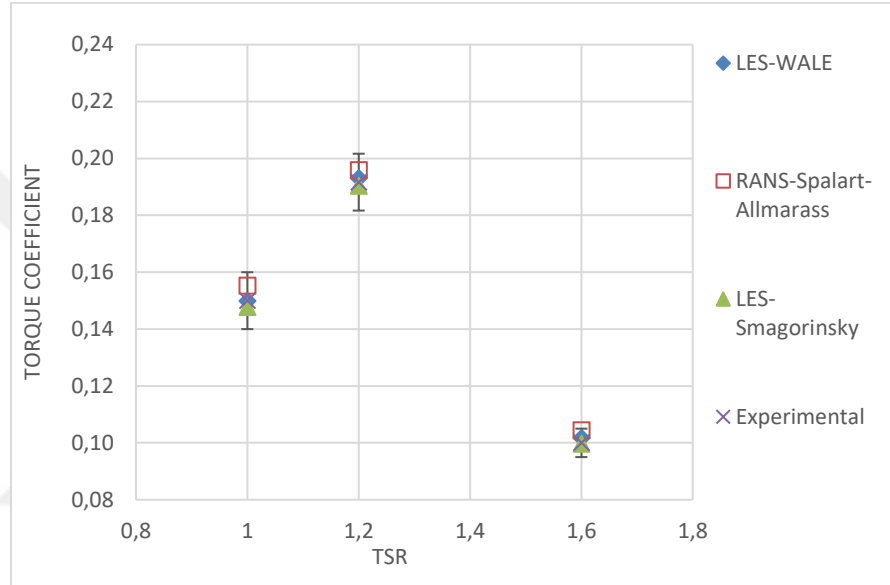


Figure 3.23. Variations in torque coefficients according to turbulence models and experimental results (Lee et al. 2015) for 5,000,000 lattices.

To see the effect of number of lattices on predictions, numerical solutions were repeated for a lattice size of 5 000 000 and the corresponding torque coefficient predictions are displayed in Figure 3.23 along with the experimental data. The improvement in the predictions is evident from this figure where all of them appeared to be inside the error bars of the measurement. Solutions obtained with WALE method showed the best agreement with experiment for this configuration.

Overall, the agreement between the numerical predictions and experimental measurements was found to be satisfactory for 5 000 000 lattices. Therefore, this lattice size was used in the rest of the study.

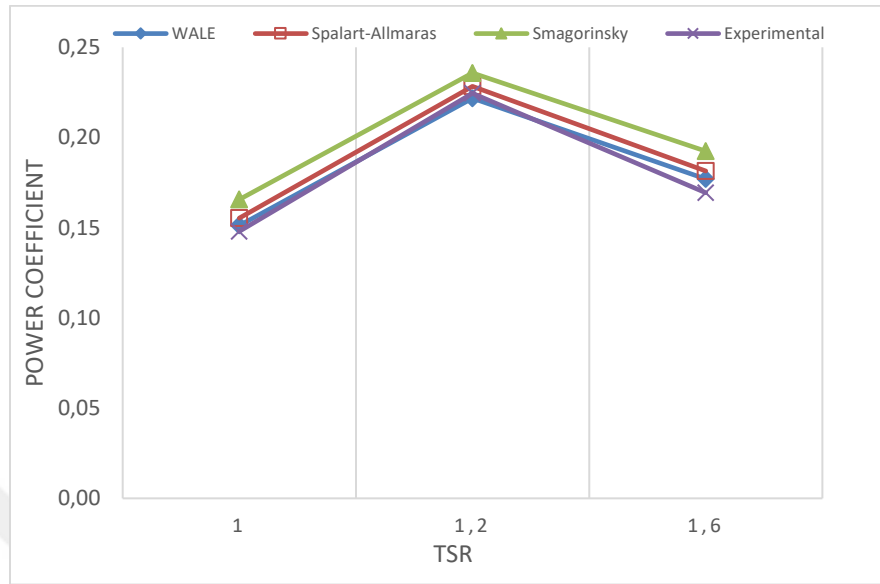


Figure 3.24. Variations in power coefficients according to turbulence models and experimental results (Lee et al. 2015) for 5,000,000 lattices

Figure 3.24 clearly demonstrates the good agreement between the WALE turbulence model and experimental results. Additionally, it can be observed that the power coefficient reaches to maximum at TSR 1.2 for 5 000 000 lattices.

Comparison of flow field predictions with different turbulence models

It is known that structure of a VAWT causes more wake-blade interactions than a HAWT. Based on this idea, this study aimed to reveal the effect of different turbulence models on the wake predictions for a VAWT.

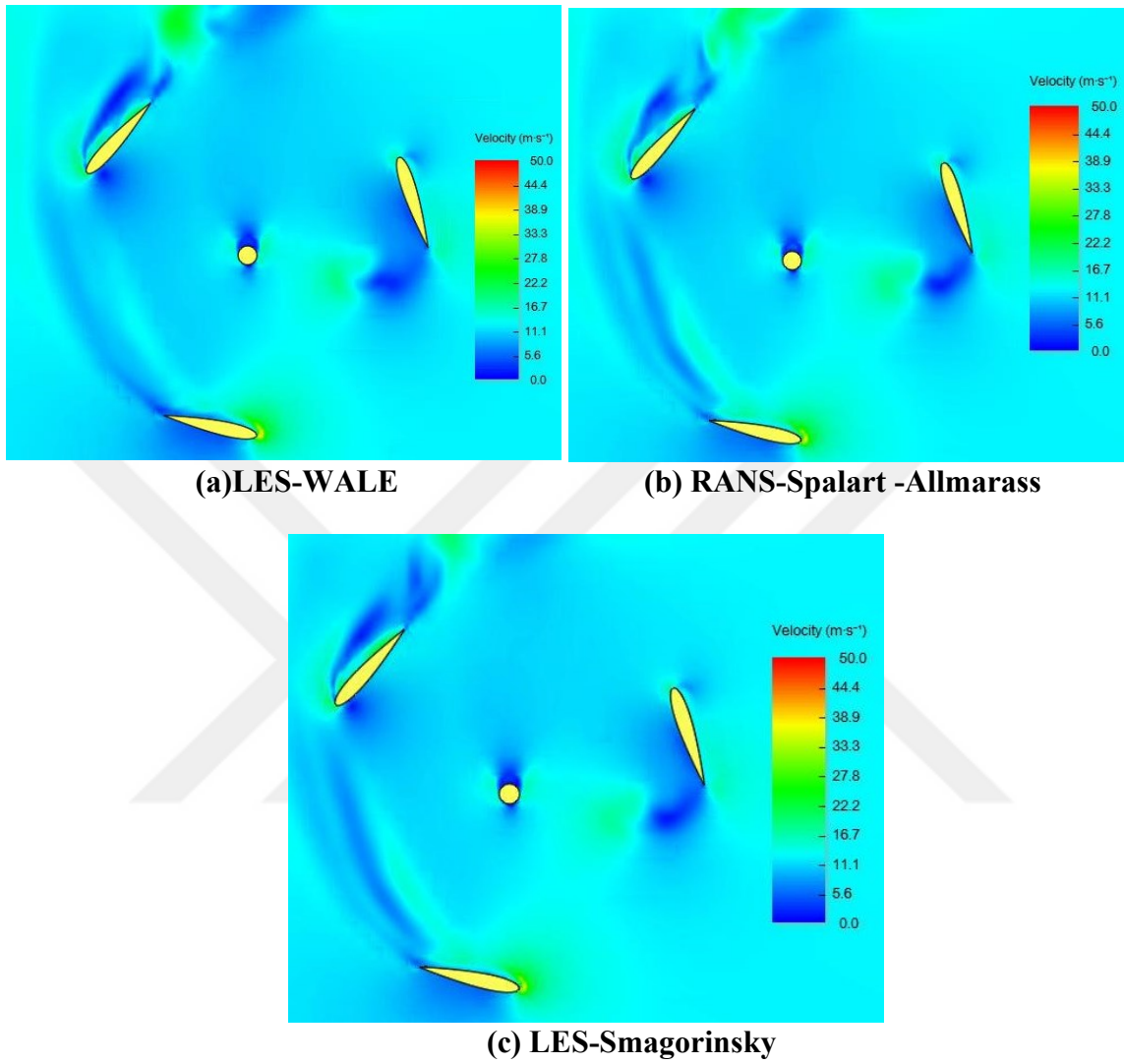


Figure 3.25. Velocity contours around VAWT in different turbulence models for TSR 1 and $\theta = 77^\circ$

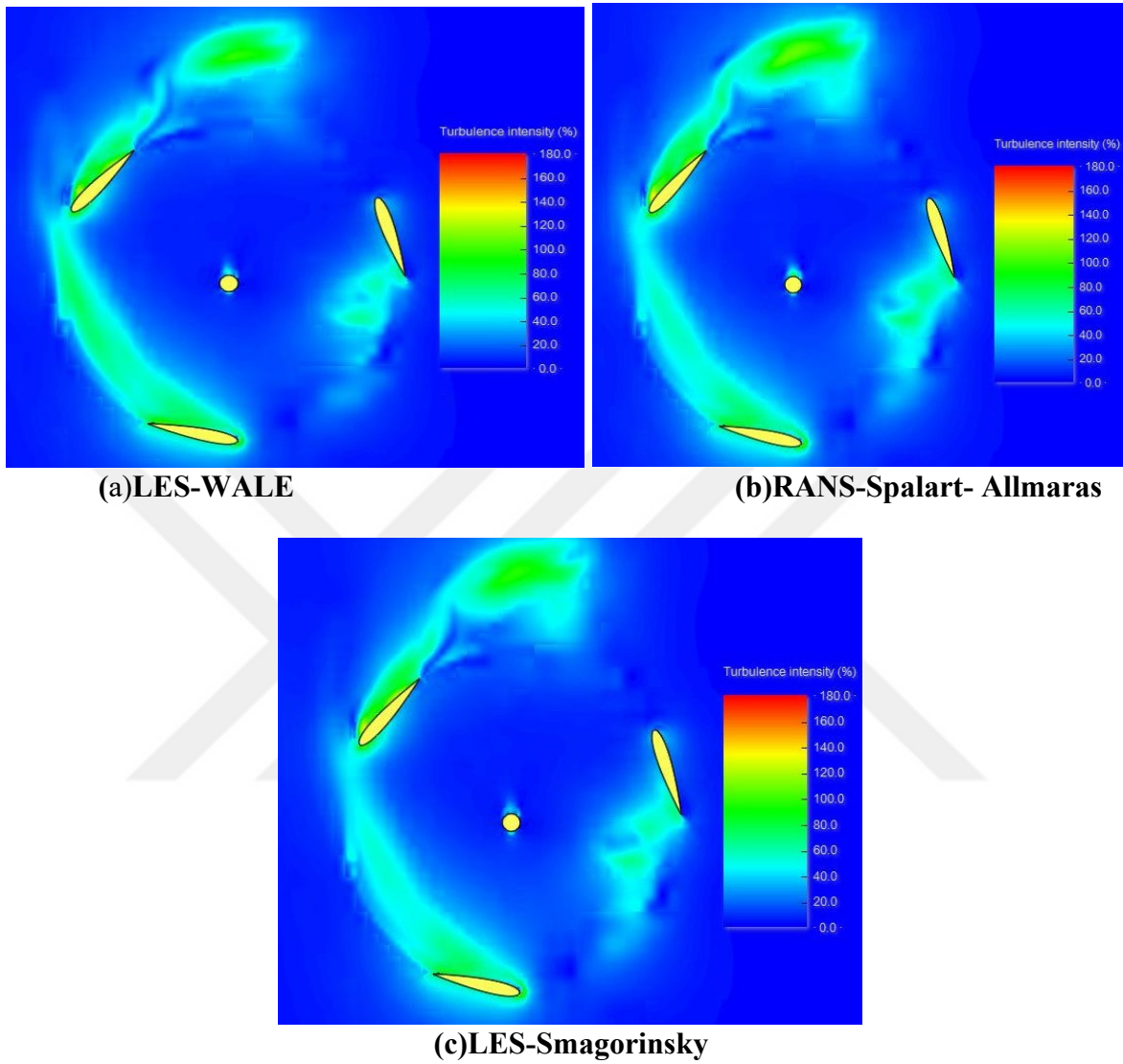


Figure 3.26. Turbulence Intensity (%) around VAWT in different turbulence models for TSR 1 and $\theta = 77^\circ$

Contours distributions were taken from XFlow CFD software involving velocities and turbulence intensities at TSR1 and azimuthal angle , θ of 77° as shown Figure 3.25. The velocity distributions on the cutting plane were demonstrated in Figure 3.26. It shows that the boundary layer of WALE and Spalart-Allmarass model give the similar separation flow from the surface of blade while the Smagorinsky turbulence model is obviously exhibiting

the flow which are separated from blade surface earlier. Figure 3.26 gives the better results that enable to understand changing viscosity effect on VAWT. Turbulence intensity provides to observe better wake predictions, the wake is separated from the trailing-edge of blade earlier and LES-WALE model contours have more intense wake structure than the other.

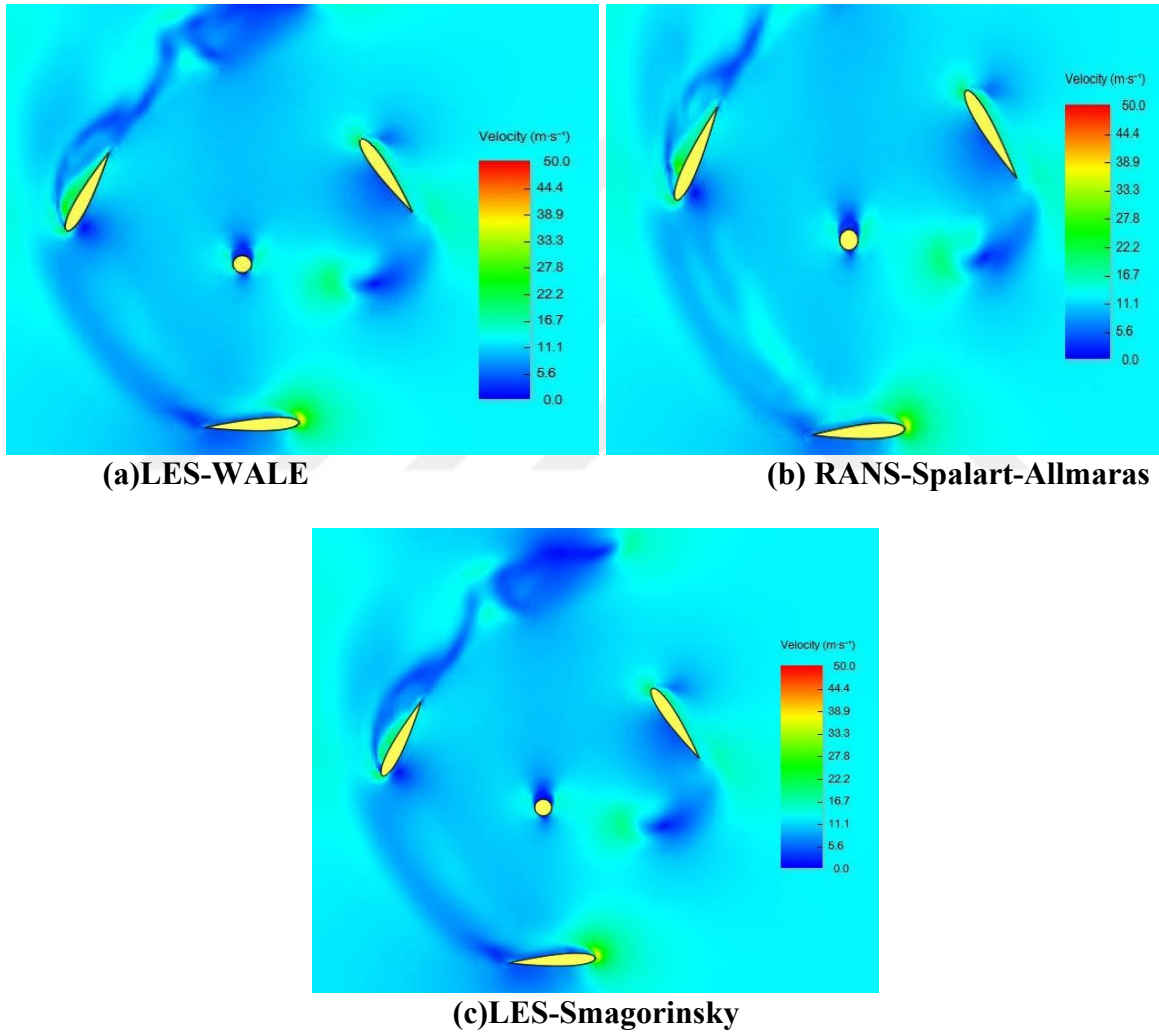


Figure 3.27. Velocity contours around VAWT in different turbulence models for TSR 1.2 and $\theta = 90^\circ$

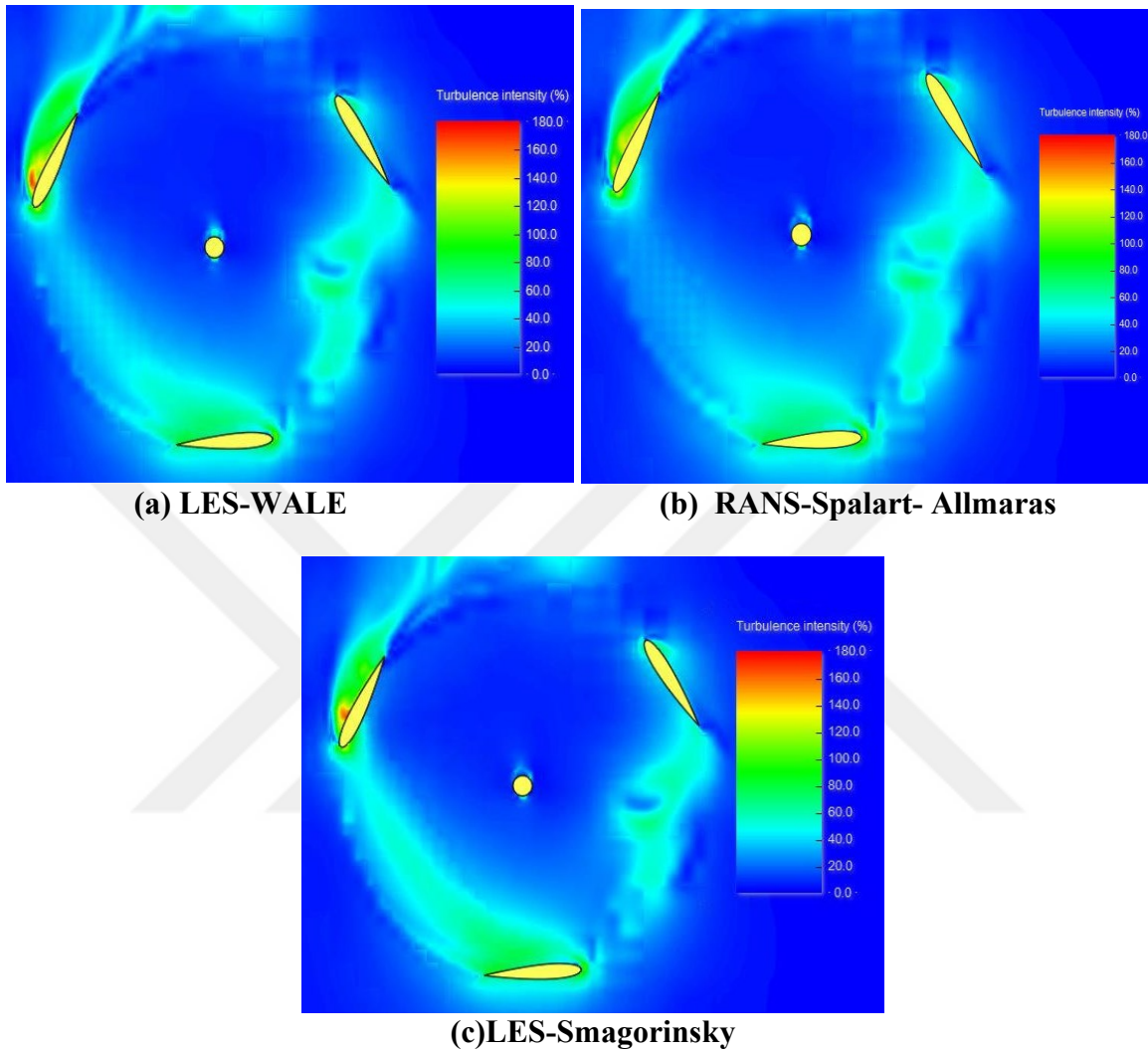


Figure 3.28. Turbulence Intensity (%) around VAWT in different turbulence models for TSR 1.2 and $\theta = 90^\circ$

For TSR 1.2 and $\theta = 90^\circ$ velocities contours are found the higher values at RANS- Spalart-Allmaras turbulence model as shown Figure 3.27. LES models shows that WALE and Smagorinsky demonstrated more turbulence density around the blade similar each other as shown Figure 3.28.

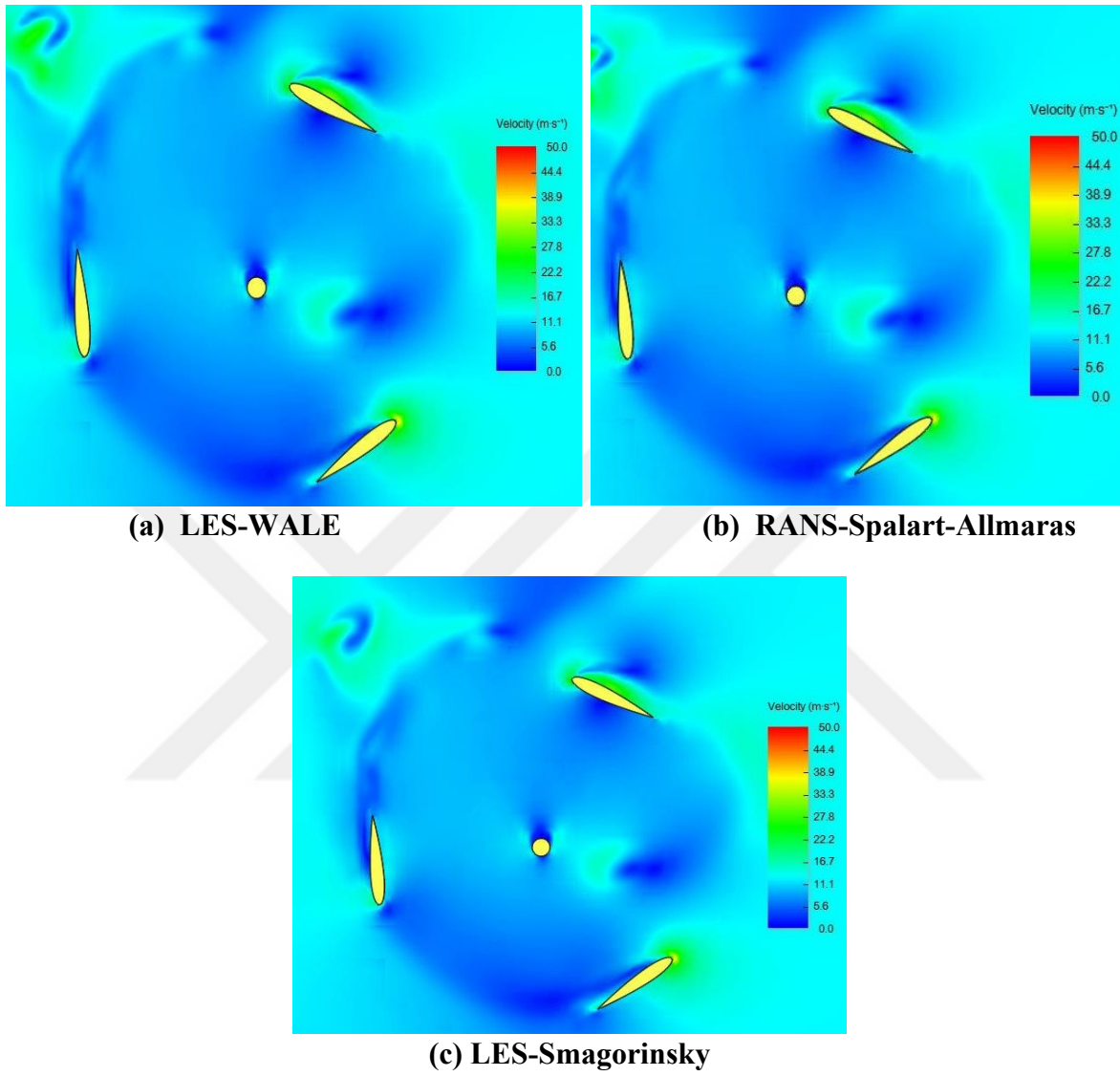


Figure 3.29. Velocity contours around VAWT in different turbulence models for TSR 1.6 and $\theta = 10^\circ$

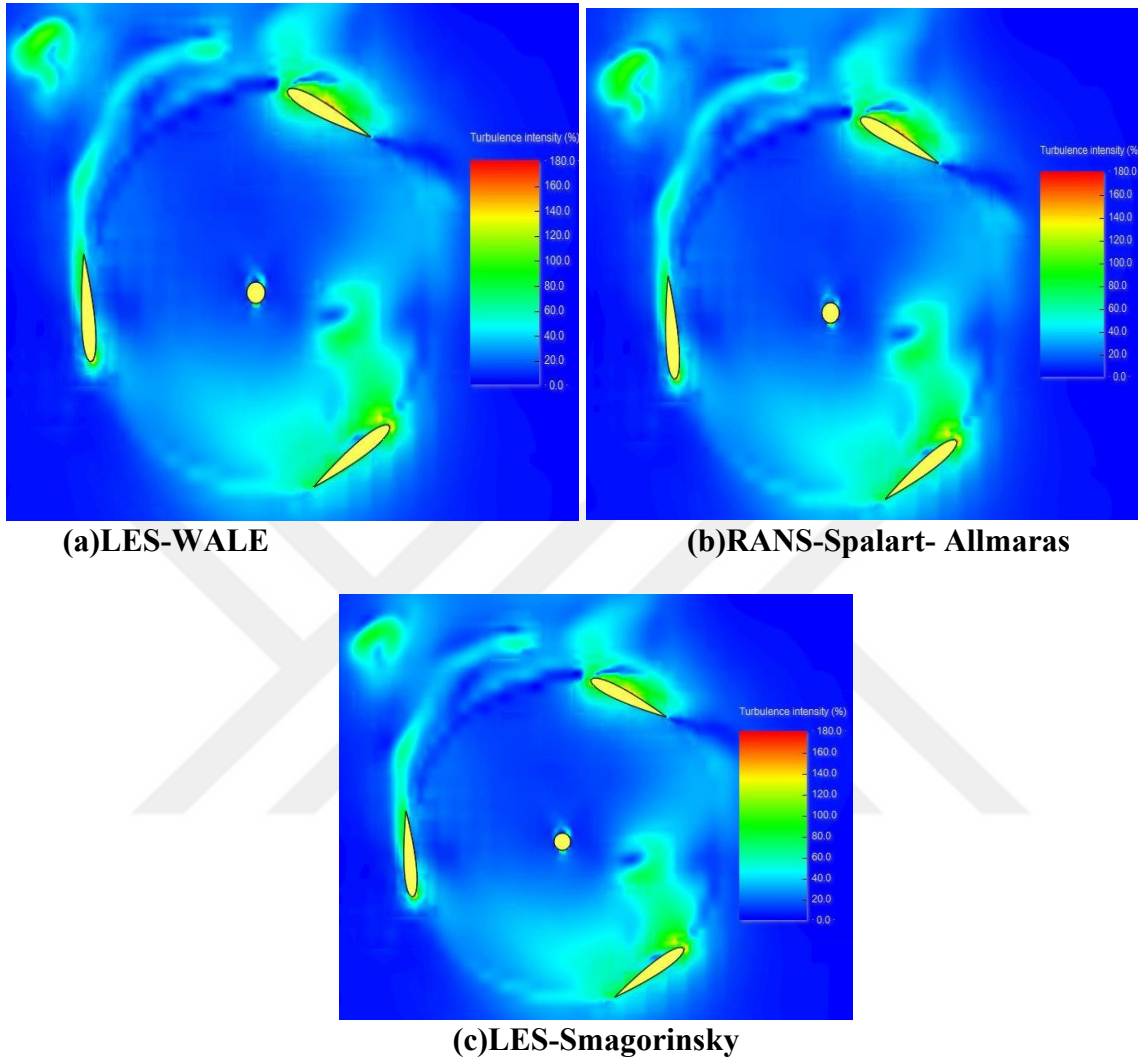


Figure 3.30. Turbulence Intensity (%) around VAWT in different turbulence models for TSR 1.6 and $\theta = 180^\circ$

As for TSR 1.6 and $\theta = 180^\circ$, RANS and LES modes reveal the similar wake structures and the higher rotor speed can be visualized to demonstrate wake predictions using with vorticity structures. RANS model, Spalart-Allmaras, is good agreement with LES-WALE model at higher rotor speed. LES -Smagorinsky showed that wake is cutting from flow separation of trailing-edge earlier as shown Figure 3.30.

Investigation of Vorticity at maximum power coefficient

At high angles of attack and low TSR, VAWTs undergo dynamic stall, leading to complex flow structures around the blades. This phenomenon involves large separated flow regions, vortex shedding, and potential blade vortex interactions. Despite the challenging aerodynamics, VAWTs can still generate power at low TSR due to these flow characteristics. Understanding and accurately modelling these flow phenomena are crucial for optimizing VAWT design and performance under dynamic stall conditions.

Vorticity behind the blades and turbine is a significant factor, performing at maximum power and torque coefficient, to enables to examine the influence of different turbulence models. As seen in below Figure 3.31-3.32-3.33, it can be seen obviously the vorticity measurements while the wind passes the around turbine and blades, it rises the value of vorticity around the blade as it rotates and entry to the path of the other blade flowing in the wind tunnel.

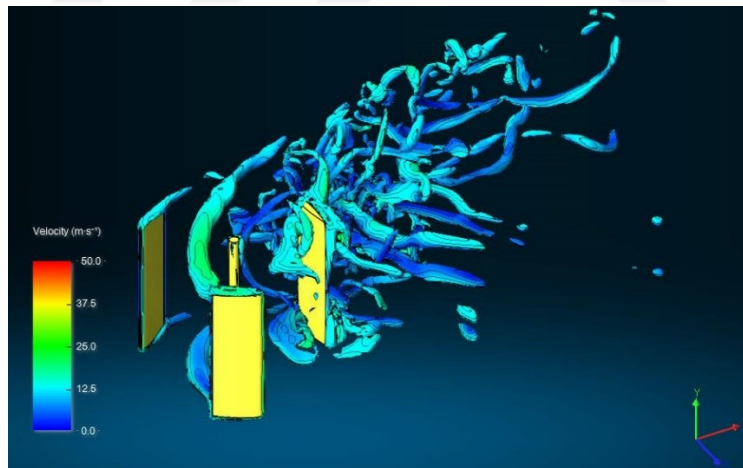


Figure 3.31. Vorticity coloured by field Velocity for LES-WALE turbulence model for TSR 1.2.

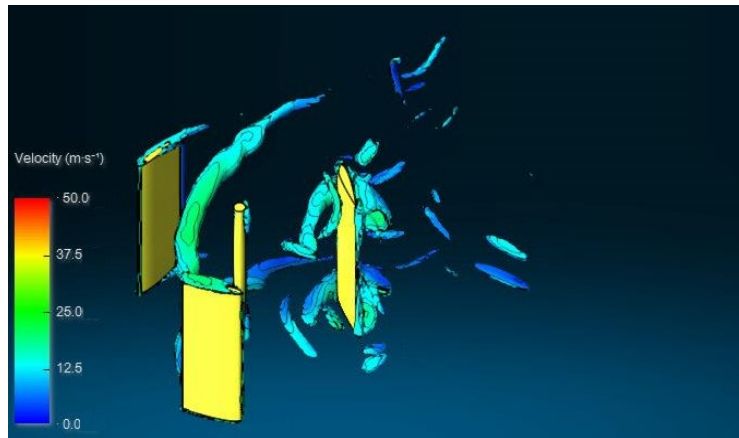


Figure 3.32. Vorticity coloured by field Velocity for RANS-Spalart-Allmaras turbulence models for TSR 1.2.

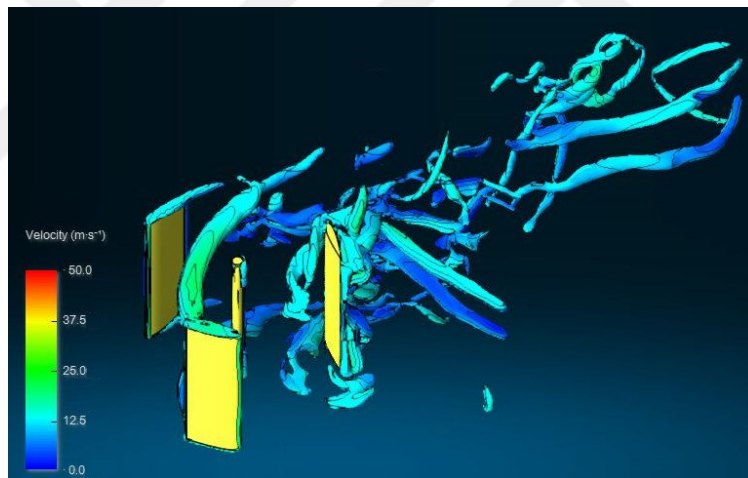


Figure 3.33. Vorticity coloured by field Velocity for LES-Smagorinsky turbulence models for TSR 1.2.

One of the basic schemes, vorticity coloured by field velocity was used to observe the wake structures in flow field. It is beneficial when compared with RANS, LES turbulence models reveal clearly existing of wake structures and velocities behind the VAWT. Furthermore, LES models, WALE and Smagorinsky, demonstrate individually the different wake structures even if they are at same class. The visualization in Figures 3.31,32 and 3.33 depicts the turbulent structures, showcasing distinct vortex shapes formed behind the

rotating blades under varying turbulence models (WALE, Smagorinsky, Spalart-Allmarass). These figures provide insights into the characteristic features of the vortices when the wind originates from the left side. In the figures, when the TSR is 1.2, WALE turbulence model shows the existing of long vorticity intense when compared to RANS-Spalart Allmarass and smagorinsky turbulence models. Thus, it was clear that LES WALE turbulence models enable to observe the wake predictions behind the VAWT.



4. FLUID-STRUCTURE INTERACTION

Fluid-Structure Interaction (FSI) is a pivotal concept in many engineering fields, including aerospace, civil, and biomedical engineering, embodying the complex relationships between fluids and solid structures. FSI simulations demand advanced numerical methods and proficient computational techniques.

Darrieus wind turbines, a specific type of vertical axis wind turbine, present an excellent case study for FSI due to the intricate interaction between the rotating blades and the surrounding wind flow. Analysing the aerodynamic performance of Darrieus wind turbines necessitates accurate simulations of this interaction (Ferreira, C. J., van Bussel, G. J., & Scarano, F., 2006).

A concurrent or weak coupling technique, where fluid and solid simulations run simultaneously and reciprocally update their boundary conditions, is now prevalent for such FSI simulations (Donea, J., & Huerta, A., 2003). Deiterding (2011a) has introduced a surface-mesh-based coupling routine for the continual updating of boundary data between the fluid and solid domains. This routine was further enhanced to communicate not only pressure forces but also viscous surface traction forces.

The Lattice Boltzmann Method (LBM), an innovative numerical technique for fluid flow simulations, has been incorporated in FSI studies in recent years, including Darrieus turbine analysis (Krüger, T., Kusumaatmaja, H., Kuzmin, A., Shardt, O., Silva, G., & Viggen, E. M., 2021). Combined with a level set signed distance function for the representation of the deforming solid's triangular surface mesh, the LBM provides an accurate depiction of the complex deformation within the fluid domain.

The modelling of the turbine's complex motion employs the Newton-Euler 6DOF motion solver (Kane, T. R., & Levinson, D. A., 1985). Slender deformable components, like the turbine blades, are represented either as quasi 2D using the Euler-Bernoulli beam solver or fully 3D using the Timoshenko Beam solver (Euler, L., 1759; Timoshenko, S. P., 1921).

The crucial task of data exchange between processors for different solvers is proficiently managed by the Eulerian-Lagrangian coupler (Deiterding, 2011b). AMROC's space-filling curve facilitates efficient domain decomposition, allowing only the fluid processors containing parts of the solid boundary to partake in the communication of interface values (Deiterding, R., 2001).

In conclusion, although simulating fluid-structure interactions, such as those in Darrieus wind turbines, poses significant computational challenges, recent advances including the use of LBM, level set methods, and efficient coupling routines have made it possible to produce more accurate and realistic FSI simulations.

4.1. Application LBM-FSI analysis on Darrieus-Wind turbine

FSI in the context of wind turbine performance is a critical area of investigation, given the increasing emphasis on renewable energy sources. XFlow, with its capability to analyse FSI scenarios, offers an advanced simulation platform to study this interaction with heightened precision.

XFlow employs the Lattice Boltzmann Method (LBM) solver, an avant-garde technique known for its computational efficiency and suitability for complex flow problems, especially in the realm of FSI. The LBM's mesoscopic approach makes it particularly adept at handling turbulent flow problems, which are inherent to wind turbine simulations.

A salient feature of the LBM solver within XFlow is its ability to accurately simulate self-rotated wind turbine performance. The self-rotation of the turbine is a result of the aerodynamic forces acting upon it, an interaction between the fluid (air) and the solid structure (turbine blades). This intricate phenomenon requires high-fidelity simulations to ensure that the turbine's rotational speed and subsequent energy generation are estimated with precision. The dynamic nature of these simulations, considering the constantly changing interaction between the wind and the rotating blades, underscores the need for a solver that can adapt in real-time. XFlow, with its adaptive LBM solver, fits this need perfectly.

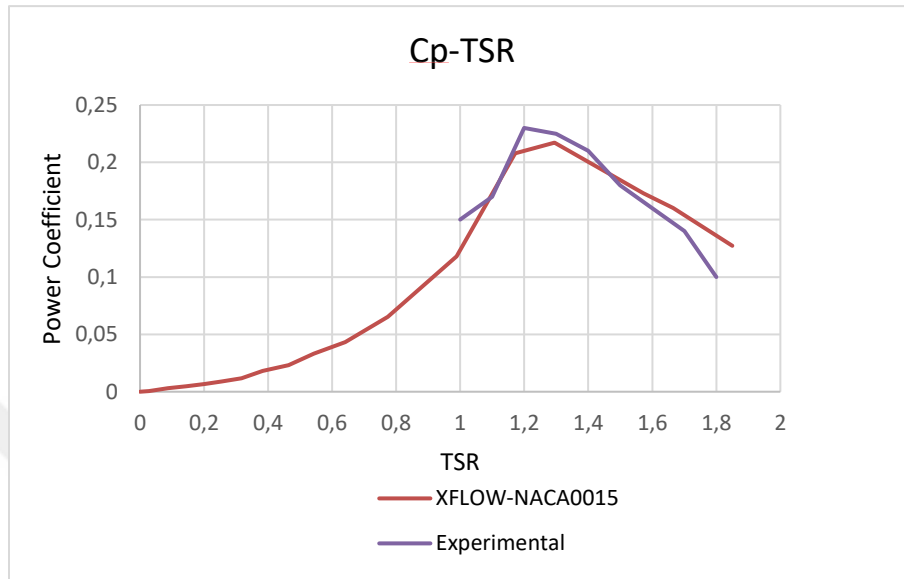


Figure 4.1. Power coefficient comparison between Experimental and LBM-FSI (Lee et al 2015)

A testament to XFlow's robust FSI capabilities is the comparative study with experimental data, particularly from the Tip Speed Ratio (TSR) parameter. TSR is a pivotal metric in wind turbine analysis, indicating the efficiency of the turbine at varying wind speeds. Figure 4.1 demonstrates comparing XFlow's simulation outcomes with experimental TSR data, it's possible to validate the software's accuracy and reliability. This exercise further consolidates the software's position as a trustworthy tool for simulating wind turbine performance, especially in scenarios involving intricate fluid and solid interactions

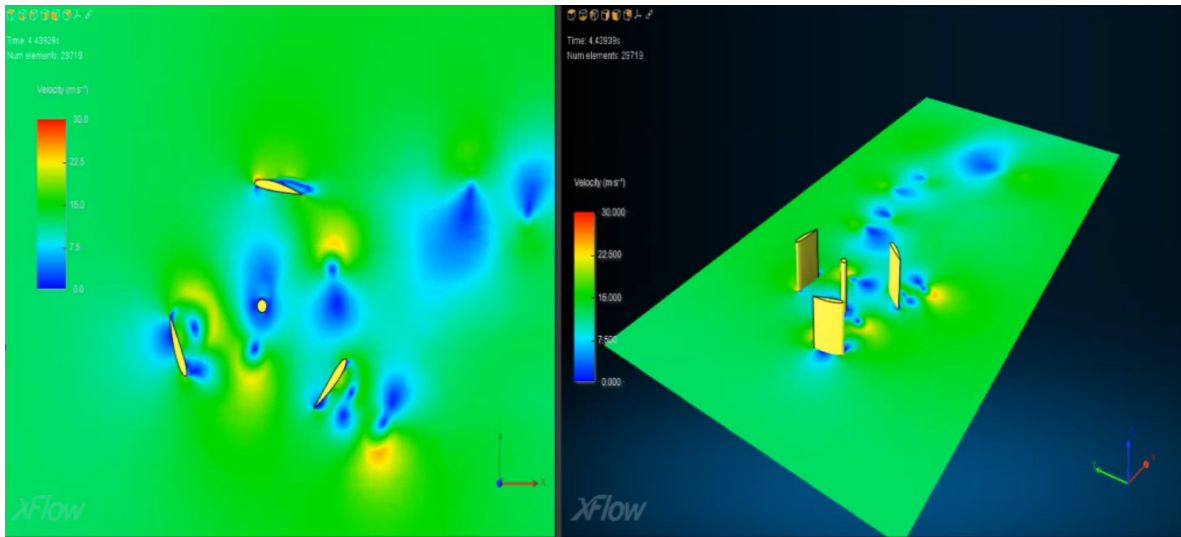


Figure 4.2. Contours velocity of Dairius wind turbine by using LBM- Xflow.

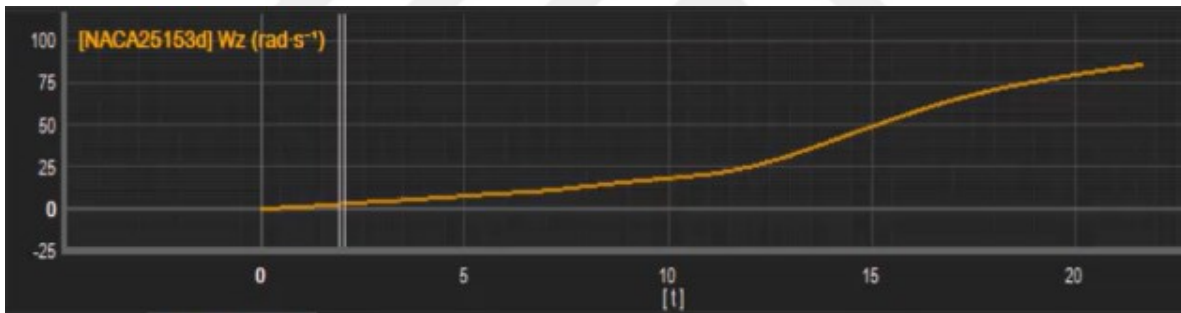


Figure 4.3. Angular velocity changes from self-starting position on Dairius wind turbine

Figure 4.2 demonstrates the contours change in the aerodynamic force effecting and Figure 4.3, the angular velocity of turbine is increasing from self-starting position.

5. CONTRIBUTIONS

This doctoral research represents a significant advancement in the field of computational fluid dynamics (CFD) through the development and exploration of solution-adaptive, meshless computational methods rooted in the Lattice Boltzmann Method (LBM), specifically tailored for the precise simulation of complex flows around oscillating circular cylinders. By addressing the inherent limitations of traditional mesh-based CFD approaches, this study introduces innovative solutions that not only enhance computational efficiency but also ensure high accuracy across a diverse spectrum of engineering applications. The multifaceted contributions of this research are delineated as follows:

5.1. Significance of Solution-Adaptive Meshless Methods

At the heart of this investigation lies the formulation of a novel solution-adaptive methodology, which capitalizes on the inherent strengths of the Lattice Boltzmann Method within a meshless CFD framework. This groundbreaking approach circumvents the necessity for predefined meshes, instead employing scattered or field nodes to delineate the problem domain, while dynamically adapting the computational mesh in response to evolving flow characteristics. Such adaptivity engenders a substantial reduction in computational costs while concurrently preserving high levels of accuracy, rendering it particularly well-suited for the analysis of intricate flow phenomena.

5.2. Integration with Lattice Boltzmann Method

This research represents a pioneering effort in the integration of the Lattice Boltzmann Method (LBM) with solution-adaptive meshless techniques, thereby establishing a robust framework for simulating fluid flow dynamics. This integration capitalizes on the unique strengths of both approaches to overcome the limitations of traditional methods and achieve more accurate and efficient simulations. The mesoscopic nature of the LBM, rooted in microscopic modeling and kinetic theory, provides a solid foundation for representing fluid behavior at a fundamental level. By discretizing the fluid domain into discrete particles and simulating their interactions according to kinetic principles, LBM offers a simplified yet

profound representation of fluid dynamics. This approach ensures alignment between macroscopic outcomes and theoretical predictions, thereby enhancing the accuracy of simulations. Moreover, by integrating LBM with solution-adaptive meshless techniques, this research enables the accurate modeling of complex flows spanning multiple scales and phases. Meshless techniques, which eschew the need for predefined grids and instead rely on scattered nodes to define the computational domain, offer flexibility and adaptability in capturing flow dynamics. The combination of LBM's mesoscopic approach with solution-adaptive meshless techniques allows for the seamless adaptation of the computational mesh based on flow characteristics, leading to enhanced computational efficiency without compromising accuracy.

Overall, this integration of LBM with solution-adaptive meshless techniques represents a significant advancement in the simulation of fluid flow dynamics. By leveraging the strengths of both approaches, this research opens new avenues for accurately modeling complex flows encompassing multi-scale and multiphase phenomena, thereby contributing to the advancement of computational fluid dynamics and its applications across various fields of science and engineering.

5.3. Enhanced Handling of Complex Geometries

Conventional computational fluid dynamics (CFD) methodologies frequently face hurdles when confronted with the task of effectively tackling complex geometries and boundary conditions. However, the research at hand accentuates the remarkable effectiveness of the proposed methodologies in seamlessly addressing intricate geometries and complex boundary conditions. This superiority is primarily ascribed to two key factors: the inherent flexibility of the lattice-based grid structure in the Lattice Boltzmann Method (LBM) and the dynamic adaptability intrinsic to the meshless framework.

The lattice-based grid structure utilized in LBM exhibits notable flexibility, allowing for the representation of complex geometries with relative ease. Unlike traditional mesh-based methods, which often necessitate intricate mesh generation procedures to conform to irregular geometries, LBM inherently accommodates such complexities through its lattice

structure. This inherent flexibility minimizes the computational overhead associated with mesh generation while ensuring accurate representation of intricate geometries.

Furthermore, the dynamic adaptability characteristic of the meshless framework employed in the proposed methodologies further enhances their efficacy in handling complex flow domains. By eschewing the reliance on predefined meshes and instead utilizing scattered or field nodes to define the problem domain, the meshless framework offers unparalleled versatility in adapting to evolving flow conditions. This adaptability allows for the seamless adjustment of the computational mesh based on flow characteristics, thereby enabling robust modeling of intricate flow domains without the constraints imposed by traditional mesh-based approaches. Collectively, these attributes empower the proposed methodologies to excel in addressing the challenges posed by complex geometries and boundary conditions. By leveraging the inherent flexibility of the lattice-based grid structure in LBM and the dynamic adaptability of the meshless framework, these methodologies facilitate accurate and efficient modeling of intricate flow domains, thereby advancing the frontier of computational fluid dynamics research.

5.4. Application to Engineering Problems

The findings of this research undergo rigorous validation against benchmark problems, a standard practice in the field of computational fluid dynamics (CFD), to ensure their accuracy and reliability. Subsequently, these validated findings are applied to investigate flow phenomena around circular cylinders, a topic of significant interest due to its relevance across various industrial applications. This application serves as a compelling demonstration of the efficacy of the proposed method in capturing critical flow phenomena inherent in such scenarios. One of the key flow phenomena addressed in this context is separation, where the flow detaches from the surface of the cylinder due to adverse pressure gradients. Properly capturing separation phenomena is crucial for accurately predicting aerodynamic forces and understanding the overall flow behavior around the cylinder. Additionally, the method is evaluated for its ability to capture vortex shedding, a phenomenon characterized by the formation of alternating vortices in the wake region of

the cylinder. Vortex shedding plays a vital role in determining the aerodynamic forces acting on the cylinder and is essential for various engineering applications such as structural design and energy extraction from fluid flows.

Furthermore, the transition to turbulence is another critical aspect examined in this study. As the flow around the cylinder undergoes transition from laminar to turbulent regimes, there are significant changes in flow characteristics, including increased drag and fluctuating forces. Accurately predicting this transition is essential for understanding the overall flow dynamics and its implications for engineering design.

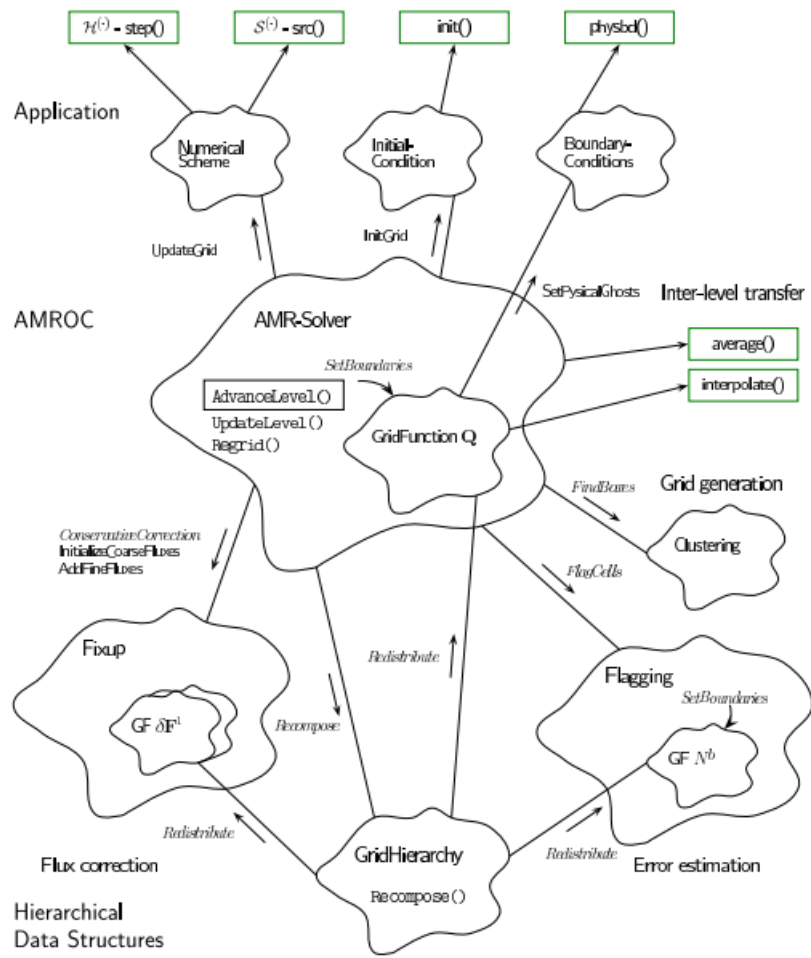
By successfully capturing these pivotal flow phenomena—separation, vortex shedding, and transition to turbulence—the proposed method demonstrates its efficacy in providing accurate predictions of flow-induced forces around circular cylinders. This not only enhances our understanding of complex flow behaviors but also contributes to improving the design and performance of various industrial applications, ranging from aerospace to renewable energy. Thus, the findings of this research have far-reaching implications for advancing the field of fluid dynamics and its practical applications in engineering.

5.5. Performance in Turbulent Flow Simulation

The proposed methodologies showcased in this research exhibit exceptional proficiency in simulating turbulent flows, a cornerstone in the realm of CFD. This prowess stems from several key factors, including efficient parallelization strategies and the innate framework of the LBM, which is adept at seamlessly integrating complex physics into simulations. Efficient parallelization strategies play a pivotal role in enhancing the scalability and performance of turbulent flow simulations. By distributing computational tasks across multiple processors or computing nodes, parallelization significantly reduces simulation time and enables the handling of larger and more complex problem domains. This allows for the simulation of turbulent flows with higher fidelity and resolution, leading to more accurate predictions of flow behavior. Moreover, the inherent framework of LBM lends itself well to the incorporation of complex physics, including thermal effects and fluid-structure interactions. LBM's mesoscopic approach, which models fluid dynamics at a

microscopic level, allows for the representation of intricate physical phenomena with relative simplicity. This facilitates the accurate simulation of turbulent flows in various scenarios, making it particularly advantageous for aerodynamic analyses and the examination of multiphase flows characterized by phase change phenomena.

Additionally, enhancements to the AMROC code further bolster the efficiency of turbulent flow simulations. AMROC, a computational framework for solving partial differential equations, is optimized for the implementation of turbulence models such as the constant Smagorinsky model. These enhancements improve the computational robustness of the approach, ensuring reliable and accurate simulations of turbulent flows across a range of engineering applications. In summary, the proposed methodologies leverage efficient parallelization strategies, harness the innate framework of LBM, and benefit from enhancements to the AMROC code to excel in simulating turbulent flows. This capability is invaluable for conducting aerodynamic analyses and comprehensively examining multiphase flows, thereby advancing our understanding of complex fluid dynamics phenomena and informing engineering design processes.



Design of the AMROC framework. GF stands for GridFunction.

Figure 5.1. AMROC implements the AMR method in C++ and can be used directly in C++ or with Fortran functions called from C++ interface objects.(Deiterding ,2011)

6. CONTRIBUTION TO WIND ENERGY RESEARCH

An exemplary application of the methodologies developed in this research is their utilization in a thorough investigation of the aerodynamic performance of Darrieus-type vertical axis wind turbines (VAWTs). This investigation encompasses the utilization of various turbulence models to analyze and evaluate the turbine's performance comprehensively. Notably, the research extends beyond mere validation of these turbulence models in terms of torque and power coefficients; it delves deeper to provide invaluable insights into the nuanced influence of turbulence models on the aerodynamic performance of VAWTs. The study rigorously assesses the predictive accuracy and reliability of different turbulence models employed in simulating the flow dynamics around Darrieus-type VAWTs. By comparing the simulated results with experimental data, the research evaluates the capability of each turbulence model to accurately predict important performance parameters such as torque and power coefficients. This validation process ensures that the chosen turbulence models can effectively capture the intricate flow phenomena present in VAWTs' operating conditions, thus enhancing confidence in the simulation results. Furthermore, the research goes beyond mere validation by examining how different turbulence models impact the aerodynamic performance of VAWTs in nuanced ways. By systematically comparing the results obtained with various turbulence models, the study identifies and elucidates the specific strengths and limitations of each model in capturing critical flow features and predicting turbine performance. These insights offer valuable guidance for researchers and engineers in selecting the most suitable turbulence model for simulating VAWTs under different operating conditions.

Overall, this contribution significantly enriches the body of knowledge within the realm of wind energy research by providing comprehensive insights into the aerodynamic performance of Darrieus-type VAWTs. The thorough investigation of turbulence models and their impact on turbine performance not only advances our understanding of VAWT aerodynamics but also informs the development of more accurate and reliable simulation tools for optimizing wind turbine design and operation.

7. ADVANCING SIMULATION TECHNIQUES

Through the utilization of cutting-edge simulation techniques, this doctoral research has made significant strides in advancing our understanding of complex flow dynamics. By employing solution-adaptive methods with dynamic mesh adaptation, the study has been able to finely tune computational meshes in response to evolving flow conditions, resulting in more accurate and efficient simulations. Furthermore, rigorous comparative analyses against experimental data and alternative numerical models have provided invaluable insights into the behavior of complex flows, shedding light on phenomena that were previously poorly understood. The integration of these advanced simulation techniques has not only propelled the frontier of simulation methodologies but has also laid the groundwork for future innovations in fluid dynamics research. By delving deep into the intricacies of flow dynamics and associated phenomena, this research has expanded our knowledge base and paved the way for the development of more sophisticated simulation tools and analytical approaches.

In summary, this doctoral thesis represents a significant leap forward in the simulation of complex flows. Its comprehensive, efficient, and rigorously accurate approach transcends conventional CFD methodologies, offering new avenues for analyzing fluid dynamics. The manifold contributions of this research have the potential to catalyze a paradigm shift in the way engineers and scientists approach the analysis of fluid dynamics, with far-reaching implications across diverse fields of study and numerous industrial applications. Furthermore, it is noteworthy that the successful completion of the AEROFSI project, funded by the Ministry of Science, Industry, and Technology, underscores the practical relevance and real-world impact of this research endeavor. This collaboration highlights the importance of academic-industry partnerships in driving innovation and advancing scientific knowledge for the benefit of society as a whole.

8. CONCLUSIONS

In this work, Darieus-type 500 W straight-bladed Vertical axis wind turbine (VAWT) was investigated with using a XFlow commercial software, using a meshingless Lattice Boltzman method, for different turbulence models in fine and coarse lattices.

In order to assess the validity of the proposed numerical modelling, offering the meshingless lattice Boltzmann method, which users can avoid the remeshing process for CFD simulation enables to investigate the aerodynamic performance of wind turbine with less amount of lattices, save the time and computational costs.

Firstly, the influence of different turbulence modelling approaches on solution time was investigated depending on lattice sizes, it was appeared that RANS model can give faster solution than LES approaches even when number of lattices is increased and the increase of number of lattices is not changing the solution time proportionally.

Secondly, experimental data of torque coefficient obtained from (Lee Y., Lim H 2015) were compared with different turbulence models in fine and coarse lattices. LES models, WALE and Smagorinsky demonstrated the best agreement with experimental data. Especially, Smagorinsky model can be useful when less computational cost is required and WALE is a suitable turbulence model that is best agreement with experimental data in order to capture the maximum torque and power coefficient as users are analysing the VAWT at the fine lattices.

The effect of turbulence models is achieved by the velocity and turbulence intensity contours at different TSR values. RANS and LES model are giving an observable wake structure behind the rotating blades, especially at maximum torque coefficient. However, LES model is useful when compared to result with experimental in fine lattices.

Result is also given by the investigation of vorticity at maximum power coefficient at varying azimuthal angle. LES models are exhibiting considerably difference the wake predictions when compared with RANS. LES-WALE model has more intense the wake

structures, it is reasonable effect of LES model constant, and its operator based on formula Including rotation tensor.

In addition, a comprehensive sensitivity analysis has been conducted to further investigate the behaviour of VAWTs. Surprisingly, the study reveals that capturing the flow phenomenon does not require an excessively fine grid, despite the significant flow separation observed. The simulation results are compared with both numerical and experimental data, and although some discrepancies exist, they are relatively minor when using fine lattices.

Moreover, the study highlights an interesting finding: the power performance of the VAWT increases as the TSR rises, particularly at low TSRs. However, at high TSRs, the interaction between vortices and blades becomes more prominent and significantly influences the blade's performance. This work was supported by Scientific Research Projects Commission (BAPKO) of Marmara University, under the grant number FEN-C-DRP-130515-0179.

REFERENCES

- [1] Amine, B. A. M. E., Chekib, K., & SASSI, B. N. (2012). Computational Fluid Dynamics study of Vertical Axis Wind Turbines in different shapes. In VIème Congrès International sur les Energies Renouvelables et l'Environnement (pp. 19-21). Hammamet, Tunisie.
- [2] Barrios, F.A., Tam, L., & Moghimi, S. (2016). A review on meshless methods for computational fluid dynamics simulations. *Journal of Computational Science*, 17, 17-33. <https://doi.org/10.1016/j.jocs.2016.05.005>
- [3] Belytschko, T., Lu, Y. Y., & Gu, L. (1994). Element-free Galerkin methods. *International Journal for Numerical Methods in Engineering*, 37(2), 229-256.
- [4] Berger, M. J., & Collela, P. (1989). Local Adaptive Mesh Refinement for Shock Hydrodynamics. *Journal of Computational Physics*, 82(1), 64-84. [https://doi.org/10.1016/0021-9991\(89\)90035-1](https://doi.org/10.1016/0021-9991(89)90035-1)
- [5] Bhatnagar, P. L., Gross, E. P., & Krook, M. (1954). A Model for Collision Processes in Gases. I. Small Amplitude Processes in Charged and Neutral One-Component Systems. *Physical Review*, 94(3), 511-525. <https://doi.org/10.1103/PhysRev.94.511>
- [6] Chen, L., Huang, H., & Li, X. (2022). Microscale gas flow simulation using the lattice Boltzmann method with thermal creep boundary conditions. *Physical Review E*, 102(1), 013306.
- [7] Chen, S., & Doolen, G. D. (1998). Lattice Boltzmann Method for Fluid Flows. *Annual Review of Fluid Mechanics*, 30(1), 329-364. <https://doi.org/10.1146/annurev.fluid.30.1.329>
- [8] Chen, Q., Peng, Y., Zhang, D., & Shu, C. (2019). Large eddy simulation of turbulent flow over a circular cylinder with the lattice Boltzmann method. *International Journal of Heat and Fluid Flow*, 75, 69-83.
- [9] Chen, Y., Wang, Y., & Chen, H. (2022). Lattice Boltzmann simulation of microfluidics applications. *Micromachines*, 13(1), 8.
- [10] Deiterding, R. (2001). Construction and application of an AMR algorithm for

- distributed memory computers. Proceedings of the 3rd International Workshop on Parallel Algorithms for Irregularly Structured Problems, 198-204.
- [11] Deiterding, R. http://rdeiterding.website/pub/amroc_usage.pdf
 - [12] Deiterding, R. (2002). An Adaptive Mesh Refinement Method for Simulations of Strongly Transient Detonations and DDT. *Journal of Computational Physics*, 175(1), 59-82. <https://doi.org/10.1006/jcph.2001.6960>
 - [13] Deiterding, R. (2011a). Block-structured adaptive mesh refinement in object-oriented C++.
 - [14] Deiterding, R. (2011b). Parallel adaptive simulation of multi-material flows with strong shocks.
 - [15] Donea, J., & Huerta, A. (2003). *Finite element methods for flow problems*. John Wiley & Sons.
 - [16] Elkhoury, M., Kiwata, T., & Aoun, E. (2015). Experimental and numerical investigation of a three-dimensional vertical-axis wind turbine with variable-pitch. *Journal of Wind Engineering and Industrial Aerodynamics*, 139, 111–123.
 - [17] Euler, L. (1759). *Methodus inveniendi lineas curvas maximi minimive proprietate gaudentes*.
 - [18] Ferreira, C. J., van Bussel, G. J., & Scarano, F. (2006). Visualization by PIV of dynamic stall on a vertical axis wind turbine. *Experiments in fluids*, 42(5), 737-747.
 - [19] Filler, G. L., Savory, E., & Schultz, W. W. (1991). Hydrodynamics of bodies in steady and unsteady vortex streets. *AIAA journal*, 29(10), 1660-1666.
 - [20] Germano, M., Piomelli, U., Moin, P., & Cabot, W. H. (1991). A dynamic subgrid-scale eddy viscosity model. *Physics of Fluids A: Fluid Dynamics*, 3(7), 1760-1765.
 - [21] Gingold, R. A., & Monaghan, J. J. (1977). Smoothed particle hydrodynamics: Theory and application to non-spherical stars. *Monthly Notices of the Royal Astronomical Society*, 181(3), 375-389.
 - [22] Ginzburg, I., d'Humières, D., & Krafczyk, M. (2014). Mesoscopic modeling of surfactant-covered droplets with the lattice Boltzmann method. *Computers & Mathematics with Applications*, 68(7), 863-880.
 - [23] Guo, Z., Chen, S. (2013). *Lattice Boltzmann Method: Fundamentals and*

Engineering Applications. John Wiley & Sons.

- [24] Holman, D. M., Brionnaud, R. M., & Abiza, Z. (n.d.). Solution to industry benchmark problems with the lattice-Boltzmann code XFlow. Next Limit Technologies.
- [25] Hou, S., et al. (1996). A lattice Boltzmann subgrid model for high Reynolds number flows. *Fields Institute Communications*, 6, 151–165.
- [26] Howell, R., Qin, N., Edwards, J., & Durrani, N. (2010). Wind tunnel and numerical study of a small vertical axis wind turbine. *Renewable Energy*, 35, 412-422.
- [27] Huang, C., Zhou, P., & Meng, J. (2022). Lattice Boltzmann modeling of gas–liquid two-phase flow in porous media. *Journal of Petroleum Science and Engineering*, 195, 107978.
- [28] Huang, Y., Xu, H., Liu, G., & Wang, X. (2022). Application of the lattice Boltzmann method to simulate complex geometries and boundary conditions. *Computers & Mathematics with Applications*, 83, 235-247.
- [29] Hughes, T. J., Liu, W. K., & Zimmermann, T. K. (1981). Lagrangian-Eulerian finite element formulation for incompressible viscous flows. *Computer methods in applied mechanics and engineering*, 29(3), 329-349.
- [30] Islam, M., Fartaj, A., & Cariveau, R. (2008). Analysis of the design parameters related to a fixed-pitch straight-bladed vertical axis wind turbine. *Wind Engineering*, 32(5), 491–507.
- [31] Jiange, H., Yong, H., Yuan, S., & Li, Z. (2016). Numerical study on the characteristics of laminar wakes behind rotating cylinders at high Reynolds numbers. *Computers & Fluids*, 134, 16-28.
- [32] Kane, T. R., & Levinson, D. A. (1985). *Dynamics, theory and applications*. McGraw-Hill New York.
- [33] Krafczyk, M., Tölke, J., & Luo, L. S. (2003). Large eddy simulations with a multiple-relaxation-time LBE model. *International Journal of Modern Physics B*, 17, 33-39.
- [34] Krüger, T., Kusumaatmaja, H., Kuzmin, A., Shardt, O., Silva, G., & Viggen, E. M. (2021). *The Lattice Boltzmann Method: Principles and Practice*. Springer Nature.

- [35] Laloglu, C., Alpman, E. (2023). Mesh-Free Analysis of a Vertical Axis Wind Turbine Using Lattice Boltzmann Method and Various Turbulence Models. Department of Mechanical Engineering, Marmara University, 34722 Istanbul, Turkey. *Appl. Sci.* 2023, 13(15), 8800; <https://doi.org/10.3390/app13158800>.
- [36] Laloglu, C., Deiterding, R. (2017). Simulation of the Flow around an Oscillating Cylinder with Adaptive Lattice Boltzmann Methods. Conference Paper, May 2017. DOI: 10.4203/ccp.111.19
- [37] Lee, Y., & Lim, H. (2015). Numerical study of the aerodynamic performance of a 500 W Darrieus-type vertical-axis wind turbine. *Renewable Energy*, 83, 407-415.
- [38] Li, Q., He, Y. L., & Zhang, W. M. (2017). Lattice Boltzmann simulation of electrokinetic flow in microchannels. *Journal of Colloid and Interface Science*, 490, 777-786.
- [39] Li, S., Zheng, Y., Zhang, X., & Luo, K. H. (2022). Large-scale lattice Boltzmann simulations of turbulent channel flows on a GPU cluster. *International Journal of Heat and Mass Transfer*, 157, 119874.
- [40] Lilly, D. K. (1992). A proposed modification of the Germano subgrid-scale closure method. *Physics of Fluids A: Fluid Dynamics*, 4(3), 633-635.
- [41] Liu, H., Guo, Z., Li, H., & Fang, H. (2019). Lattice Boltzmann simulation of the dynamic behavior of droplet coalescence. *Journal of Colloid and Interface Science*, 536, 128-
- [42] Liu, W. K., Jun, S., & Zhang, Y. F. (1995). Reproducing kernel particle methods. *International Journal for Numerical Methods in Fluids*, 20(8-9), 1081-1106.
- [43] Liu, Y., Wang, S., Li, S., & Zhao, X. (2018). Lattice Boltzmann simulation of flow through porous media: Effect of porosity and permeability. *International Journal of Heat and Mass Transfer*, 123, 317-328.
- [44] Liu, Y., Wang, Y., Zhang, H., & Zhang, Y. (2022). Lattice Boltzmann simulation of two-dimensional heat transfer in porous media with solid matrix heterogeneity. *International Journal of Heat and Mass Transfer*, 157, 119876.
- [45] Lausová L., Kološ I., Michalcová V, (2019). Comparison of 2D grid simulations for flow past cylinder at high Reynolds numbers. *Civil and Environmental Engineering*,

- 15(1), 70-78. DOI: 10.2478/cee-2019-0010.
- [46] Lucy, L. B. (1977). A numerical approach to the testing of the fission hypothesis. *The Astronomical Journal*, 82(12), 1013-1024.
 - [47] Michael, C. S., & Daniel, T. T. Jr. (2006). *Lattice Boltzmann Modeling: An Introduction for Geoscientists and Engineers* (1st ed.). Springer.
 - [48] Musavi, S. M., Ashrafizaadeh, M. (2016). A mesh-free lattice Boltzmann solver for flows in complex geometries. *International Journal of Heat and Fluid Flow*, 59, 10-19.
 - [49] Nazarinia, M., Lai, J. C. S., & Xiao, F. (2013). Wake structures behind a forced cylinder at low Reynolds numbers. *Journal of Fluids and Structures*, 42, 35-48.
 - [50] Park, J. W. (2011). Application of mesh free lattice Boltzmann method to the analysis of very high temperature reactor lower plenum. *Atw. Internationale Zeitschrift fuer Kernenergie*, 43(9).
 - [51] Park, J. W. (n.d.). Application of Mesh Free Lattice Boltzmann Method to the Analysis of Very High Temperature Reactor Lower Plenum. *Atw. Internationale Zeitschrift fuer Kernenergie*, 57(11), 636-640.
 - [52] Qian, Y. H., d'Humieres, D., & Lallemand, P. (1992). Lattice BGK models for Navier-Stokes equation. *EPL (Europhysics Letters)*, 17(6), 479.
 - [53] Safdari, A., & Kim, K. C. (2015). Aerodynamic and Structural Evaluation of Horizontal Archimedes Spiral Wind Turbine. *Journal of Clean Energy Technologies*, 3(1), 34-38.
 - [54] Sagaut, P. (2006). *Large Eddy Simulation for Incompressible Flows: An Introduction*. Springer. <https://doi.org/10.1007/978-3-540-26344-9>
 - [55] Sarpkaya, T. (2004). A critical review of the intrinsic nature of vortex-induced vibrations. *Journal of Fluids and Structures*, 19(4), 389-447.
 - [56] Shan, X., Chen, H., & Ouyang, Q. (2015). Lattice Boltzmann model for simulating flows with multiple phases and components. *Physical Review E*, 52
 - [57] Smagorinsky, J. (1963). General circulation experiments with the primitive equations: I. The basic experiment. *Monthly Weather Review*, 91(3), 99-164.
 - [58] Succi, S. (2001). *The Lattice Boltzmann Equation: For Fluid Dynamics and*

Beyond. Oxford University Press.

- [59] Sussman, M., & Puckett, E. G. (2000). A coupled level set and volume-of-fluid method for computing 3D and axisymmetric incompressible two-phase flows. *Journal of Computational Physics*, 162(2), 301-337.
- [60] Tang, Y., Zhang, Z., & Luo, K. H. (2023). Multiphase lattice Boltzmann simulations of boiling heat transfer: Influence of substrate wettability. *International Journal of Heat and Mass Transfer*, 152, 119550.
- [61] Tang, Y., Ding, C., & Peng, X. F. (2023). Lattice Boltzmann simulation of multiphase flows with phase change phenomena. *International Journal of Heat and Mass Transfer*, 174, 121594.
- [62] Timoshenko, S. P. (1921). On the correction for shear of the differential equation for transverse vibrations of prismatic bars. *London, Edinburgh, and Dublin Philosophical Magazine and Journal of Science*, 41(245), 744-746.
- [63] Tokumaru, P. T., Dimotakis, P. E., & Lim, J. (2008). Effects of transverse oscillation on flow past a circular cylinder. *Journal of Fluid Mechanics*, 615, 253-272.
- [64] Vreman, A. W., Geurts, B. J., & Kuerten, J. G. (1997). Realizability-preserving dynamic subgrid-scale models in large-eddy simulation. *Physics of Fluids*, 9(9), 2607-2613
- [65] Wang, Y., Zhou, H., & Zhang, X. (2023). Large-scale lattice Boltzmann simulation of transitional channel flows on GPU clusters. *International Journal of Heat and Mass Transfer*, 170, 120693.
- [66] William, P., James, R., & Smith, A. B. (2004). *Fluid dynamics in the natural environment*. John Wiley & Sons.
- [67] Williamson, C. H. K., & Roshko, A. (1988). Vortex formation in the wake of an oscillating cylinder. *Journal of Fluids and Structures*, 2(4), 355-381. [https://doi.org/10.1016/0889-9746\(88\)90009-6](https://doi.org/10.1016/0889-9746(88)90009-6)
- [68] Weickert, M., Teike, G., Schmidt, O., & Sommerfeld, M. (2010). Investigation of the LES WALE turbulence model within the lattice Boltzmann framework. *Computers & Mathematics with Applications*, 59, 2200-2214.

- [69] Wood, S. L., & Deiterding, R. (2015). A Lattice Boltzmann Method for Horizontal Axis Wind Turbine Simulation. In 14th International Conference on Wind Engineering - Porto Alegre, Brazil.
- [70] Wu, K., Wei, W., & Yan, Y. (2023). Lattice Boltzmann modeling of fluid-structure interaction of a flexible airfoil in a transitional flow. *International Journal of Heat and Mass Transfer*, 176, 121292.
- [71] Zhang, L., Luo, K. H., & Lu, X. (2023). Lattice Boltzmann simulation of convective heat transfer enhancement with microencapsulated phase change material in thermal energy storage systems. *International Journal of Heat and Mass Transfer*, 172, 121104.
- [72] Zhang, Q., Li, X., & Wang, J. (2023). Simulation of complex flow using lattice Boltzmann method with irregular geometries. *Computers & Fluids*, 224, 105238.
- [73] Zhuo, C. S., & Zhong, C. W. (2013). LES-based filter-matrix lattice Boltzmann model for simulating turbulent natural convection in a square cavity. *International Journal of Heat and Fluid Flow*, 42, 10-22.