



**Politecnico
di Torino**

Politecnico di Torino

MSc in Environmental and Land Engineering

Academic Year 2025/2026

RANS Simulation of Turbulent Flow Over Inclined Canopies of Solar Panel Array

Supervisor:

Prof. Costantino Manes

Co-supervisors:

Mohammad Karimipour

Elia Buono

Candidate:

Amirbahador Kazemimiarkolaei

Acknowledgements

I express my sincere gratitude to the Politecnico di Torino for providing me with the opportunity to pursue my Master's degree. I feel truly fortunate to have been part of such a stimulating academic environment and especially to have carried out my studies within the DIATI Department.

I would like to extend my deepest appreciation to my supervisor, Professor Costantino Manes. From the very beginning of my studies, I was inspired by his course in Environmental Fluid Mechanics, which strongly influenced my decision to pursue my thesis under his supervision. His guidance throughout this work has been invaluable, not only from a scientific and professional perspective, but also through his supportive and thoughtful approach.

I am also sincerely grateful to my co-supervisor, Mohammad Karimipour, for his continued support, patience, and insightful advice during the development of this work. Close collaboration and constructive feedback were essential throughout the process. I also thank another co-supervisor, Elia Buono, for his collaboration and for providing the experimental data that formed a fundamental part of this study.

My appreciation also goes to all the professors from whom I had the privilege to learn during my academic journey. Their dedication and teaching have contributed significantly to my growth.

Finally, and most importantly, I would like to express my deepest gratitude to my parents. Although they are not physically with me, their love, support, and encouragement have always been my greatest source of strength and motivation. Everything I have achieved is rooted in what they have given me.

Abstract

Ground-mounted solar panel canopies are increasingly being deployed in agricultural and open-field environments, where their interaction with environmental flows governs local momentum redistribution and near-surface dynamics. Accurate prediction of these effects remains challenging, particularly in the presence of surface roughness, geometric blockage, and flow separation. This study critically assesses the performance of a two-equation Reynolds-Averaged Navier-Stokes (RANS) approach, focusing on the k - ω Shear Stress Transport (SST) model for turbulent flow over inclined solar panel configurations.

Two-dimensional simulations are conducted within an open-channel framework and systematically compared with experimental measurements at multiple inclination angles. The evaluation focuses on velocity profiles, wake structures, and integrated mass flux to examine both global consistency and local predictive accuracy.

Although the model preserves the imposed mass flow rate and reproduces global flow trends, it fails to capture the correct spatial redistribution of momentum. For positive inclinations, velocities beneath the panel are consistently over-predicted, whereas for negative inclinations, most notably at -20° , the model misrepresents flow redirection, leading to substantial deviations of up to 26.9%. These discrepancies expose a fundamental limitation of eddy-viscosity closures; the inability of the Boussinesq hypothesis to represent anisotropic and non-equilibrium turbulence in separated and curved flows. The findings demonstrate that, despite its robustness, the k - ω SST model is inherently limited for canopy-induced flow configurations, which motivates the need for more advanced turbulence modeling approaches.

Table of Contents

Abstract	iv
List of Tables	viii
List of Figures	ix
1 Introduction	1
2 Literature Review	4
2.1 Fundamental Flow Physics	4
2.2 Flow Over Solar Panel Canopies and Arrays	8
2.2.1 Experimental Investigations of Solar Panel Arrays	8
2.2.2 Numerical Simulations of Isolated Panels	9
2.2.3 Numerical Simulations of Panel Arrays	11
3 Methodology	14
3.1 governing equations	14
3.2 k-omega SST Turbulence Model	16
3.2.1 Standard k-omega Formulation	16
3.2.2 Characteristics of the Standard k-omega Model	17
3.2.3 $k-\omega$ SST Model	17
3.2.4 Suitability for the present study	19
3.3 Computational Domain and Boundary Conditions	20
3.3.1 Open-Channel Flow Description	20
3.3.2 Streamwise Periodic Boundary Conditions	22
3.3.3 Driving Flow and Turbulence Specification	23
3.3.4 Wall and Free-Surface Boundary Conditions	24
3.3.5 Two-Dimensional Approach	25

3.4	Mesh Design	26
3.4.1	Mesh Topology and Refinement Strategy	26
3.4.2	Near-Wall Resolution Strategy	28
3.4.3	Grid Independence Study and Mesh Quality	29
3.5	Numerical Discretization and Solver Settings	30
3.5.1	Solver Formulation	30
3.5.2	Spatial Discretization	32
3.5.3	Convergence Criteria	33
3.6	Simulation Cases and Operating Conditions	33
4	Results	35
4.1	Numerical Convergence and Y^+ Values	35
4.2	Comparison of Numerical and Experimental Results	38
4.2.1	Influence of Inclination Angle on Mean Wake Topology	39
4.2.2	Streamwise Velocity (U_x) Profiles	44
4.2.3	Velocity Field Analysis: Magnitude Contours and Vector Representation	51
5	Conclusion	54
	Bibliography	56

List of Tables

3.1	Specifications and characteristic dimensions of the computational domain shown in Figure 3.2	21
3.2	analysis of aerodynamic forces to mesh refinement	29
3.3	Mesh quality	30
3.4	Summary of the numerical schemes and discretization methods. . .	32
3.5	Convergence criteria and monitoring parameters for the steady-state simulations.	33
3.6	Summary of the simulation cases and computational domain dimensions.	34
4.1	Quantitative comparison of the integrated mass flux between the numerical predictions and experimental data. Values represent the projected mass flow rate across a 1 m spanwise width. The percentage difference is calculated relative to the experimental value.	48

List of Figures

2.1	Kinematical studies of flows around obstacles [14]	5
2.2	Conceptual illustration of flow separation and wake development behind an inclined surface [14]	7
2.3	Sectional flow streamlines superimposed with the vorticity contours [26]	10
2.4	The surface pressure coefficient distribution of the rear-row photo- voltaic panels at a wind direction angle of 0° : (a) upper surface; (b) lower surface [5]	11
2.5	Sectional streamlines superimposed on static pressure contours (Pa), depicting the aerodynamic flow field through the canopy for normal- ized panel spacings of (a) $S^*/\Delta = 1$ and (b) $S^*/\Delta = 2$ [26].	12
2.6	Experimental arrangement of multiple tilted flat plates in tandem [24]	13
3.1	Flow resolved by DNS, LES, and RANS [4]	16
3.2	Geometric configuration and design parameters of the inclined solar panel array within the simulated channel	20
3.3	Cumulative grain size distribution of the bed material	22
3.4	Periodic Boundaries of domain	23
3.5	No-slip wall boundaries of the domain	24
3.6	Resolved geometric representation of the channel bed roughness elements	25
3.7	Close-up of the structured mesh and refinement zones for the inclined solar panel unit cell	27
3.8	Close-up of the structured mesh and refinement zones at the resolved gravel layer	28
3.9	General solver configuration used in the simulations	30

3.10	Viscous model settings in ANSYS Fluent showing the implementation of the k - ω SST turbulence model and the associated model constants used in the simulations	31
4.1	Residuals for the $+35^\circ$ panel inclination	36
4.2	Distribution of y^+ along the bottom wall of the computational domain for the $+35^\circ$ inclination case	37
4.3	Distribution of y^+ along the surfaces of the panel for the $+35^\circ$ inclination case	37
4.4	Mean flow streamlines of simulation (left) and experimental results (right) for the $+35^\circ$ panel inclination	41
4.5	Mean flow streamlines of simulation (left) and experimental results (right) for the $+20^\circ$ panel inclination	41
4.6	Mean flow streamlines of simulation (left) and experimental results (right) for the -35° panel inclination	42
4.7	Mean flow streamlines of simulation (left) and experimental results (right) for the -20° panel inclination	43
4.8	Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of $+35^\circ$. .	45
4.9	Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of $+20^\circ$. .	46
4.10	Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of -35° . .	47
4.11	Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of -20° . .	47
4.12	Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of $+35^\circ$	51
4.13	Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of $+20^\circ$	52
4.14	Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of -35°	53
4.15	Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of -20°	53

List of Equations

3.1	Continuity Equation	14
3.2	Navier-Stokes Momentum Equation	14
3.3	Continuity Equation	15
3.4	RANS Momentum Equation	15
3.5	Transport Equation for Turbulent Kinetic Energy (k)	16
3.6	Transport Equation for Specific Dissipation Rate (ω)	16
3.7	Turbulent Viscosity	17
3.8	SST Eddy Viscosity	18
3.9	Mass Flow calculation	23
3.10	Friction velocity	27
3.11	Wall Shear Stress	27
3.12	Skin Friction	27
3.13	non-dimensional Roughness Height	29
3.14	Reynolds Number	34
4.1	Mass Flux	48
4.2	Reynolds Stress Closure under Boussinesq Hypothesis	49
4.3	Linear Eddy-Viscosity Relation for Reynolds Stress Anisotropy	49

Chapter 1

Introduction

Ground-mounted solar panel canopies are increasingly integrated into agricultural lands and open-field installations, where they coexist with natural terrain and environmental flows. In such settings, the panels are exposed to atmospheric wind, leading to complex interactions between the flow and the canopy structure. These interactions influence not only structural loading but also local flow redistribution, which directly affects near-wall velocity fields and shear stress distribution beneath and around the panels. Although aerodynamic loads are often a primary concern in engineering design, an equally important challenge lies in accurately predicting the underlying flow behavior, particularly in configurations where surface roughness, geometric blockage, and flow separation interact simultaneously. In this context, reliable modeling of the flow field becomes essential for understanding and optimizing real-world solar canopy systems [1].

Despite increasing use of numerical simulations, significant challenges remain in capturing the coupled effects of turbulence, geometric complexity, and surface roughness. In realistic applications such as solar farms deployed on natural or agricultural terrain, the ground surface is inherently rough and heterogeneous, strongly influencing near-wall momentum transport. When combined with panel-induced separation and wake interaction, the flow develops strong velocity gradients and non-equilibrium turbulence structures. These features are not fully captured by conventional Reynolds-averaged approaches, particularly those relying on simplified turbulence closures. This motivates a detailed comparison between numerical predictions and experimental observations under controlled conditions.

From a fluid mechanics perspective, flow over inclined panels is characterized by

strong adverse pressure gradients, streamline curvature, and recirculation regions. These features induce non-uniform momentum redistribution and place significant demands on turbulence models. Experimental investigations under controlled conditions, such as open-channel configurations, provide valuable reference data to understand these mechanisms and assess numerical predictions. Such configurations enable detailed observation of flow separation, reattachment, and wake development while maintaining well-defined boundary conditions [2].

Computational Fluid Dynamics (CFD) has become a standard tool for analyzing turbulent flows in engineering applications, with Reynolds-averaged Navier-Stokes (RANS) models offering a practical balance between computational cost and predictive capability. Among these, eddy-viscosity models such as formulations k - ε and k - ω are widely used. These models are based on the Boussinesq hypothesis, which assumes a linear relationship between Reynolds stresses and the mean strain rate through an isotropic turbulent viscosity [3]. Although this assumption simplifies the closure problem, it imposes fundamental limitations in flows where turbulence exhibits strong anisotropy, history dependence, and misalignment between stress and strain conditions commonly encountered in separated and swirling flows [4].

Among these models, the k - ω Shear Stress Transport (SST) model has been developed to improve predictions in wall-bounded and separated flows by combining the near-wall accuracy of the formulation k - ω with the free-stream robustness of the model k - ε . It has demonstrated improved performance in flows with adverse pressure gradients and near-wall effects. However, despite these advantages, the model remains fundamentally constrained by the eddy-viscosity assumption. Accordingly, a primary objective of the present study is to critically evaluate the performance of the k - ω SST model in predicting flow behavior over inclined solar panel configurations, particularly under conditions involving strong separation and curvature.

In addition to turbulence modeling, surface roughness representation plays a critical role in accurately capturing near-wall flow behavior. Conventional approaches often rely on equivalent sand-grain roughness models, which may not adequately represent complex geometries. In the present work, the gravel bed is explicitly resolved as geometric elements within the computational domain, allowing the direct simulation of roughness-induced flow features. This approach provides a more physically consistent framework for examining the interaction between surface roughness and separated flow structures.

In the context of solar panel aerodynamics, previous studies have focused mainly on isolated panels or simplified configurations, providing fundamental insights into separation mechanisms and pressure distributions [1]. However, comparatively fewer studies have addressed the detailed flow behavior within panel arrays or canopy-like configurations, where wake interactions between successive panels significantly alter the inflow conditions and turbulence structure. These interactions introduce additional complexity through flow blockage, momentum redistribution, and enhanced turbulence intensity, making accurate prediction more challenging. Although numerical simulations have been widely used to investigate such systems, discrepancies between computational and experimental results persist, particularly in regions of strong flow deflection and near-wake development [5].

These challenges motivate a detailed comparison between numerical predictions and high-resolution experimental data, with particular emphasis on configurations that progressively increase flow complexity. Rather than treating the panel inclination as an isolated parameter, it is used as a controlled means to modify the severity of flow separation, streamline curvature, and momentum redistribution within the domain. This approach enables a more rigorous assessment of the model's predictive behavior under increasingly demanding flow conditions.

The present study addresses this need through a combined computational and experimental analysis of turbulent flow over inclined solar panel configurations under open-channel conditions. Four panel inclinations ($+20^\circ$, $+35^\circ$, -20° , and -35°) are considered, allowing systematic variation of flow deflection and aerodynamic blockage. Numerical simulations are performed using a two-dimensional RANS framework with the $k-\omega$ SST turbulence model, while experimental PIV data serve as a reference for validation.

Chapter 2

Literature Review

2.1 Fundamental Flow Physics of Turbulent Flow Over Surface-Mounted and Inclined Bluff Bodies

The study of turbulent flow over surface-mounted bluff bodies represents a cornerstone of fluid mechanics with wide implications across diverse engineering disciplines [6]. These flows are characterized by the complex interaction between wall-bounded turbulence and the specific geometric constraints of the obstacle [7]. As the incoming boundary layer encounters a non streamlined body, it is subjected to intense adverse pressure gradients that fundamentally alter the local velocity field [6, 8]. These gradients inevitably lead to large-scale flow separation and the subsequent development of a highly turbulent wake [7, 9]. Understanding the dynamics of these separated regions is essential for predicting pressure distributions, force coefficients, and wake recovery mechanisms in wall-bounded turbulent flows [10]. Consequently, establishing a rigorous physical baseline for these mechanisms is essential for the advancement of both experimental measurements and numerical validations.

A fundamental distinction of separation behind wall-mounted obstacles is its inherently three-dimensional nature, which defies traditional two-dimensional intuition [11]. Unlike simplified planar flows, the mean streamlines and shear-stress patterns in these configurations are governed by a complex topology that involves specific singular points [11, 12], including nodes and saddles, as illustrated in Figure 2.1. These topological features require that the number of nodes must be

equal to the number of saddles across the obstacle and the surface of the ground [11]. The global structure of the flow is constrained by these topological features, such that the total number of nodes and saddles must satisfy specific kinematic balances on the combined surface of the obstacle and the plane [11]. Contrary to earlier assumptions of closed separation "bubbles," high-fidelity visualization indicates that these regions are generally open, lacking a closed surface of mean streamlines [13]. This kinematic reality ensures that the fluid is continuously exchanged between the wake and the surrounding stream, fundamentally altering the transport and mixing mechanisms within the near-field flow [13].

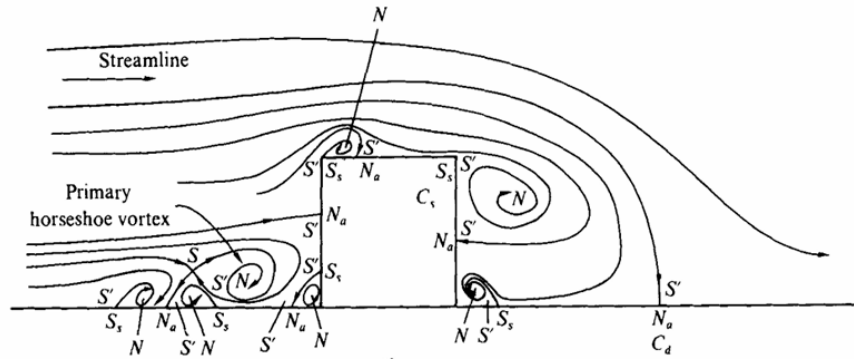


Figure 2.1: Kinematical studies of flows around obstacles [14]

At the base of a surface-mounted obstacle, the interaction between the incoming boundary layer and the stagnation pressure on the windward face creates a dominant vortical structure known as the horseshoe vortex [15, 14]. This system typically originates from the roll up of spanwise vorticity in the upstream boundary layer as it is deflected by the strong adverse pressure gradient generated by the body [14]. The resulting vortex tubes wrap around the end of the obstacle and extend far downstream, significantly influencing the momentum redistribution within the near-wall region [15]. In complex cases, multiple vortices can form at this junction, their number and stability being sensitive to both incident flow parameters and the geometric sharpness of the obstacle's [7, 15]. This intricate interaction between the wall boundary layer and obstacle-induced vortices plays a significant role in shaping the topology of the separation line and subsequent wake development. As a persistent and robust physical feature, this vortex system remains a primary driver of the complex aerodynamic behavior observed around surface structures [15].

Following separation at the leading edge of a surface-mounted bluff body, the flow develops a detached shear layer that originates at the point of separation and

extends downstream along the body and into the wake [8, 13]. This shear layer encloses a recirculation region characterized by reversed mean flow and reduced momentum relative to the incoming boundary layer [13]. Within this region, the mean pressure remains significantly lower than that of the surrounding flow, contributing to the overall aerodynamic loading on the obstacle. As the shear layer evolves downstream, it entrains high-momentum fluid from the outer flow, progressively weakening the recirculation zone. The location in which the mean flow reattaches to the downstream surface defines the length of reattachment, a key parameter commonly used to characterize the development of wake behind obstacles on the surface [6, 8]. The size and persistence of this recirculation region play a central role in the regulation of wake recovery, pressure distribution, and the downstream influence of the obstacle [8].

The structure and evolution of the separated flow downstream of obstacles mounted on the surface are strongly influenced by the characteristics of the incoming boundary layer, particularly its intensity of turbulence and its integral length scales [7, 10]. Variations in upstream turbulence can significantly modify the onset of separation, the growth of the separated shear layers, and the rate of entrainment into the wake [7, 14]. Elevated turbulence levels generally enhance momentum exchange throughout the shear layer, promoting increased mixing and, in many cases, earlier recovery of the mean flow [10, 16]. In contrast, low-turbulence inflow conditions tend to maintain larger recirculation zones with stronger velocity deficits and reduced entrainment [7]. These observations indicate that wake behavior cannot be interpreted solely as a function of obstacle geometry, but must be understood in conjunction with the upstream flow environment.

The Reynolds number further governs the balance between inertial and viscous mechanisms within separated regions, influencing both the coherence and the breakdown of vortical structures in the wake. As the Reynolds number increases, the separated shear layers become more unstable, leading to an earlier transition to turbulence and a progressively more fragmented wake topology [17]. Although the qualitative features of separation and wake formation remain similar across a wide range of Reynolds numbers, quantitative metrics such as the length of reattachment, the intensity distributions of turbulence, and the recovery of pressure exhibit a strong dependence on Reynolds number [18, 19]. This sensitivity complicates direct comparisons between experimental and numerical studies and highlights the difficulty of achieving dynamic similarity in wall-mounted bluff-body flows.

When the obstacle is inclined relative to the incoming flow, as in the case of plates mounted on the surface or configurations similar to a canopy, the separation process and the wake structure undergo substantial modifications compared to vertical bluff bodies, as conceptually illustrated in Figure 2.2 [20]. The inclination introduces an asymmetric pressure distribution between the windward and leeward surfaces, altering both the location of the flow separation and the trajectory of the separated shear layers. For moderate inclination angles, partial flow attachment may persist along portions of the upper surface, while separation is delayed or weak relative to normal-incidence configurations [20]. However, at higher angles of inclination, however, the flow typically experiences strong adverse pressure gradients near the leading edge, resulting in pronounced separation and the formation of an extended recirculation region downstream of the plate [21]. This sensitivity to the angle of inclination leads to significant variability in wake size, intensity of turbulence, and mean velocity recovery, emphasizing that inclined canopies represent a transitional class of flows that bridge attached boundary-layer behavior and fully separated bluff-body wakes.

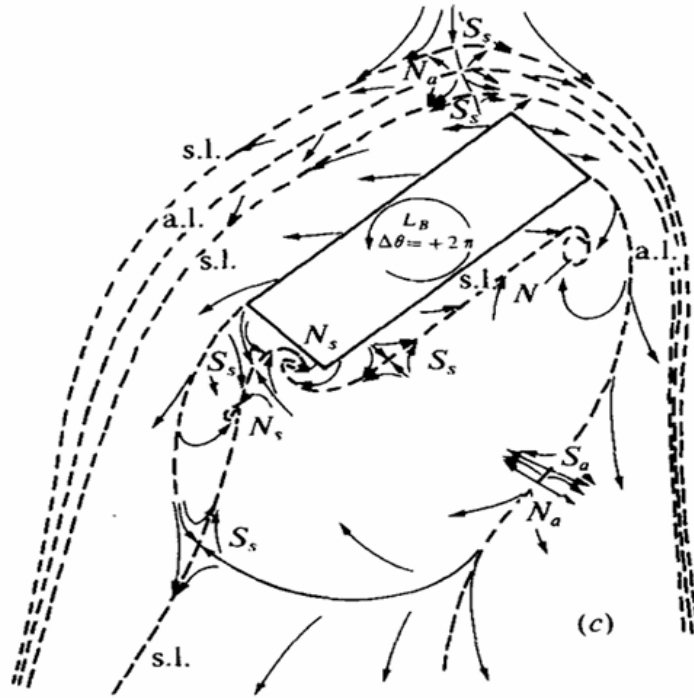


Figure 2.2: Conceptual illustration of flow separation and wake development behind an inclined surface [14]

2.2 Flow Over Solar Panel Canopies and Arrays

2.2.1 Experimental Investigations of Solar Panel Arrays

Experimental investigations play a crucial role in understanding the aerodynamic behavior of solar panels exposed to atmospheric flows from the boundary layer [22]. Due to the strong interaction between inclined surfaces, wall-bounded turbulence, and separated wakes, experimental measurements remain essential for capturing flow features that are difficult to model reliably. However, experimental studies on solar panels face inherent challenges related to geometric scaling, Reynolds number similarity, and the reproduction of realistic boundary layer profiles [22]. Wind tunnel experiments often require compromises between model size, blockage effects, and measurement resolution, which directly influence the accuracy of measured loads and flow structures [22]. Despite these limitations, controlled experiments provide indispensable information on pressure distributions, wake development, and load fluctuations on solar panels, forming a critical foundation for subsequent numerical analysis and validation [12].

Early experimental studies on isolated solar panels and inclined flat plates have provided fundamental insight into the aerodynamic response of tilted surfaces exposed to flows from the boundary layer [12, 23]. Wind tunnel measurements on independent panels demonstrate that the flow separates at the leading edge for moderate to high tilt angles, forming a shear layer that governs wake development and pressure distribution on the panel surface [12, 24]. The degree of separation, location reattachment, and wake length are strongly dependent on panel inclination, incoming turbulence intensity, and boundary layer thickness [12, 24]. Pressure measurements reveal highly non-uniform load distributions, with peak suction occurring near the leading edge and reduced pressures in the separated region downstream [23]. These isolated panel experiments establish a baseline understanding of separation-driven loading mechanisms, which is essential before considering the additional complexity introduced by panel–panel interactions in array configurations.

Experimental investigations of solar panel arrays and canopy configurations highlight the strong influence of panel–panel interactions on the flow field and the resulting aerodynamic loads [25]. Wind tunnel studies on multi-row panel arrangements show that upstream panels generate separated wakes that significantly modify the inflow conditions experienced by downstream panels [25]. This

wake interference leads to reduced mean wind loads on sheltered panels, but can also increase unsteady loading due to enhanced turbulence intensity within the canopy. The extent of wake interaction depends on parameters such as panel spacing, tilt angle, array density, and incoming boundary layer characteristics [25]. These experiments demonstrate that flow behavior within panel arrays cannot be extrapolated directly from isolated-panel results, underscoring the need to explicitly account for array effects when analyzing canopy scale flow dynamics.

Despite the growing body of experimental work on wind flow around solar panels, significant limitations remain in the availability and applicability of experimental data for detailed flow analysis [23, 22]. Many studies focus primarily on surface pressure measurements and global load coefficients, providing limited information on the underlying flow structures within the wake region of the panel and near-wall regions [23, 12]. High-resolution velocity measurements inside panel canopies are comparatively scarce, particularly for configurations involving dense assemblages, rough ground conditions, or complex inflow turbulence representative of atmospheric boundary layers [25]. Additionally, experimental constraints often require reduced Reynolds numbers, simplified geometries, or truncated arrays, which can obscure scale-dependent effects and limit direct extrapolation to full-scale installations [22]. These gaps highlight the need for complementary numerical investigations capable of resolving detailed flow physics while maintaining geometric and dynamic similarity with realistic solar panel configurations [26].

2.2.2 Numerical Simulations of Isolated Panels

Numerical simulations have been extensively employed to investigate wind flow in isolated solar panels, where geometric simplicity allows a detailed examination of flow separation and wake formation [26]. In this context, CFD studies aim to resolve the interaction between the incoming boundary layer and the inclined panel surface, providing information on pressure distribution, aerodynamic loading, and near-wake structure [12, 26]. By isolating a single panel, these investigations establish a fundamental reference for understanding more complex interactions in panel arrays and canopy configurations.

CFD studies consistently demonstrate that the tilt angle of the panel plays a dominant role in determining the extent and intensity of the flow separation as illustrated by the sectional streamline and vorticity distributions shown in Figure 2.3. As the inclination increases, stronger adverse pressure gradients develop

on the leeward side of the panel, leading to earlier separation and a thicker wake region [24]. The numerical results typically show an enlargement of the recirculation zone and increased instability in the separated shear layer with increasing tilt angle. These effects directly influence both pressure distributions and aerodynamic loads, highlighting the tilt angle as a primary design parameter in wind-resilient panel installations [26].

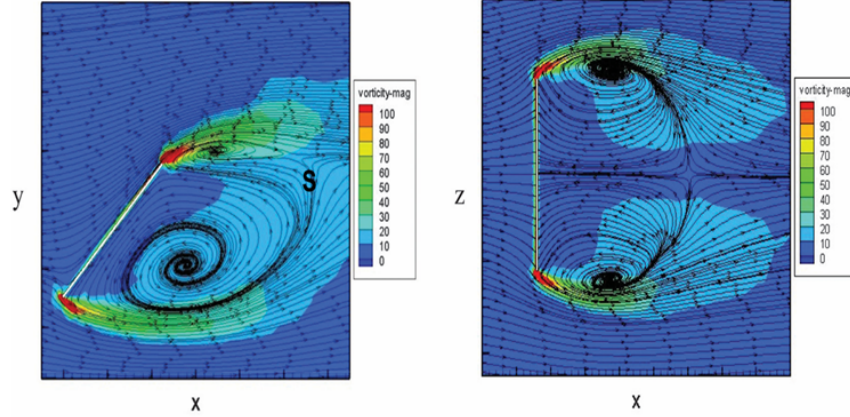


Figure 2.3: Sectional flow streamlines superimposed with the vorticity contours [26]

Extending these analyzes to photovoltaic systems,[5] performed CFD simulations on fixed double-row Photovoltaic arrays to identify critical wind direction angles and edge-module vulnerability. Their study revealed that specific wind approach angles amplify pressure coefficients on edge panels due to flow acceleration and vortex formation along the array boundaries, as illustrated by the pressure coefficient distributions shown in Figure 2.4, emphasizing the sensitivity of panel loading to directional effects.

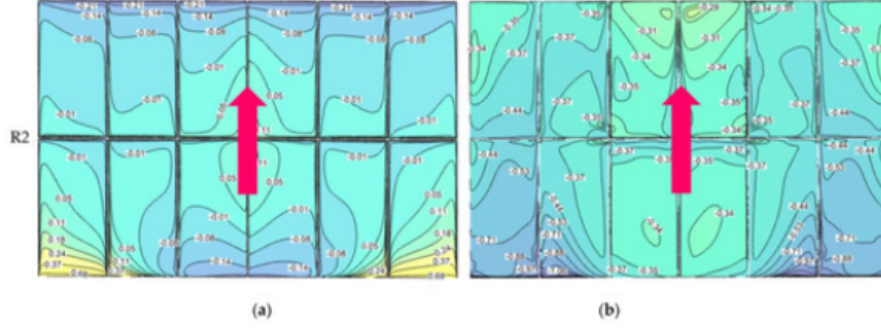


Figure 2.4: The surface pressure coefficient distribution of the rear-row photovoltaic panels at a wind direction angle of 0° : (a) upper surface; (b) lower surface [5]

Although isolated-panel simulations provide valuable insight into local flow physics, they inherently neglect wake interference effects present in practical installations. The numerical results indicate that the wake downstream of a single panel can extend several panel heights, suggesting strong sensitivity to downstream boundary conditions and domain size [26]. Therefore, conclusions drawn from isolated configurations may not be directly transferable to panel arrays, where upstream wakes interact with subsequent panels and modify the local inflow conditions. This limitation motivates further numerical investigations of repeated configurations on a canopy-scale.

2.2.3 Numerical Simulations of Panel Arrays

Numerical investigations of solar panel arrays have been conducted to capture flow interactions that are absent in isolated-panel configurations [26, 27]. In array arrangements, the wake generated by an upstream panel directly alters the inflow conditions experienced by downstream panels, leading to modified separation behavior, pressure distributions, and aerodynamic loads [26]. CFD studies have shown that these interactions can significantly amplify or attenuate wind-induced forces, depending on panel spacing, alignment, and tilt angle [27]. As a result, array scale simulations are essential for a realistic assessment of wind effects in utility-scale photovoltaic installations.

At the canopy scale, numerical studies indicate that tightly spaced panels collectively act as a roughness element, altering the mean velocity profile and the turbulence structure of the approaching flow [27]. The formation of an internal

boundary layer above the canopy and the development of recirculation zones within the panel rows have been consistently observed (Figure 2.5)[26]. These canopy-induced modifications lead to reduced mean velocities within the array but increased intensity and turbulence shear near the edges of the panel [27]. Such effects are highly sensitive to the spacing and inclination of the panels, which underscores the complexity of flow behavior within photovoltaic canopies.

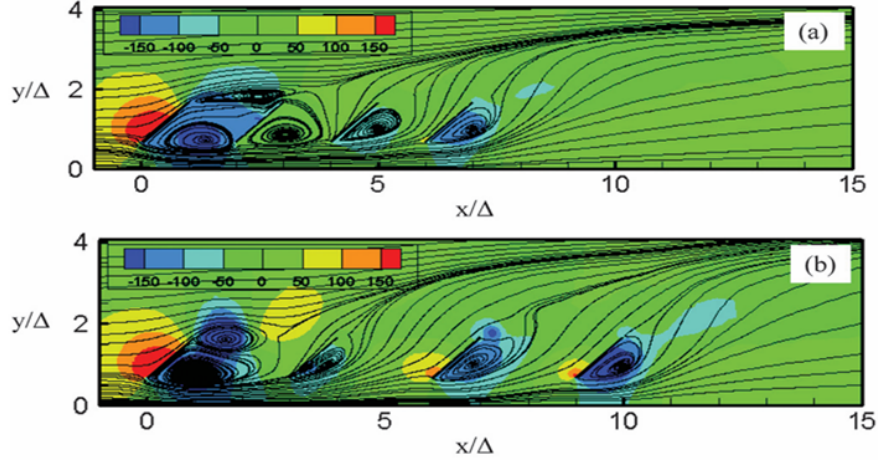


Figure 2.5: Sectional streamlines superimposed on static pressure contours (Pa), depicting the aerodynamic flow field through the canopy for normalized panel spacings of (a) $S^*/\Delta = 1$ and (b) $S^*/\Delta = 2$ [26].

To effectively model large or infinite panel arrays, several numerical studies employ periodic boundary conditions in spanwise or streamwise directions [26]. This approach enables the representation of repeated panel units while maintaining a manageable computational cost, similar to the array configurations illustrated in Figure 2.6. Periodic simulations are particularly suited for studying statistically stationary flow within the interior of large arrays, where edge effects are negligible. However, the precision of such methods depends on the appropriate height of the domain, the specification of the input turbulence, and the treatment of near-wall roughness, all of which influence wake recovery and canopy scale flow development [28].

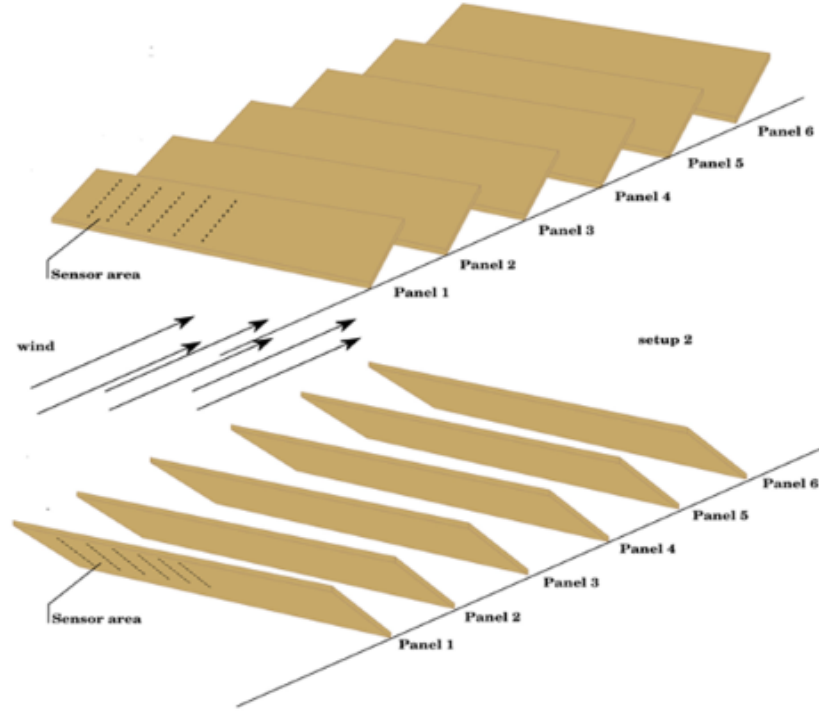


Figure 2.6: Experimental arrangement of multiple tilted flat plates in tandem [24]

The experimental and numerical studies reviewed in this section highlight the complex flow phenomena associated with inclined solar panels and panel arrays, including flow separation, wake interaction, and strong sensitivity to geometric and inflow parameters. Although numerical simulations are widely used to investigate these effects, their predictive ability is fundamentally constrained by the assumptions underlying the turbulence model [28]. This motivates a focused discussion on commonly adopted modeling approaches and their limitations, which is presented in the following section.

Chapter 3

Methodology

3.1 Governing Equations and Turbulence Modeling

The flow over the inclined solar panel canopy is modeled as an incompressible, turbulent flow governed by the Navier–Stokes equations. The fluid is assumed to be Newtonian with constant density and viscosity. Given the low Mach number conditions associated with the present configuration, the incompressible flow assumption is considered valid. Under these assumptions, the governing equations for mass and momentum conservation are expressed as follows:

$$\nabla \cdot \mathbf{u} = 0 \quad (3.1)$$

$$\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \mathbf{u} = -\frac{1}{\rho} \nabla p + \nu \nabla^2 \mathbf{u} \quad (3.2)$$

where $\mathbf{u}$ is the velocity vector, p the pressure, ρ the fluid density, and ν the kinematic viscosity.

For flows with high Reynolds numbers, resolving all turbulent scales through Direct Numerical Simulation (DNS) is computationally prohibitive for engineering-scale problems. Therefore, a Reynolds-averaged formulation is adopted. Within the Reynolds-averaged Navier-Stokes (RANS) framework, instantaneous flow variables are decomposed into mean and fluctuating components. By substituting this decomposition into the Navier–Stokes equations and applying a time-averaging operation, the governing RANS equations are obtained;

$$\frac{\partial \overline{u_i}}{\partial x_i} = 0 \quad (3.3)$$

$$\frac{\partial \overline{u_i}}{\partial t} + \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = -\frac{1}{\rho} \frac{\partial \overline{p}}{\partial x_i} + \nu \frac{\partial^2 \overline{u_i}}{\partial x_j \partial x_j} - \frac{\partial}{\partial x_j} (\overline{u'_i u'_j}) \quad (3.4)$$

The additional term $-\overline{u'_i u'_j}$, known as the Reynolds stress tensor, represents the momentum transport induced by turbulent velocity fluctuations. These stresses arise from correlations of velocity fluctuations and introduce additional unknowns into the system, leading to the well-known closure problem.

The time-averaging operation is performed over a time interval that is large compared to the characteristic timescales of turbulence but small relative to the timescale of the mean flow. As a result, the RANS equations describe only the evolution of the mean flow, while the effects of turbulence are entirely represented through the Reynolds stresses.

A widely used approach to model these stresses is the Boussinesq approximation, which assumes a linear relationship between Reynolds stresses and the mean strain rate. In this framework, turbulence is treated as a diffusive process analogous to molecular momentum transport, and Reynolds stresses are modeled using turbulent eddy viscosity ν_T . The flow can therefore be interpreted as a pseudo-fluid with an effective viscosity $\nu_{\text{eff}} = \nu + \nu_T$, which explains the enhanced turbulent mixing.

Although this approximation assumes isotropic turbulence and local equilibrium between stress and strain-conditions that are not always satisfied in flows involving separation, curvature, and recirculation, it remains widely used in engineering applications due to its robustness and computational efficiency.

Since turbulent viscosity is not a fluid property but depends on the local turbulence state, additional modeling is required to determine it. The Eddy-viscosity turbulence models express ν_T in terms of characteristic turbulent velocity and length scales, obtained through algebraic relations or additional transport equations. Based on the number of these equations, the turbulence models are classified as zero-, one-, and two-equation models. Among these, two-equation models are most commonly used in engineering practice, as they provide a practical balance between accuracy and computational cost by independently determining the relevant turbulence scales.

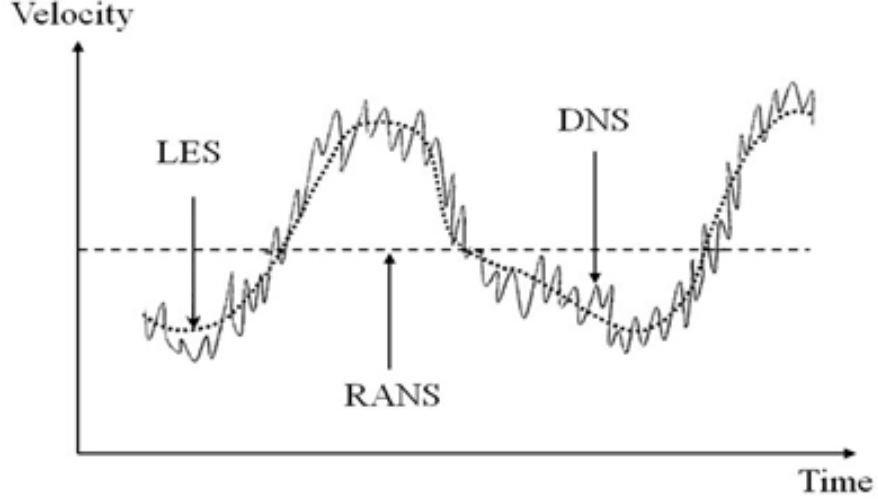


Figure 3.1: Flow resolved by DNS, LES, and RANS [4]

3.2 $k - \omega$ SST Turbulence Model

3.2.1 Standard $k - \omega$ Formulation

The model $k-\omega$ is a two-equation eddy-viscosity turbulence model in which the transport equations are solved for the turbulent kinetic energy k and the specific dissipation rate ω . Although there are several two-equation models, the formulation $k-\omega$ distinguishes itself by using ω as the second variable, which represents the inverse of the turbulent dissipation time scale. Since $\omega \sim \varepsilon/k$, it can be interpreted as a measure of the rate at which turbulent kinetic energy is dissipated. The modeled transport equations for the standard $k-\omega$ model can be written as:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = P_k - \beta^* k \omega + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \quad (3.5)$$

$$\frac{\partial \omega}{\partial t} + U_j \frac{\partial \omega}{\partial x_j} = \alpha \frac{\omega}{k} P_k - \beta \omega^2 + \frac{\partial}{\partial x_j} \left[\left(\nu + \frac{\nu_T}{\sigma_\omega} \right) \frac{\partial \omega}{\partial x_j} \right] \quad (3.6)$$

where P_k denotes the production of turbulent kinetic energy.

The turbulent viscosity is calculated from the following:

$$\nu_T = \frac{k}{\omega} \quad (3.7)$$

Thus, the model determines the turbulent velocity scale through $k^{1/2}$ and the turbulence time scale through $1/\omega$, enabling the computation of the eddy viscosity required to close the RANS equations.

3.2.2 Characteristics of the Standard $k - \omega$ Model

- **Near-wall behavior:**

The model $k-\omega$ exhibits several important advantages compared to the standard formulation $k-\varepsilon$. One of its main strengths lies in its near-wall behavior. The model performs well in the viscous sub-layer and is capable of accurately reproducing the logarithmic law of the wall without requiring empirical wall functions. Consequently, it is particularly suitable for wall-bounded flows, provided that a sufficient resolution of the mesh is ensured, typically with the first point in the grid located at $y^+ < 5$.

- **Behavior at low turbulence levels:**

In addition, the formulation $k-\omega$ remains well behaved in regions of low turbulence intensity. Unlike the $k-\varepsilon$ model, which may encounter numerical difficulties as k and ε approach zero due to terms involving ε^2/k , the $k-\omega$ model avoids such singular behavior. Furthermore, for flow in the boundary-layer subjected to streamwise adverse pressure gradients, the $k-\omega$ model has been shown to provide improved predictions compared to the standard $k-\varepsilon$ model, particularly in decelerating flows.

- **Adverse pressure gradients:**

Despite these advantages, the standard model $k-\omega$ has a known limitation: it is sensitive to the prescribed boundary condition in the free-stream for ω . As a result, the computed solutions may depend on the chosen inlet value of ω , which can reduce the robustness in external flow simulations. This limitation motivates the development of improved formulations, such as the $k-\omega$ SST model.

3.2.3 $k-\omega$ SST Model

The Shear Stress Transport model (SST) $k-\omega$, introduced by Menter (1994), was developed to combine the advantages of the standard $k-\omega$ and $k-\varepsilon$ turbulence models while reducing their respective shortcomings. As discussed previously, the standard model $k-\omega$ provides accurate predictions in the near-wall region and

performs well under adverse pressure gradients, but exhibits sensitivity to the prescribed free-stream boundary condition for ω . In contrast, the k - ε model is more robust in the outer flow and free-stream region, but requires wall functions or low-Reynolds-number modifications near solid boundaries. The SST formulation addresses these limitations through a systematic blending strategy.

- **Blending of k - ω and k - ε :**

The SST model is formulated within the k - ω framework, but its model coefficients and certain terms are blended between the formulations k - ω and k - ε using smooth blending functions. In the near-wall region, the blending function activates the standard k - ω model, thereby preserving its superior treatment of the viscous sub-layer and logarithmic boundary-layer region. Away from the wall, the model gradually transitions toward a transformed formulation k - ε , which improves robustness in the free-stream and reduces sensitivity to inlet boundary conditions for ω .

This blending ensures that the model behaves as a k - ω model in the inner region of the boundary layer and as a k - ε model in the outer region, thus combining the strengths of both formulations within a single consistent framework.

- **Shear Stress Limiter:**

A key distinguishing feature of the SST model is the introduction of a shear-stress limiter in the definition of the turbulent eddy viscosity. While the standard k - ω model computes the turbulent viscosity as demonstrated in (3.7), the SST formulation modifies this relation to limit the excessive growth of turbulent viscosity in regions of strong shear. The eddy viscosity is computed as follows:

$$\nu_T = \frac{a_1 k}{\max(a_1 \omega, S F_2)}, \quad (3.8)$$

where S denotes the magnitude of the mean strain-rate tensor and F_2 is a blending function. This limiter prevents over-prediction of turbulent shear stress that can occur in adverse pressure gradient flows, thereby improving the prediction of flow separation and boundary-layer behavior in decelerating regions.

- **Cross-Diffusion Term:**

An additional modification introduced in the SST model is the inclusion of a cross-diffusion term in the ' ω ' transport equation. This term arises from the

transformation between the k - ε and k - ω formulations and becomes active in the outer region of the boundary layer. The cross-diffusion term contributes to a smoother transition between model behaviors and improves numerical stability while preserving near-wall accuracy. Through the combined action of blending functions, the shear-stress limiter and the cross-diffusion term, the k - ω SST model achieves improved predictive capacity for wall-bounded flows involving adverse pressure gradients and separation.

3.2.4 suitability for the present study

The present study investigates turbulent flow over an inclined solar panel canopy in an open-channel configuration, where flow separation, adverse pressure gradients, and streamline curvature play a dominant role. Therefore, accurate prediction of boundary-layer behavior and separation is essential for reliable numerical results.

As discussed in the preceding sections, the k - ω SST model combines the near-wall accuracy of the standard k - ω model with the free-stream robustness of the formulation k - ε . Its ability to resolve the viscous sub-layer without relying on empirical wall functions makes it particularly suitable for simulations in which the near-wall region is directly resolved. Furthermore, the incorporation of a shear-stress limiter improves the prediction of flows with adverse pressure gradients and the ability of the model to capture separation phenomena.

Given that flow over an inclined solar panel involves boundary-layer development, deceleration along the panel surface, flow separation, and wake formation downstream, the k - ω SST model provides a practical balance between physical fidelity and computational efficiency. Compared to higher-fidelity approaches such as Large Eddy Simulation (LES), the SST model enables tractable simulations for engineering-scale domains while offering improved performance over standard two-equation models, particularly in flows with adverse pressure gradients and near-wall effects.

3.3 Computational Domain and Boundary Conditions

3.3.1 Open-Channel Flow Description

The numerical simulations replicate an open-channel laboratory experiment in which water flows horizontally over a gravel bed and interacts with a series of inclined panel elements arranged in tandem. The channel has a width of 610 mm, while the panels have a spanwise width of 605 mm, effectively covering the entire channel width and minimizing sidewall effects. The geometric configuration of the experimental arrangement and the principal design parameters are illustrated in Figure 3.2.

The channel is horizontal and the flow is driven in the streamwise direction. In the experimental configuration, depth of the water varies slightly along the streamwise direction due to the friction slope; however, a representative water depth of approximately 0.5 m can be considered a good approximation, as this value is imposed in a reference measurement section for each operating condition. This depth is used as the characteristic vertical scale of the computational domain.

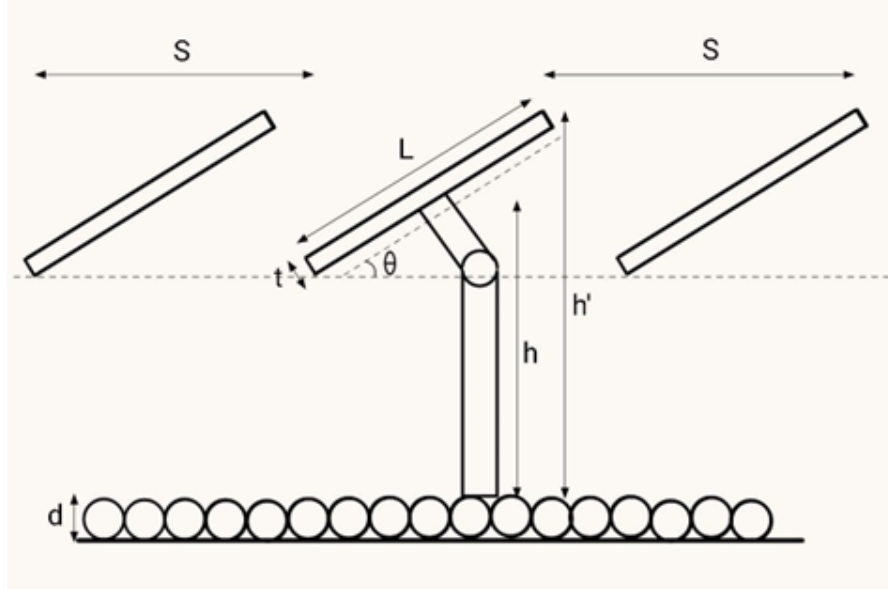


Figure 3.2: Geometric configuration and design parameters of the inclined solar panel array within the simulated channel

Each panel has a length of 100 mm and a thickness of 2 mm. Four configurations

are analyzed, corresponding to the inclination angles of 20° , 35° , -20° , and -35° . Because the panels are inclined differently, the vertical distance between the center of the panel and the channel bed changes with the angle of inclination. However, panels with the same absolute angle of inclination ($\pm 20^\circ$ or $\pm 35^\circ$) have the same vertical center position, since only the direction of the inclination is reversed. As a result, the four configurations correspond to two distinct vertical heights.

The panels are placed one after another in the streamwise direction, forming a tandem array. The main geometric parameters of the configurations are summarized in Table 3.1.

Table 3.1: Specifications and characteristic dimensions of the computational domain shown in Figure 3.2

Configuration	Panel Length(L)	Panel thickness(t)	Inclination Angle(θ)	Panel Spacing(S)	Panel height from gravel	
					h	h'
1	100 mm	2 mm	35°	100 mm	118 mm	-
2	100 mm	2 mm	20°	100 mm	-	147 mm
3	100 mm	2 mm	-35°	100 mm	118 mm	-
4	100 mm	2 mm	-20°	100 mm	-	147 mm

The channel bed consists of a gravel layer characterized by a median particle diameter $d_{50} = 6.66$ mm shown in Figure 3.3. In the numerical model, this roughness layer was explicitly resolved in the computational geometry.

In the physical experiment, the upper boundary corresponds to a free surface exposed to air. In the present simulations, a single-phase approach is adopted, and the free surface is approximated as a shear-free boundary condition (zero shear stress). The effects of Surface deformation are neglected, and the interface is assumed to remain flat. Since the focus of the study is on the interaction between the turbulent flow and the inclined panel, rather than on the free-surface dynamics, gravitational effects are not explicitly modeled, and their influence is implicitly neglected in the flow solution.

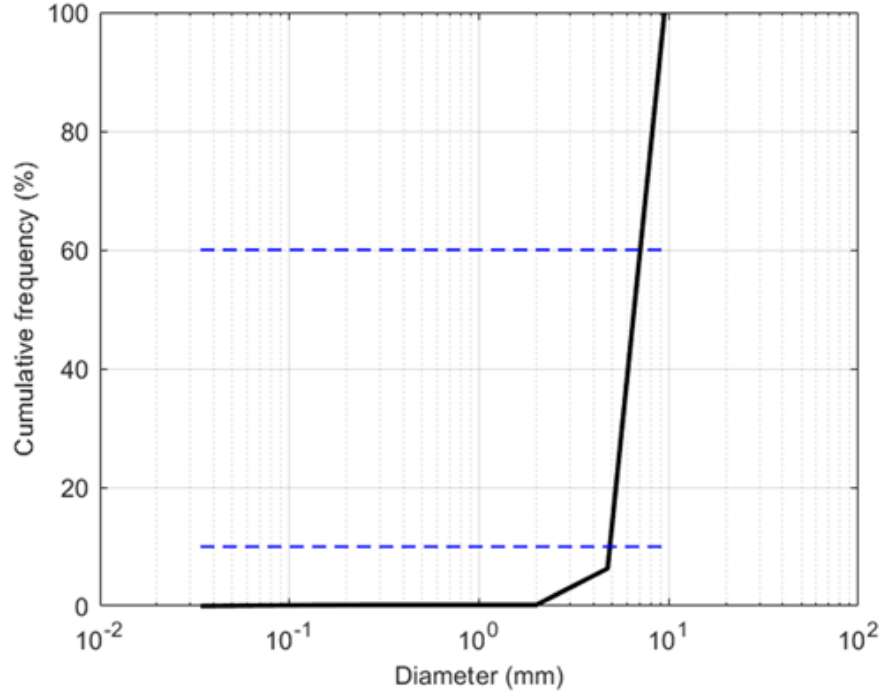


Figure 3.3: Cumulative grain size distribution of the bed material

3.3.2 Streamwise Periodic Boundary Conditions

To represent the tandem arrangement of panels while limiting computational cost, a single periodic unit cell is modeled in the streamwise direction. In the experimental configuration, identical panels are arranged sequentially with a uniform center-to-center spacing of 100 mm. Accordingly, the computational domain length is set equal to this spacing, so that the domain represents one repeating unit of the array. The panel is positioned centrally within the domain, with the left and right boundaries located midway between adjacent panels to ensure geometric continuity between periodic cells.

Periodic boundary conditions are imposed at the streamwise boundaries, enforcing equality of all flow variables between corresponding locations. This approach represents an infinitely repeating panel array and implies spatially periodic fully developed flow conditions such that the flow structure repeats from one panel spacing to the next. The flow is driven by prescribing a constant mass flow rate across the periodic boundaries, allowing the required streamwise pressure gradient to adjust automatically. Steady-state RANS simulations are performed under this periodic forcing. Non-conformal periodic boundary conditions were employed to

allow mesh flexibility across periodic interfaces.

The two-dimensional computational domain used for the simulations is illustrated in Figure 3.4.

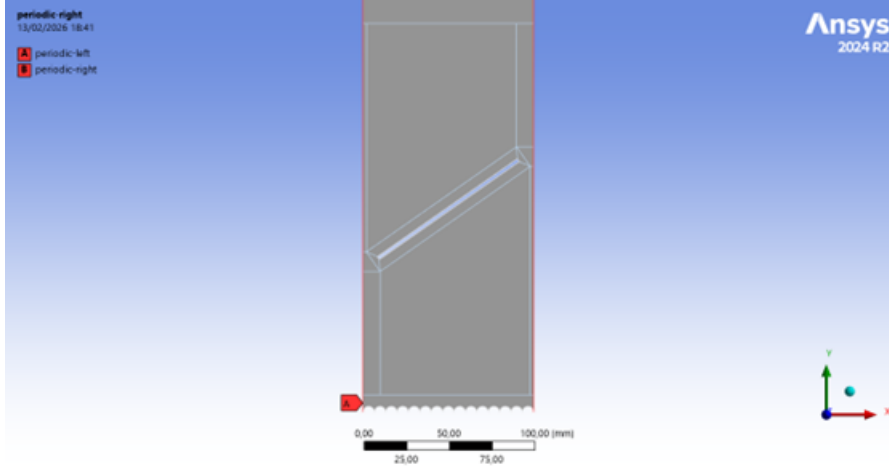


Figure 3.4: Periodic Boundaries of domain

3.3.3 Driving Flow and Turbulence Specification

The experimental setup imposed a volumetric discharge of 95 L/s in a channel with a spanwise length of 610 mm (0.61 m). The working fluid in both the experiment and the numerical model is water, with a density of $\rho = 998.2 \text{ kg/m}^3$. Since numerical simulations are performed in two dimensions, the spanwise direction is not explicitly resolved and the solver assumes a unit spanwise width of 1 m. Therefore, the experimental discharge was converted to an equivalent discharge per unit width before being imposed in the CFD model, ensuring that the same volumetric flux through the cross-section is preserved. The flow in the periodic computational domain was driven by prescribing a constant mass flow rate across the streamwise periodic boundaries.

$$\begin{aligned}
 Q_{exp} &= 95 \text{ L/s} = 0.095 \text{ m}^3/\text{s} \\
 Q_{2D} &= \frac{Q_{exp}}{0.61} = \frac{0.095}{0.61} \approx 0.156 \text{ m}^3/\text{s} \\
 \dot{m} &= \rho Q_{2D} = 1000 \times 0.156 \approx 156 \text{ kg/s} \\
 \dot{m} &= 156 \text{ kg/s}
 \end{aligned} \tag{3.9}$$

3.3.4 Wall and Free-Surface Boundary Conditions

All solid surfaces in the computational domain were treated as no-slip impermeable walls. This includes the surfaces of the inclined panel and the rough channel bed. The panel surfaces were modeled as hydraulically smooth walls with zero velocity at the boundary. The wall boundary assignments adopted in the computational domain are shown in Figure 3.5

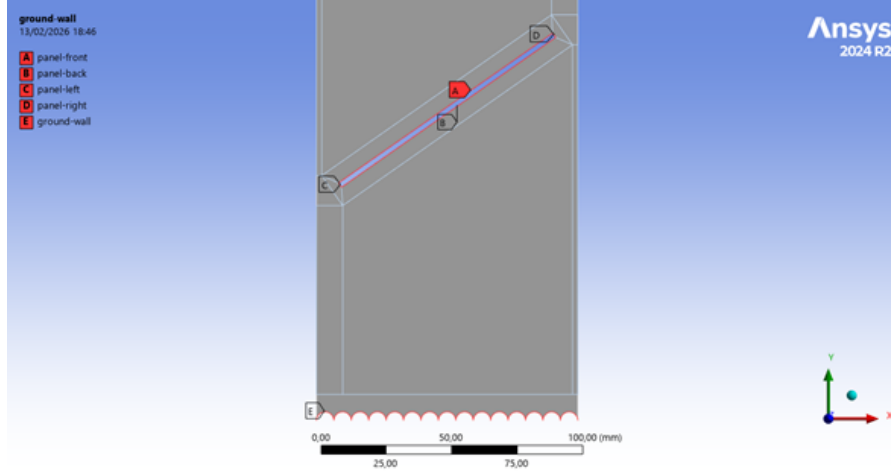


Figure 3.5: No-slip wall boundaries of the domain

The roughness of the channel bed was not represented by an equivalent sand-grain roughness model. Instead, the gravel layer was explicitly resolved in the computational geometry, as shown in Figure 3.6. The bed consisted of uniformly distributed semicircular roughness elements of diameter 6.66 mm, corresponding to the experimental median grain size (d_{50}). Each element represents the upper arc of a circular particle whose lower half is embedded in the bed, forming a continuous sequence of hemispherical protrusions along the bottom boundary. This approach allows the numerical solution to directly capture roughness-induced flow separation and local recirculation.

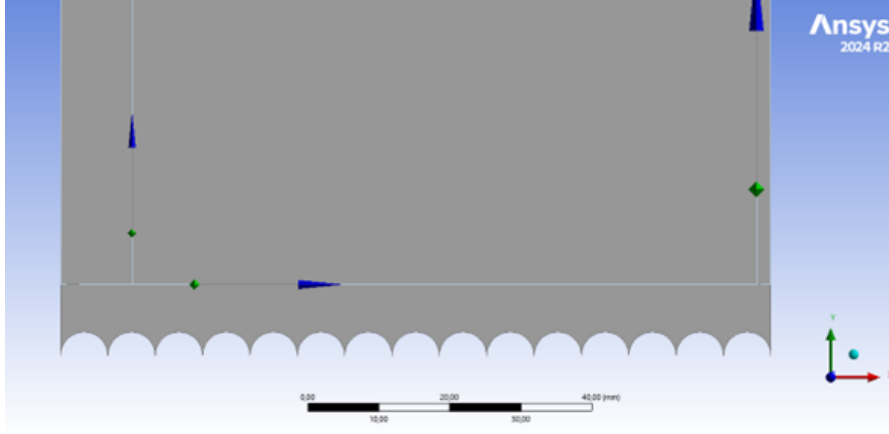


Figure 3.6: Resolved geometric representation of the channel bed roughness elements

The upper boundary of the domain was modeled as a rigid shear-free lid, imposing zero tangential stress while preventing normal velocity penetration. This simplification neglects free surface deformation and gravitational effects, which is consistent with the horizontal channel configuration and the imposed constant mass flow rate.

Details about the near-wall resolution strategy, roughness, Reynolds number, and y^+ considerations are presented in Section 3.5.

3.3.5 Two-Dimensional Approach

The computational model was formulated in two dimensions to represent a spanwise section of the experimental open-channel flow. The panels extend across nearly the entire channel width (605 mm within a 610 mm channel), resulting in negligible lateral clearance. Under these conditions, large-scale spanwise gradients in the mean flow are expected to be weak in the central region, and the flow can be reasonably approximated as quasi-two-dimensional. This geometric configuration supports the assumption that the dominant flow features are primarily governed by streamwise and vertical variations.

From a physical standpoint, the primary phenomena of interest, namely flow separation at the panel leading edge, shear layer development, and wake interaction between successive panels, are largely controlled by streamwise pressure gradients and vertical momentum exchange. These mechanisms can be effectively captured within a two-dimensional framework, particularly when the objective is to analyze

mean flow quantities such as velocity fields, recirculation regions, and aerodynamic forces. The two-dimensional formulation therefore provides a computationally efficient representation of the dominant flow physics relevant to the present study.

It is nevertheless acknowledged that turbulence is inherently three-dimensional and that spanwise coherent structures and secondary motions cannot be explicitly resolved within a two-dimensional Reynolds-averaged formulation. In the present approach, these effects are statistically modeled through the turbulence closure, rather than resolved directly. Consequently, the results should be interpreted as representing the time-averaged behavior of the flow, with the understanding that certain three-dimensional effects may not be captured explicitly.

3.4 Mesh Design

3.4.1 Mesh Topology and Refinement Strategy

The computational domain was discretized using a structured quadrilateral mesh. Local mesh refinement was applied in regions characterized by strong velocity gradients, including the leading and trailing edges of the inclined panel, the near-wake region, and the vicinity of the gravel bed roughness elements. A quadrilateral mapping strategy was employed in the region surrounding the roughness geometry to ensure a uniform cell distribution and adequate resolution of the semicircular gravel elements, as illustrated in Figure 3.7 and 3.8. The near-wall mesh growth rate was limited to 1.2 to avoid excessive cell stretching and to maintain numerical stability.

Particular attention was devoted to the resolution of the near-wall mesh, as the adopted k - ω SST turbulence model is formulated to resolve the viscous sub-layer directly. For this reason, the mesh was designed to achieve a wall-normal resolution corresponding to $y^+ \approx 1$, allowing an accurate prediction of wall shear stress without relying on empirical wall functions. This requirement is especially important for flows that involve separation and strong pressure gradients, where wall-function approaches may introduce significant inaccuracies.

Since the dimensionless wall distance y^+ is an output of the simulation, an initial estimate of the first cell height was obtained using empirical relations based on friction velocity. The friction velocity was approximated from an estimated wall shear stress, which was in turn related to the mean flow velocity through a

skin-friction coefficient. This preliminary estimate provided a guide for selecting the near-wall mesh size in the initial grid generation. The relations used for this estimate are given below:

$$u_* = \sqrt{\frac{\tau_w}{\rho}} \quad (3.10)$$

$$\tau_w = \frac{1}{2} \rho U^2 C_f \quad (3.11)$$

$$C_f = [2 \log_{10}(Re) - 0.65]^{-2.3} \quad (3.12)$$

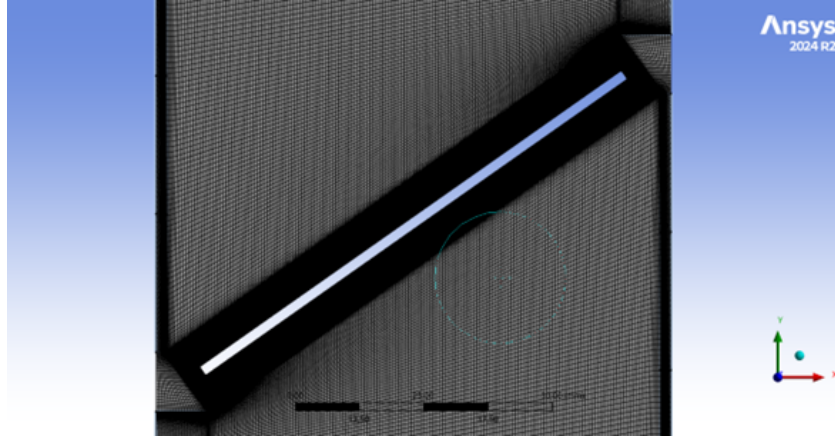


Figure 3.7: Close-up of the structured mesh and refinement zones for the inclined solar panel unit cell

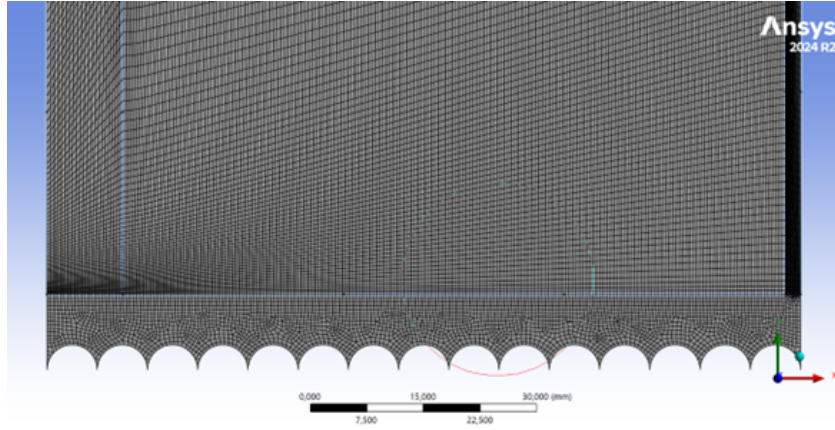


Figure 3.8: Close-up of the structured mesh and refinement zones at the resolved gravel layer

3.4.2 Near-Wall Resolution Strategy

The SST $k-\omega$ turbulence model is capable of resolving the viscous sub-layer directly when the near-wall mesh satisfies $y^+ \approx 1$. Therefore, no wall functions were used in the present simulations. Instead, the first cell height was selected to target $y^+ \approx 1$, ensuring direct resolution of the near-wall velocity gradients and the shear stress of the wall.

For the panel surfaces, the first cell height was approximately 8.57×10^{-2} mm along the main edges and 6.40×10^{-2} mm along the thickness edges. Around the semicircular roughness elements of the gravel bed, a uniform cell size of 0.5 mm was applied.

The roughness of the gravel bed was not represented using an equivalent sand-grain roughness model. In conventional wall-function approaches, the effect of roughness is introduced through a modified logarithmic velocity profile, characterized by dimensionless roughness height K_s^+ . However, for the present configuration, the characteristic roughness height relative to the viscous length scale results in large values of K_s^+ , corresponding to a hydrodynamically rough regime. Under such conditions, empirical roughness corrections become unreliable, particularly when the mesh resolution is smaller than the physical size of the roughness elements.

Consequently, the gravel bed was explicitly resolved as geometric semicircular elements within the computational domain, allowing the numerical solution to directly capture roughness-induced flow separation and form drag effects. The

dimensionless roughness height is defined as;

$$K_s^+ = \frac{\rho K_s u_*}{\mu} \quad (3.13)$$

$K_s^+ < 2.25$ Hydrodynamically Smooth

$2.25 < K_s^+ < 90$ Transitional

$9 < K_s^+ < 90$ Hydrodynamically Rough

3.4.3 Grid Independence Study and Mesh Quality

Mesh independence was assessed by comparing the drag and lift forces in the panel across the three mesh densities, shown in Table 3.2. All meshes produced nearly identical drag and lift values, indicating negligible sensitivity to further refinement. Since the variation between the three meshes was insignificant and the computational cost was already low, the coarsest mesh (166,000 cells) was selected for the final simulations.

Table 3.2: analysis of aerodynamic forces to mesh refinement

Mesh	Nodes	Lift (N)	Drag (N)
1	166,000	-0.250	0.17
2	265,000	-0.253	0.17
3	478,000	-0.253	0.17

Mesh quality was assessed using standard metrics, namely aspect ratio and skewness. For the selected mesh, the aspect ratio varied between 1.0 and 25.25, indicating moderate cell stretching primarily associated with regions of targeted directional refinement. Skewness values ranged from 1.3×10^{-4} to 0.942. Although the maximum skewness approaches the upper acceptable threshold, these highly skewed elements are confined to localized regions near curved roughness features and do not compromise numerical stability or convergence.

As summarized in Table 3.3, the mesh consists of 166,065 nodes and 164,946 elements. The overall distribution of aspect ratio and skewness confirms that the mesh quality remains within acceptable limits for reliable simulations. This is further supported by the smooth and monotonic behavior of residuals and force

coefficients throughout the computations, indicating that the mesh is sufficiently refined to ensure accurate and stable results.

Table 3.3: Mesh quality

Mesh	Nodes	Elements	Aspect Ratio		Skewness	
			Min	Max	Min	Max
1	166065	164946	1	25.25	0.00013	0.942

3.5 Numerical Discretization and Solver Settings

3.5.1 Solver Formulation

The numerical simulations were performed using the pressure-based solver available in ANSYS Fluent. The steady-state formulation was adopted because the primary objective of the simulations was to capture the mean flow characteristics around the inclined solar panel configuration. The general setup of the solver is shown in figure 3.9

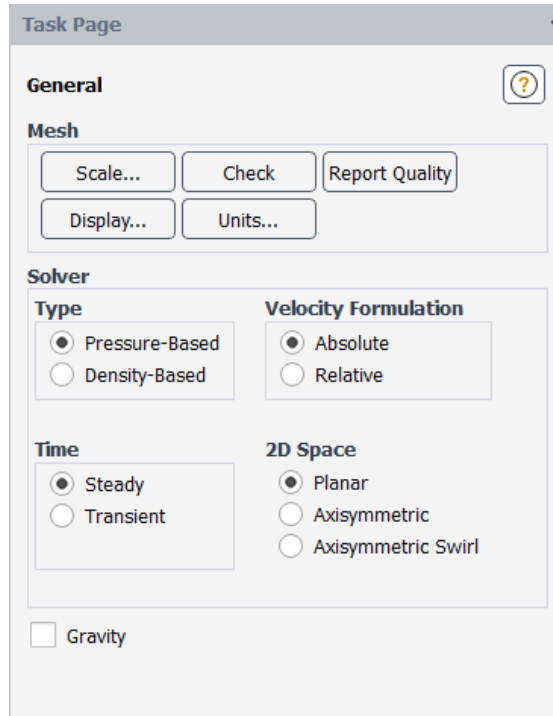
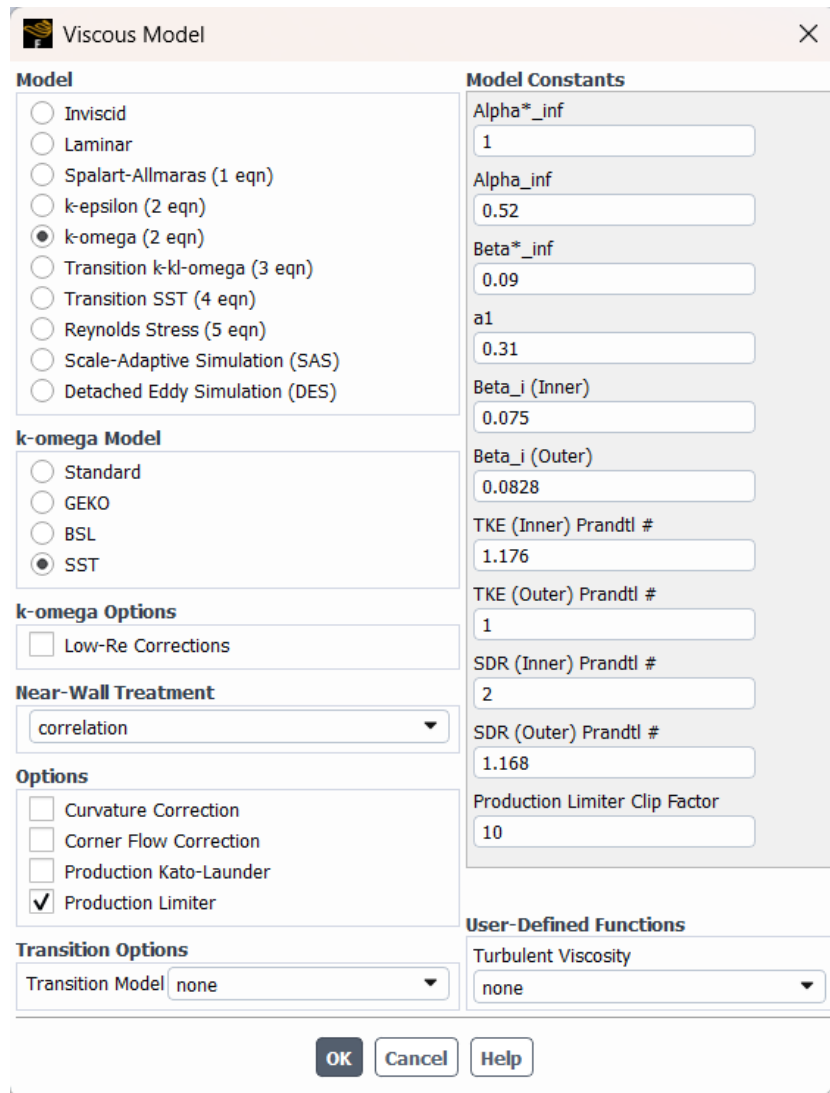


Figure 3.9: General solver configuration used in the simulations

Turbulence effects were modeled using the two-equation model k - ω Shear Stress Transport (SST). This formulation improves the prediction of separated flows and adverse pressure gradients, which are expected to occur downstream of the inclined panel.

The default model constants provided in ANSYS Fluent were retained as shown in Figure 3.10. The production limiter option was activated to prevent excessive turbulence production in regions with strong shear.



Viscous Model

Model

- ☐ Inviscid
- ☐ Laminar
- ☐ Spalart-Allmaras (1 eqn)
- ☐ k-epsilon (2 eqn)
- ☒ k-omega (2 eqn)
- ☐ Transition k-kl-omega (3 eqn)
- ☐ Transition SST (4 eqn)
- ☐ Reynolds Stress (5 eqn)
- ☐ Scale-Adaptive Simulation (SAS)
- ☐ Detached Eddy Simulation (DES)

k-omega Model

- ☐ Standard
- ☐ GEKO
- ☐ BSL
- ☒ SST

k-omega Options

☐ Low-Re Corrections

Near-Wall Treatment

correlation

Options

- ☐ Curvature Correction
- ☐ Corner Flow Correction
- ☐ Production Kato-Launder
- ☒ Production Limiter

Transition Options

Transition Model: none

Model Constants

Alpha*_inf: 1

Alpha_inf: 0.52

Beta*_inf: 0.09

a1: 0.31

Beta_i (Inner): 0.075

Beta_i (Outer): 0.0828

TKE (Inner) Prandtl #: 1.176

TKE (Outer) Prandtl #: 1

SDR (Inner) Prandtl #: 2

SDR (Outer) Prandtl #: 1.168

Production Limiter Clip Factor: 10

User-Defined Functions

Turbulent Viscosity: none

OK Cancel Help

Figure 3.10: Viscous model settings in ANSYS Fluent showing the implementation of the k - ω SST turbulence model and the associated model constants used in the simulations

3.5.2 Spatial Discretization

The governing equations were discretized using the finite volume method. Gradients were computed using the least-squares cell-based method. Spatial discretization of pressure used a second-order scheme, while second-order upwind schemes were used for the equations of momentum, turbulent kinetic energy (k) and specific dissipation rate (ω). The use of second-order spatial schemes ensures improved accuracy in resolving velocity gradients, separation regions, and wake structures downstream of the inclined panels. The principal numerical settings used in the simulations are summarized in Table 3.4.

Table 3.4: Summary of the numerical schemes and discretization methods.

Category	Setting	Value
Solver type	Solver formulation	Pressure-based
	Time formulation	Steady
	Velocity	Absolute
	Precision	Double
Pressure-Velocity coupling	Scheme	Coupled
	Pseudo-time method	Global time step
Spatial discretization	Gradient	Least-squares
	Pressure	Second-order
	Momentum	Second-order upwind
	k	Second-order upwind
	ω	Second-order upwind
Relaxation Factors	Pressure	0.5
	Momentum	0.5
	k	0.75
	ω	0.75
Convergence Criteria	Residual threshold	10^{-6}
Physical Assumptions	Gravity	Disabled
	Operation pressure	Default atmospheric

3.5.3 Convergence Criteria

The Convergence of the steady-state simulations was assessed on the basis of normalized residuals of all governing equations. As shown in table 3.5, a residual reduction threshold of 10^{-6} was imposed for the equations of continuity, momentum, turbulent kinetic energy and specific dissipation rate. In practice, residuals reached the prescribed convergence level before the maximum limit of 1000 iterations.

In addition to residual monitoring, global mass conservation across periodic boundaries was verified to ensure consistency of the imposed mass flow rate. The net mass imbalance across the domain was negligible, confirming the proper convergence of the periodic solution. Gravity was not activated in the simulations and the default operating pressure formulation was retained.

Table 3.5: Convergence criteria and monitoring parameters for the steady-state simulations.

Residual	Absolute Criteria
Continuity	10^{-6}
x-velocity	10^{-6}
y-velocity	10^{-6}
k	10^{-6}
ω	10^{-6}

3.6 Simulation cases and Operating conditions

The flow was modeled as incompressible water with constant properties, characterized by a density of 998.2 kg/m^3 and a dynamic viscosity of $\mu = 0.001003 \text{ kg/(m.s)}$. These values were selected to match the experimental setup and ensure consistency between the numerical and physical conditions.

The characteristic flow regime was defined using the Reynolds number based on the panel chord length, which represents the relevant length scale governing separation and wake development. The Reynolds number is given by:

$$Re = \frac{UL}{\nu} \quad (3.14)$$

where U is the bulk flow velocity, $L = 0.1$ m is the length of the panel, and $\nu = \mu/\rho$ is the kinematic viscosity of the water.

For the present study, a representative Reynolds number of approximately $Re \approx 3.1 \times 10^4$ was selected. This value corresponds to a bulk velocity of approximately $U \approx 0.31$ m/s, ensuring a fully turbulent flow regime consistent with the experimental mass flow rate conditions.

A total of four steady-state simulations were performed at this Reynolds number to investigate the influence of panel inclination and orientation. Two inclination angles (20° and 35°) were considered, each with positive and negative orientations relative to the incoming flow, resulting in four distinct configurations. The complete set of cases is summarized in Table 3.6.

Table 3.6: Summary of the simulation cases and computational domain dimensions.

Case	Angle	Domain Length-X (mm)	Domain Height-Y (mm)	Panel Length-Ref (mm)	Panel Thickness (mm)
1	35°	100	500	100	2
2	20°	100	500	100	2
3	-35°	100	500	100	2
4	-20°	100	500	100	2

Chapter 4

Results

This chapter presents the results obtained from the numerical simulations and examines their physical consistency and reliability. Before analyzing the flow behavior, it is essential to assess the numerical quality of the simulations, including convergence of the governing equations and adequacy of the near-wall resolution. These aspects ensure that the predicted flow field is not affected by numerical artifacts and that the turbulence model is applied within its intended limits.

Since identical meshes and solver settings were used for each pair of inclination angles ($\pm 35^\circ$ and $\pm 20^\circ$), and the flow direction was reversed to obtain the corresponding configurations, the numerical behavior was found to be consistent in all cases. Therefore, representative results corresponding to the $+35^\circ$ configuration are presented for this preliminary assessment. Following this evaluation, the analysis proceeded with a detailed comparison between numerical predictions and experimental observations.

4.1 Numerical Convergence and Y^+ Values

The convergence of the numerical solution is illustrated through the residual history of the governing equations, as shown in figure 4.1. All residuals exhibit a consistent and monotonic decrease, reaching values below the prescribed convergence criterion of 10^{-6} for all variables. The absence of oscillations or divergence indicates a stable and well-converged solution. This is particularly important given the use of periodic boundary conditions, which impose additional numerical constraints, including the consistency between inlet and outlet discretization. Despite the non-conformal

periodic setup, the smooth residual decay confirms that these requirements are adequately satisfied and that the solution can be considered reliable for further analysis.

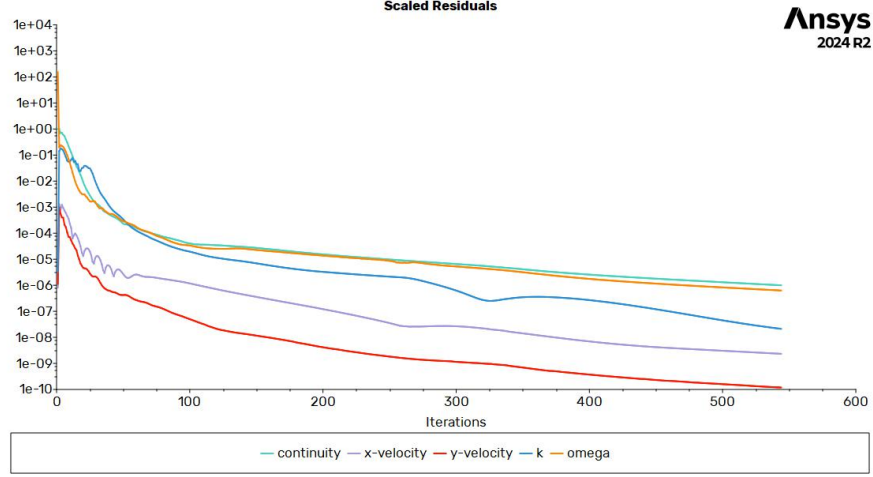


Figure 4.1: Residuals for the $+35^\circ$ panel inclination

The resolution of the near-wall was assessed through the distribution of the dimensionless wall distance Y^+ over all solid boundaries, including the ground surface and the walls of the panel. As shown in Figures 4.2 and 4.3, the obtained values remain predominantly below $Y^+ \approx 1$, with only minor localized peaks slightly exceeding this range. This indicates that the viscous sub-layer is adequately resolved across the domain, which is consistent with the requirements of the low Reynolds number formulation of the $k-\omega$ SST model. The periodic variation observed along the ground wall reflects the geometric roughness elements, while the panel surfaces exhibit smooth and continuous distributions. Overall, the results confirm that the mesh resolution is sufficient to accurately capture near-wall flow behavior without relying on wall-function approximations.

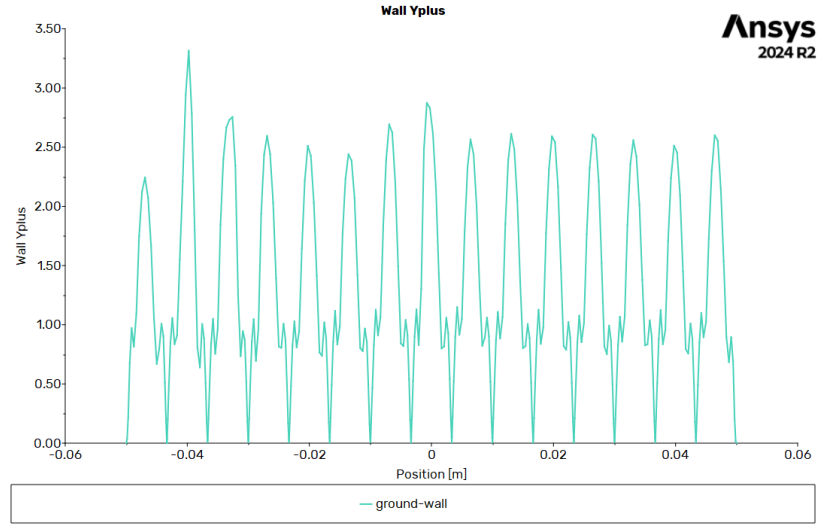
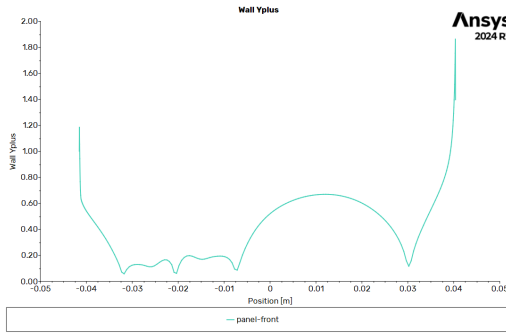
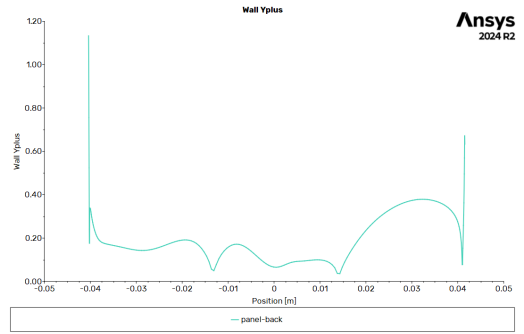


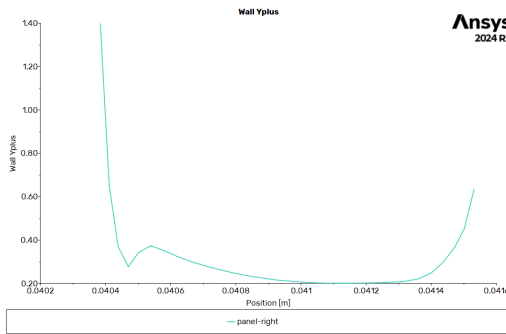
Figure 4.2: Distribution of y^+ along the bottom wall of the computational domain for the $+35^\circ$ inclination case



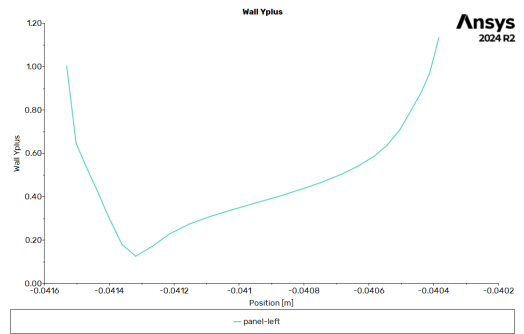
(a) Upstream surface



(b) Downstream surface



(c) Higher tip



(d) Lower tip

Figure 4.3: Distribution of y^+ along the surfaces of the panel for the $+35^\circ$ inclination case

4.2 Comparison of Numerical and Experimental Results

This section evaluates the predictive capabilities and limitations of the $k-\omega$ SST turbulence model by comparing with experimental data for four solar panel configurations corresponding to the inclinations of $+20^\circ$, $+35^\circ$, -20° , and -35° . All cases were simulated under identical inlet mass flow rate and Reynolds number conditions, allowing the inclination of the panel to serve as the primary control parameter. By varying the angle of inclination, the flow is subjected to different levels of aerodynamic blockage and associated pressure gradients, leading to progressively more complex flow behavior. In this context, the angle of inclination acts as a geometric parameter through which the model is assessed under increasing levels of flow separation and streamline curvature.

The analysis is structured to progressively examine the performance of the model, from global flow features to localized flow mechanisms. The evaluation begins with a qualitative assessment of the flow topology through streamline visualization. This step provides insight into the overall wake structure, including the location of the flow separation, the extent of the recirculation regions, and the reattachment behavior downstream of the panels. Comparison with experimental observations allows identification of discrepancies in the predicted global flow field, particularly in regions characterized by strong curvature and separation.

Following the topological assessment, the analysis transitions to a quantitative evaluation of momentum transport through the examination of horizontal velocity profiles (U_x) extracted at multiple downstream locations. These profiles provide a direct measure of momentum deficit and recovery. This comparison offers insight into the model's ability to represent turbulent transport processes and highlights the implications of the eddy-viscosity assumption in flows with strong anisotropy and history effects. In addition, the comparison provides an indication of the impact of the two-dimensional modeling assumption on the predicted velocity field.

Finally, the analysis focuses on the behavior of local flow through combined contour plots of velocity magnitude and velocity vectors. This visualization is used to examine how the flow redistributes momentum in response to the geometric blockage imposed by the panels. Under the imposed constant mass flow rate, the flow accelerates and redirects around the panels, generating strong shear layers and localized velocity gradients. By comparing numerical and experimental results

in these regions, it becomes possible to assess the model’s ability to capture the interaction between pressure gradients, shear layers, and turbulent mixing, particularly in zones where separation and reattachment dominate the flow behavior.

To ensure consistent interpretation of the results across all configurations, all spatial coordinates are presented in non-dimensional form. The flow-wise (x) and vertical (y) directions are normalized using the panel chord length, $h = 0.1$ m, which represents the characteristic length scale of the problem. The panel center is located at the midpoint of the computational domain in the streamwise direction. Consequently, the domain extends from $x/h = -0.5$ to $x/h = +0.5$, corresponding to a physical length of 100 mm. In the vertical direction, the flow depth is set to 0.5 m, resulting in a non-dimensional extent of $y/h = 0$ to $y/h = 5$. All profiles, contour plots, and vector fields presented in the following sections are expressed using this non-dimensional coordinate system, allowing direct comparison between configurations independent of geometric scaling.

4.2.1 Influence of Inclination Angle on Mean Wake Topology

An initial comparison of the $+35^\circ$ and $+20^\circ$ configurations begins on the upstream surface of the panels. In both geometries, the simulation successfully captures the fundamental flow splitting as the oncoming fluid impacts the structure. The model correctly identifies the sharp leading edge as the primary point of flow separation, where the streamlines cleanly detach to form the upper shear layer. However, when the transition to the downstream wake topology is made, significant structural deviations emerge.

For the $+35^\circ$ configuration figure 4.4, the simulation predicts a large, well-defined primary recirculation bubble. Although the location of the vortex core is broadly consistent with the experiment, the simulation overpredicts the overall size and vertical extent of the wake, producing a more coherent and spatially expanded recirculation region. Moving to a shallower $+20^\circ$ inclination figure 4.5, the overestimation of the spatial dimensions of the downstream wake becomes even more pronounced. The simulation depicts a heavily bloated, almost circular upper vortex. In stark contrast, the experimental streamlines for the $+20^\circ$ case reveal a primary wake that is highly elongated, flattened, and tightly hugging the upstream surface. Across both positive inclinations, the model consistently overpredicts the

vertical expansion of the primary downstream wake.

The most severe topological discrepancies between the simulation and the experiment are found in the overall dimensions of the recirculation zones and the formation of secondary wake structures near the trailing edges. Returning to the $+35^\circ$ configuration in figure 4.4, the simulation clearly predicts a large artificial secondary vortex along with several smaller, incorrect recirculation zones trapped directly underneath the downstream side of the panel. The experimental data explicitly contradict this topology, showing the flow sweeping cleanly and smoothly past the underside of the panel without rolling up into these distinct secondary structures. Moving to the $+20^\circ$ configuration in figure 4.5, the discrepancy in the spatial dimensions of the wake becomes the dominant feature. Although the simulation predicts a single, heavily bloated, and consolidated recirculation bubble that closes relatively quickly, the experimental results exhibit a much more elongated and complex wake. The true physical wake in the $+20^\circ$ experiment does not form a perfectly clean bubble; instead, it is characterized by an extended, chaotic regime with smaller, fragmented swirlings persisting further downstream from the trailing edge. Across both geometries, consistent with known limitations of eddy-viscosity models, the simulation severely overpredicts the spatial extent (both length and width) of the primary recirculation zones, failing to capture the physical flow bypass and the unsteady breakdown of the wake.

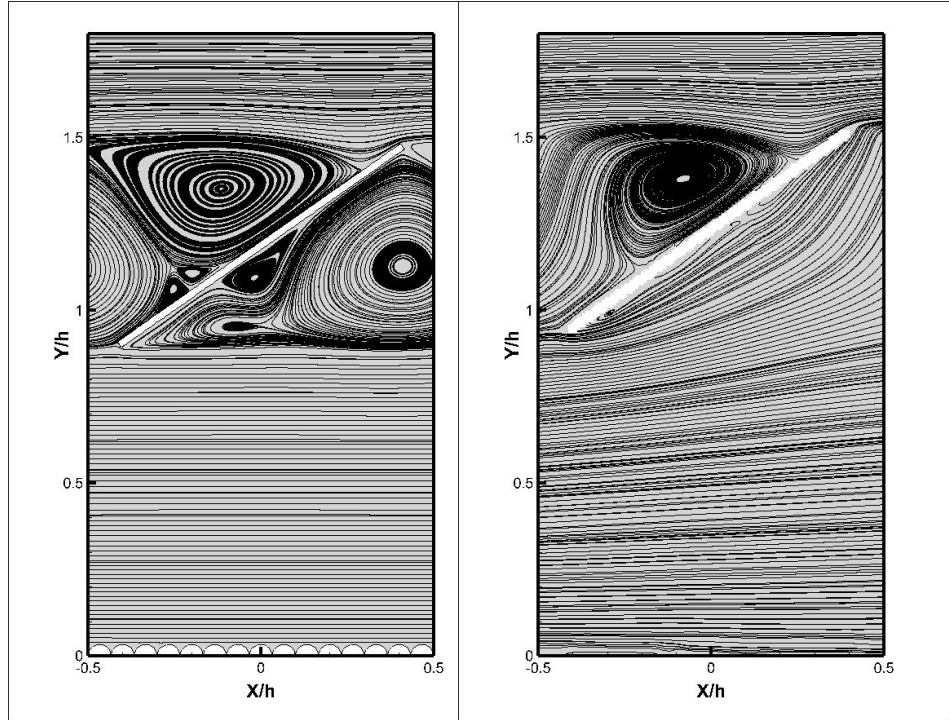


Figure 4.4: Mean flow streamlines of simulation (left) and experimental results (right) for the +35° panel inclination

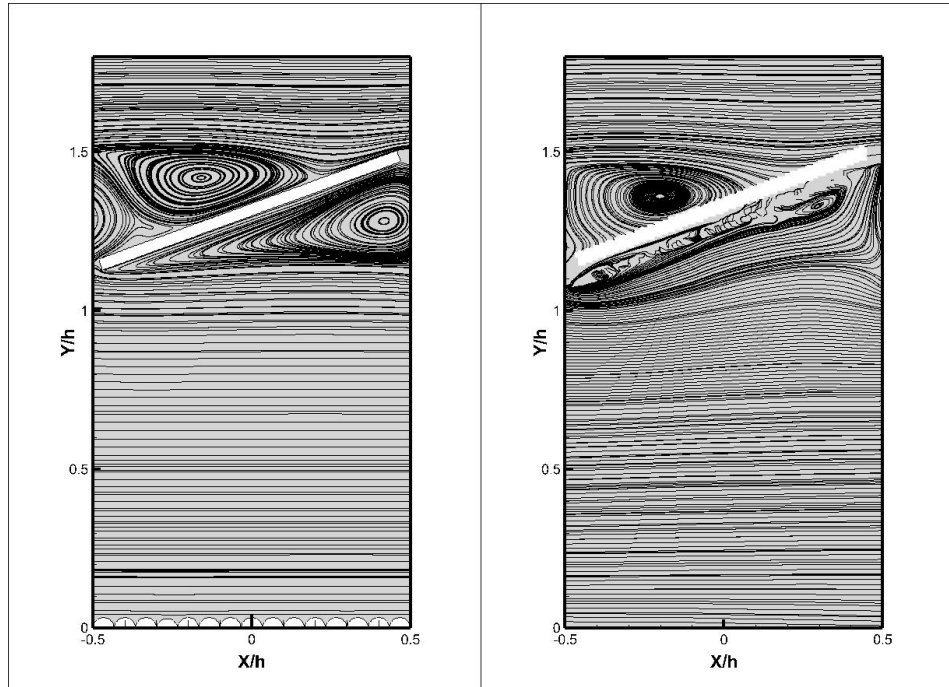


Figure 4.5: Mean flow streamlines of simulation (left) and experimental results (right) for the +20° panel inclination

For the steep configuration -35° figure 4.6, the most obvious failure is the structure of the wake underneath the panel. The simulation incorrectly breaks the wake into several small, separate recirculation bubbles trapped along the underside. The experimental data completely contradict this, showing one massive, unified recirculation bubble with a single large core positioned directly below the leading edge. The model completely fails to capture this single dominant structure.

Moving to the shallower -20° configuration figure 4.7, the simulation repeats the exact same “bloating” error observed in the positive angles. It predicts a perfectly round, heavily inflated vortex suspended under the middle of the panel. In reality, the experimental wake is severely flattened and stretched, pressing tightly against the bottom surface. Across both negative angles, the simulation struggles to capture the true, consolidated shape of the wake, forcing the flow instead into artificial, broken-up, or overly rounded shapes.

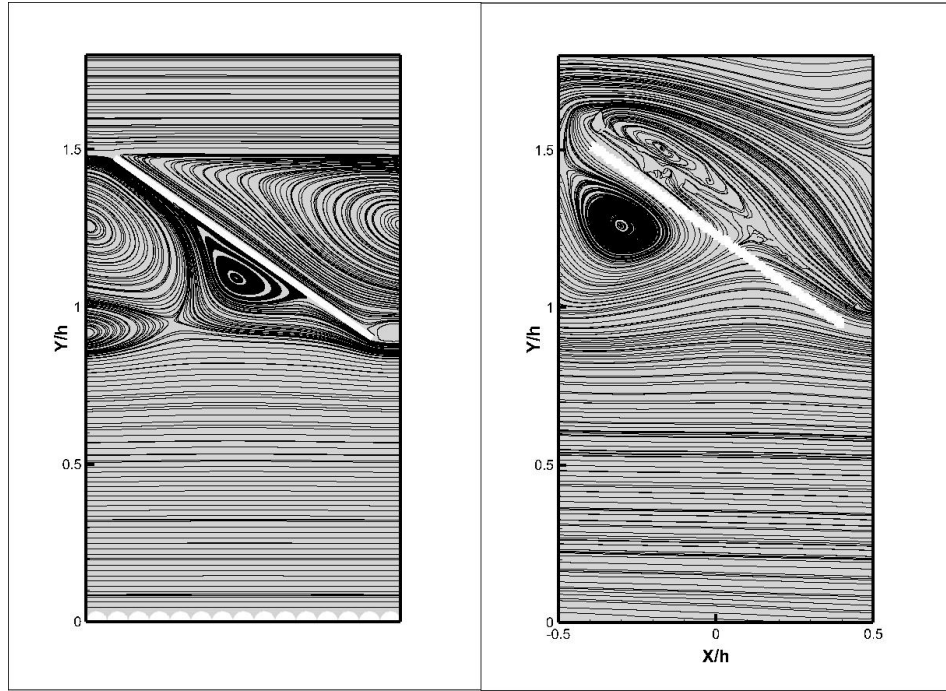


Figure 4.6: Mean flow streamlines of simulation (left) and experimental results (right) for the -35° panel inclination

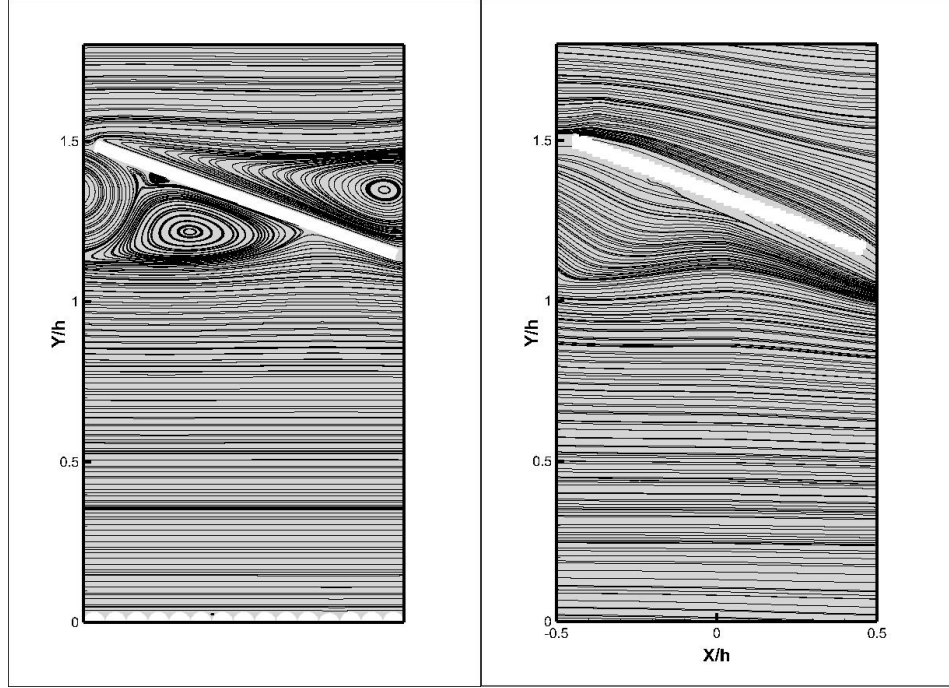


Figure 4.7: Mean flow streamlines of simulation (left) and experimental results (right) for the -20° panel inclination

Across all four configurations, the $k-\omega$ SST model consistently miscalculates the spatial boundaries and internal structures of the downstream wakes. Specifically, the simulation repeatedly overpredicts the overall dimensions of the primary recirculation zones. Rather than capturing the physical breakdown of the wake into smaller, complex eddies, the model forces the fluid into single, artificially stable bubbles. Furthermore, the model consistently fails to capture the true fluid trajectory at the trailing edges; it erroneously predicts trapped secondary vortices in regions where the experimental flow cleanly detaches and accelerates past the panel. This consistent deviation is directly related to the underlying Boussinesq approximation [3]. By enforcing a linear isotropic relationship between Reynolds stresses and the local mean strain rate, the model assumes that the turbulence is in local equilibrium and responds instantaneously to local geometric changes. As a result, momentum over-diffuses across the highly sheared layers, lacking the history-dependent “memory” required to accurately resolve these spatially evolving separation zones.

This fundamental limitation is most clearly exposed by the experimental results of the configuration -20° shown in figure 4.7. Unlike steeper inclinations, the

physical wake in the -20° experiment does not exhibit a massive, consolidated swirling vortex; instead, the flow remains tightly compressed against the underside of the panel. Nevertheless, the streamlines still exhibit distinct and continuous curvature as they track the inclined geometry. This streamline curvature introduces additional strain rates into the fluid. Because standard linear eddy-viscosity models are mathematically blind to the physical effects of streamline curvature [3], the simulation misinterprets this strain, leading to an inaccurate calculation of turbulent mixing. This mathematical failure forces the simulation to incorrectly inflate the wake into a bloated bubble. Consequently, while the $k-\omega$ SST model is traditionally favored for attached boundary layers with adverse pressure gradients [4], this evidence proves that its reliance on an isotropic eddy viscosity fundamentally compromises its predictive accuracy the moment the flow separates and curves.

4.2.2 Streamwise Velocity (U_x) Profiles

To quantitatively assess momentum transport and localized mass flux, time-averaged streamwise velocity (U_x) profiles were extracted from both the computational and experimental domains. Although the preceding streamline analysis provided a qualitative evaluation of the global flow topology, the present approach translates these observations into measurable quantities, enabling a more rigorous assessment of wake development and momentum redistribution. This comparison allows for an evaluation of the ability of the $k-\omega$ SST model to represent the flow response to the aerodynamic blockage imposed by the inclined panels.

To capture the spatial evolution of the flow field around the inclined panel, velocity profiles were extracted along three vertical lines located at $x/h = -0.25$, $x/h = 0$, and $x/h = +0.25$. The upstream location ($x/h = -0.25$) characterizes the initial flow development and its interaction with the leading-edge region of the panel. The central location ($x/h = 0$) corresponds to the geometric center and represents the region of maximum blockage, where strong acceleration and deceleration of the flow occur. Finally, the downstream location ($x/h = +0.25$) captures the flow behavior in the trailing-edge region, immediately before the full wake detachment.

For consistency between the numerical and experimental data, it is necessary to define the vertical extent of the measurement domain. Although both the experimental channel and the computational model have a total depth of water of 0.5 m (corresponding to $y/h = 5$), the Particle Image Velocimetry (PIV)

measurements do not extend to the free surface. The experimental data are limited to approximately $y = 0.39$ m (i.e., $y/h \approx 3.9$), which corresponds to the upper boundary of the measurement window.

A comparison of the flow-wise velocity profiles for the configurations $+35^\circ$ and $+20^\circ$ in figures 4.8 and 4.9 respectively, shows that within each data set, the three extraction locations ($x/h = -0.25, 0, +0.25$) exhibit a consistent and closely overlapping behavior. This indicates that for both simulation and experiment, the overall vertical distribution of the momentum is relatively uniform across the panel region, suggesting limited streamwise variation at the selected positions.

Although the overlap of the three extraction locations indicates limited streamwise variation within each dataset, a direct comparison between the numerical and experimental profiles reveals two regions of noticeable discrepancy. The first occurs in the near-bed region, particularly below the panel, where deviations in velocity magnitude are observed. The second appears in the upper part of the flow domain, approximately for $y/h \gtrsim 2.4$ – 2.5 , where the numerical results begin to diverge from the experimental profiles. These differences, although localized, highlight specific regions where the model struggles to accurately reproduce the experimental flow behavior.

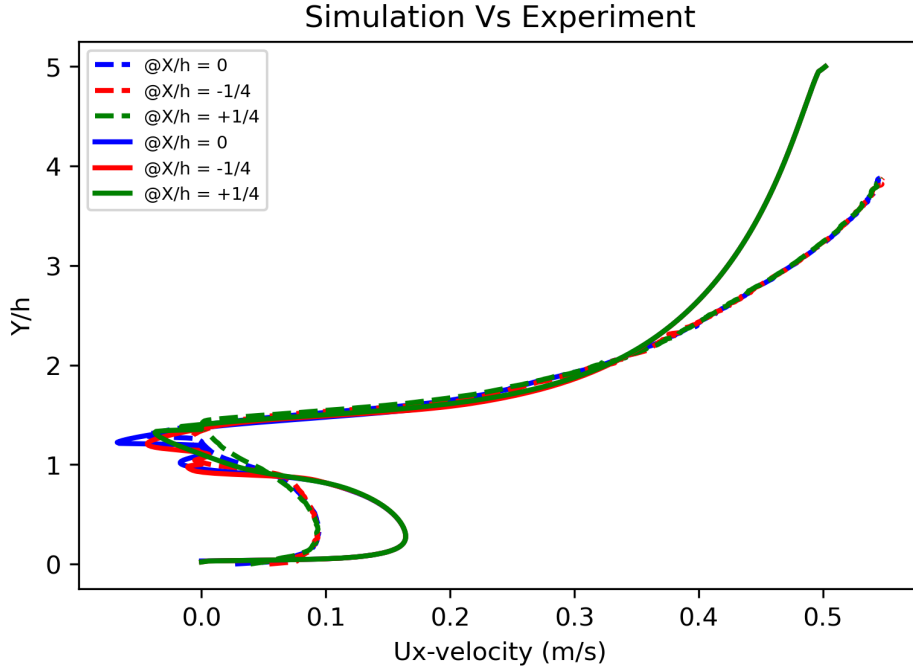


Figure 4.8: Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of $+35^\circ$

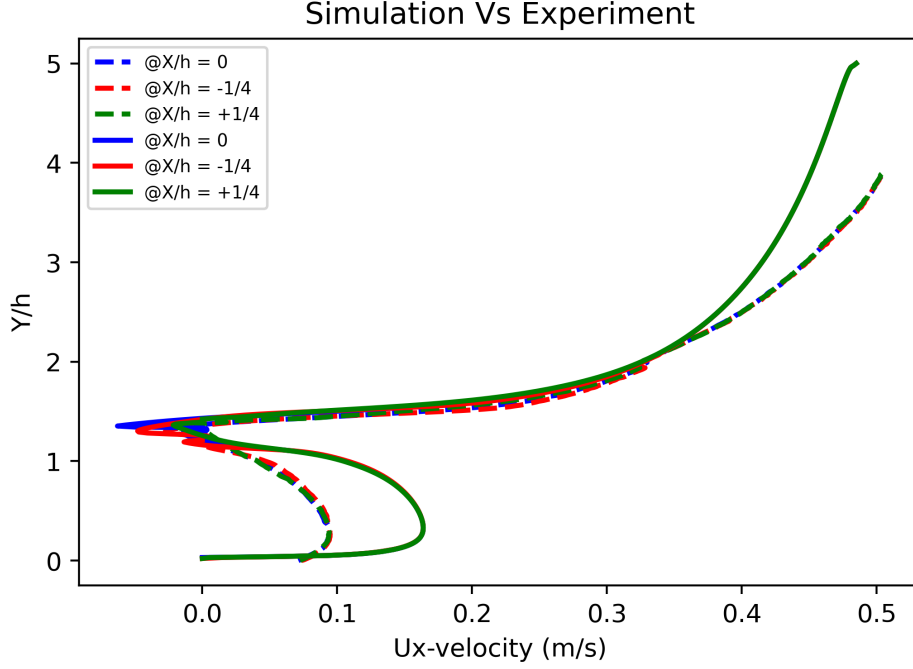


Figure 4.9: Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of $+20^\circ$

For the reverse inclination cases shown in the figures 4.10 and 4.11, the overall agreement between simulation and experiment deteriorates noticeably compared to the positive angles. Although the profiles extracted in the three stream locations ($x/h = -0.25, 0, +0.25$) still exhibit internal consistency within each data set, the overlap between numerical and experimental results is significantly reduced.

The discrepancies are particularly pronounced for the configuration -20° , where the simulation fails to reproduce the experimental velocity distribution across a large portion of the domain. In contrast to the positive inclination cases, the lower region of the flow ($y/h \lesssim 1.5$) exhibits systematically lower velocities in the simulation, while the experimental data indicate a stronger momentum retention. This suggests that the numerical model overpredicts momentum loss in the near-wall and recirculation regions.

Additional deviations are observed in the upper region of the flow ($y/h \gtrsim 2.5$), where the profiles diverge more significantly than in the corresponding positive-angle cases. The cumulative effect of these discrepancies indicates that the model struggles to correctly redistribute momentum throughout the domain when the panel orientation is reversed.

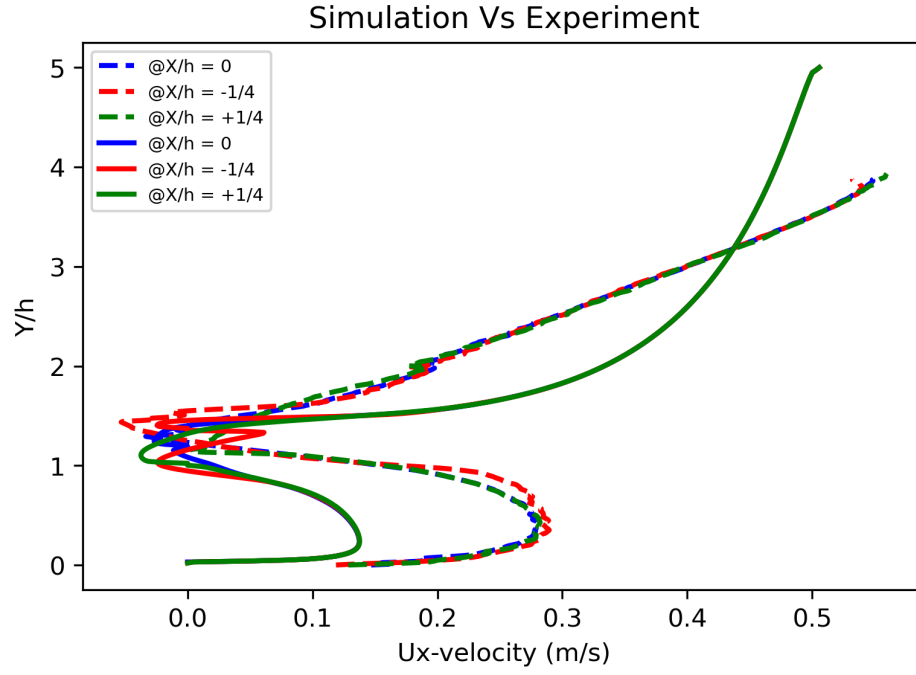


Figure 4.10: Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of -35°

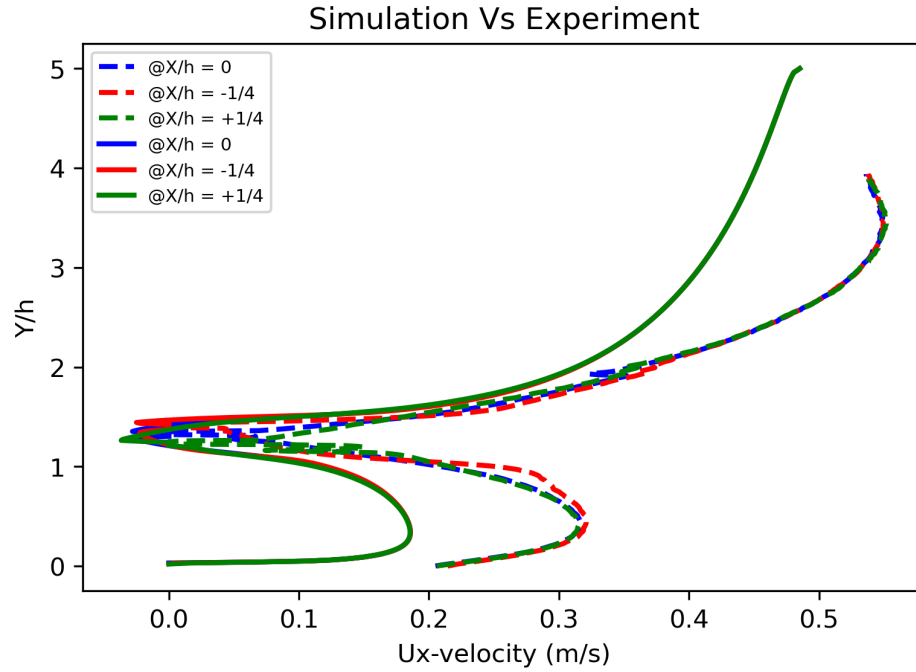


Figure 4.11: Streamwise velocity (U_x) profiles of simulation (solid lines) and experimental (dashed lines) results for panel inclinations of -20°

To quantify the agreement between simulation and experiment, the mass flux was evaluated by integrating the streamwise velocity profiles. The real experimental flow rate corresponds to 95 L/s of water ($\rho = 998.2 \text{ kg/m}^3$), while in the numerical model an equivalent mass flow rate of 156 kg/s was imposed (see Section 3.3.3), consistent with the two-dimensional formulation assuming a unit spanwise width.

Given the strong overlap observed among the three extraction locations, the comparison was performed in the central section ($x/h = 0$), which passes through the geometric center of the panel and represents the region of maximum blockage. The mass flux per unit width was computed as:

$$\dot{m} = \rho \int U_x(y) dy \quad (4.1)$$

To ensure consistency, the numerical profiles were integrated in the same vertical range as the experimental data, rather than in the full computational depth ($y/h = 5$). This avoids artificial discrepancies that arise in regions where no experimental data is available. The resulting mass flux values for both simulation and experiment, together with their relative differences, are summarized in Table 4.1.

Table 4.1: Quantitative comparison of the integrated mass flux between the numerical predictions and experimental data. Values represent the projected mass flow rate across a 1 m spanwise width. The percentage difference is calculated relative to the experimental value.

Configuration	+35°		+20°		−35°		−20°	
	Simulation	Experiment	Simulation	Experiment	Simulation	Experiment	Simulation	Experiment
Mass flux (kg/s·m)	102.1	105.7	104.7	105.7	103.6	106.6	106.6	145.9
Difference (%)	3.4%		0.93%		2.9%		26.9%	

The integrated mass flux values obtained from the velocity profiles in $x/h = 0$ are summarized in Table 4.1. For positively inclined cases (+35°, +20°) and the −35° configuration, the differences between simulation and experiment remain within approximately 1–3.5%. It is important to note that these comparisons are performed over a truncated vertical domain corresponding to the experimental measurement window. Since the total mass flow rate is exactly conserved in the numerical model, these deviations do not reflect a loss of mass conservation but

rather differences in the vertical distribution of momentum. Close agreement in these cases indicates that the simulation is capable of reproducing the overall momentum distribution within the measured region with reasonable accuracy.

In contrast, the -20° configuration exhibits a substantial deviation of approximately 26.9%, with the simulation significantly underpredicting the mass flux within the measurement window. This behavior is consistent with the flow structure previously observed, where the experimental results (see Figure 4.7) indicate a predominantly redirected flow with limited recirculation, while the simulation predicts a more pronounced separation and redistribution of momentum. As a result, the numerical model shifts a larger portion of the flow outside the measured region, leading to a deficit in the integrated mass flux. This highlights a clear limitation of the model in capturing the correct flow physics in this configuration.

The discrepancies observed in the previous sections can be directly attributed to the fundamental assumptions underlying the eddy-viscosity turbulence models. In the $k-\omega$ SST formulation, the Reynolds stresses are modeled using the Boussinesq hypothesis, which assumes a linear relationship between the Reynolds stress anisotropy and the local mean strain rate:

$$\langle u_i u_j \rangle = \frac{2}{3} k \delta_{ij} - \nu_T \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad (4.2)$$

or equivalently,

$$a_{ij} = -2\nu_T \bar{S}_{ij} \quad (4.3)$$

where a_{ij} is the Reynolds-stress anisotropy tensor and $\bar{S}_{ij}$ is the mean strain tensor rate. This formulation implies that turbulent stresses are instantaneously aligned with the local strain field, analogously to the behavior of a Newtonian fluid.

However, this assumption introduces two major limitations. First, it enforces an isotropic representation of turbulent momentum transfer through the use of a scalar turbulent viscosity ν_T , whereby the relationship between Reynolds stresses and mean strain is identical in all directions. This formulation neglects the inherently anisotropic nature of turbulence, in which the normal and shear stress components differ in magnitude and evolve independently depending on local flow conditions. As a result, the model cannot accurately represent directional variations in turbulent transport, particularly in flows involving strong shear, separation, and curvature. Second, and more fundamentally, the Boussinesq hypothesis assumes a local equilibrium between stress and strain, implying that the turbulence adjusts

instantaneously to changes in the mean flow.

This assumption is generally not valid, as turbulence evolves over a finite timescale on the order of k/ε , which may differ significantly from the characteristic timescale of the mean flow. In regions of strong curvature, separation, and recirculation, the turbulence retains memory of its prior deformation and does not respond instantaneously to local strain. Consequently, the Reynolds stresses are not aligned with the mean strain rate, violating the core assumption of the eddy-viscosity model. This leads to incorrect prediction of shear stresses and momentum redistribution, which is particularly evident in the -20° configuration, where the model fails to reproduce the experimentally observed flow structure and associated mass flux distribution.

The discrepancies observed in the upper regions of the velocity profiles (typically for $y/h \gtrsim 2.2\text{--}2.5$) can be partially attributed to the two-dimensional simplification of the numerical model. Although the simulations assume a spanwise-uniform flow, the experimental configuration is inherently three-dimensional due to the finite channel width and the presence of side walls. In real open-channel flows, lateral boundary layers and associated secondary currents redistribute momentum toward the channel center, leading to locally higher streamwise velocities away from the walls. This spanwise transport mechanism is absent in a two-dimensional formulation, resulting in an under-prediction of velocity in the upper and central regions of the flow compared to the experiment.

A further contributing factor may be the "velocity Dip" phenomenon. As reported in open-channel flow studies [29, 30], the maximum velocity can occur below the free surface in channels with moderate aspect ratios, primarily due to secondary currents and three-dimensional turbulence effects. Given the channel geometry and flow conditions of the present experiment, the higher velocities observed in the mid-to upper region are consistent with this behavior. However, since the experimental data do not extend to the free surface, the presence of a true velocity dip remains a plausible but not directly confirmed explanation from the measured profiles alone. Nevertheless, extensive literature demonstrates that the dip phenomenon is strictly governed by the aspect ratio of the channel ($A_r = B/h$), which consistently occurs in narrow channels where $A_r < 5$ [31]. Because the present experimental flume operates with a highly narrow aspect ratio of $A_r = 1.22$ ($B = 0.61$ m, $h = 0.50$ m), the formation of strong sidewall secondary currents and the subsequent velocity dip is a theoretically expected certainty, strongly supporting this justification.

4.2.3 Velocity Field Analysis: Magnitude Contours and Vector Representation

Building on the previous analysis, the velocity magnitude and vector fields primarily highlight differences in how momentum is distributed beneath the panel. As represented in Figures 4.12 and 4.13, for both the $+35^\circ$ and the $+20^\circ$ configurations, the region above the panel shows relatively good agreement between simulation and experiment, with similar velocity levels and flow alignment. In contrast, clear discrepancies emerge in the lower region: the simulation consistently predicts higher velocities beneath the panel, whereas the experimental results show a more pronounced reduction of momentum in this zone due to the upward redirection of the flow. This indicates that the model underestimates the extent to which the panel diverts momentum away from the lower region. The discrepancy is more evident for the $+35^\circ$ case, where the stronger inclination enhances flow deflection, while the $+20^\circ$ case exhibits the same trend with reduced intensity. Overall, these fields confirm that the model struggles to correctly redistribute momentum between upper and lower regions, particularly by over-predicting velocities beneath the panel.

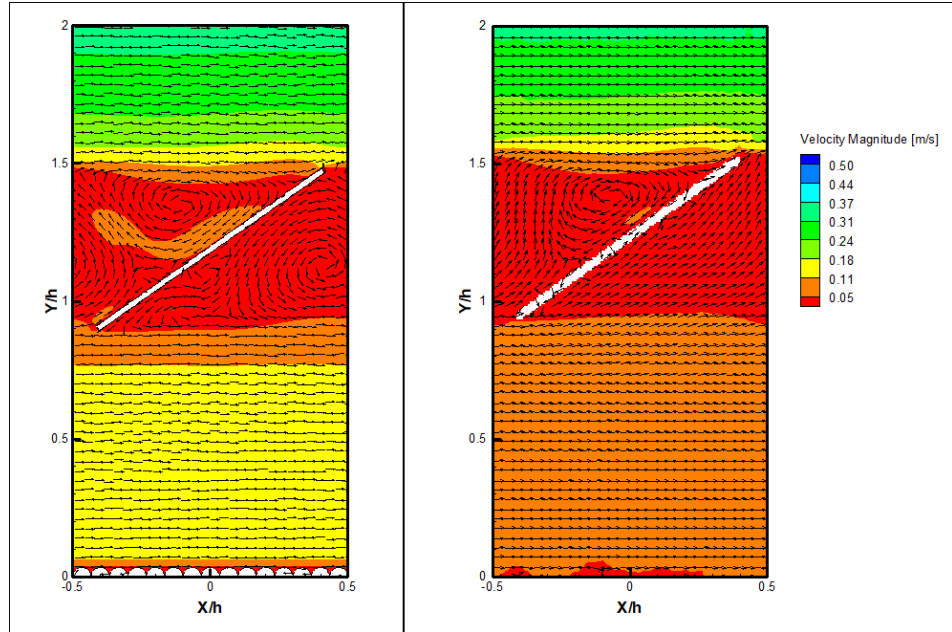


Figure 4.12: Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of $+35^\circ$

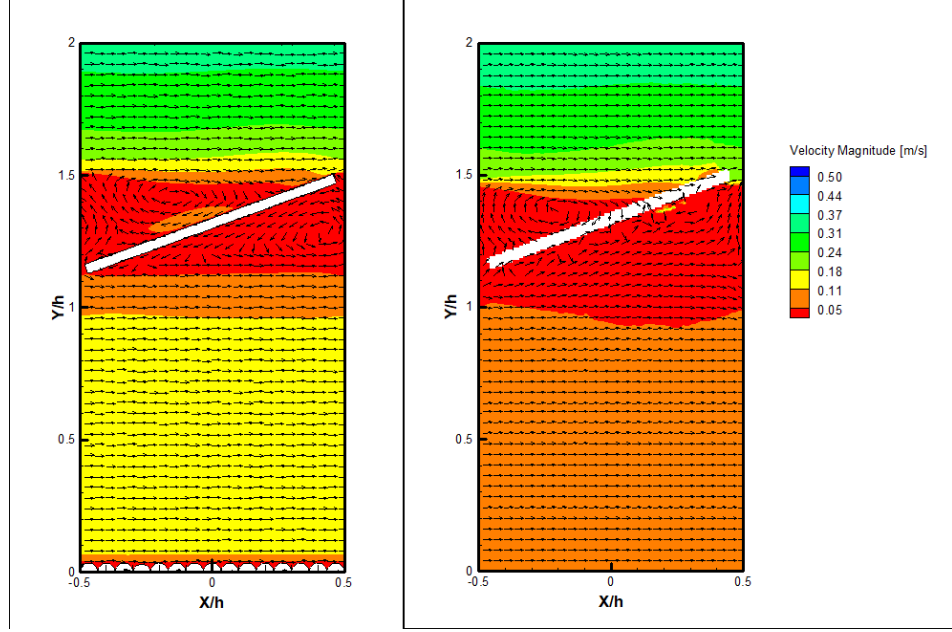


Figure 4.13: Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of $+20^\circ$

For the negative inclinations shown in Figures 4.14 and 4.15, the trend reverses. The panel guides the flow toward the lower region, and the experimental results show an increase in momentum beneath the panel. However, the simulation does not reproduce this behavior, instead predicting reduced velocities in this region, with the largest deviation occurring for the -20° configuration. In both cases, the model does not correctly follow the direction of momentum redistribution imposed by the inclination of the panel. Together, these observations confirm that the model struggles to accurately represent momentum transfer in flows involving strong curvature and directional changes. This limitation is directly related to the underlying eddy-viscosity assumption, which cannot capture the anisotropic and history-dependent nature of turbulence required to correctly resolve such complex flow behavior.

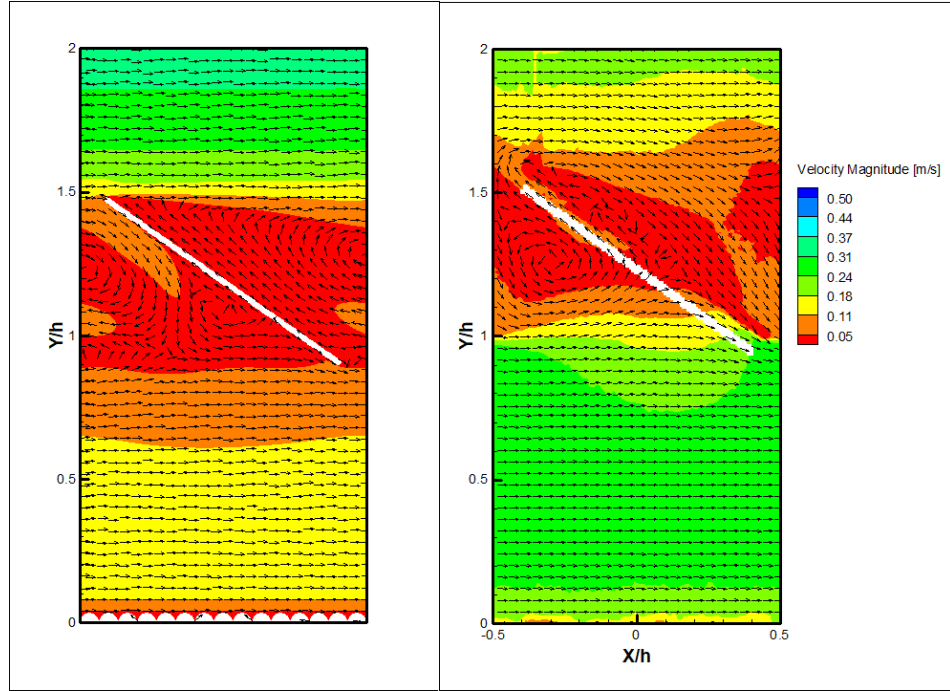


Figure 4.14: Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of -35°

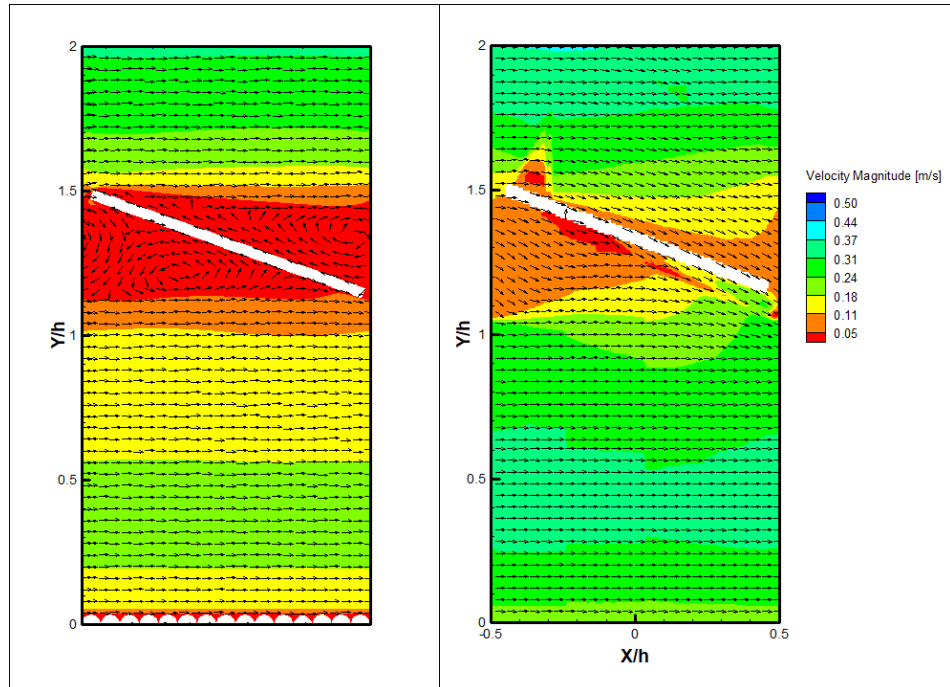


Figure 4.15: Mean velocity magnitude contours and vector fields of simulated (left) and experimental (right) results for panel inclinations of -20°

Chapter 5

Conclusion

This study assessed the ability of two-equation RANS turbulence modeling, specifically the k - ω SST model, to predict turbulent flow over inclined solar panel canopies under varying geometric configurations. The analysis shows that, although the model can reproduce global flow characteristics and maintain reasonable agreement in integrated quantities, this agreement is largely superficial. The imposed mass flow rate ensures consistency at a global level, but detailed comparisons demonstrate that the model struggles to correctly redistribute momentum within the domain. In particular, discrepancies in velocity profiles and flow structures confirm that similar bulk values do not imply an accurate representation of the underlying physics.

The limitations of the model become increasingly evident as the flow complexity increases with the inclination of the panel. Although moderate agreement is observed for configurations such as $+20^\circ$ and $+35^\circ$, significant deviations arise in more demanding cases, particularly the -20° configuration. In this case, the model fails to capture the experimentally observed flow redirection and instead predicts an incorrect separation pattern, leading to substantial errors in the momentum distribution. This confirms that model inaccuracies are strongly related to flow conditions involving curvature, separation, and rapid changes in flow direction.

More fundamentally, the results demonstrate that these deficiencies are directly related to the underlying assumptions of the eddy-viscosity framework. The Boussinesq hypothesis enforces an isotropic and instantaneous relationship between Reynolds stresses and the local strain rate, which becomes invalid in flows characterized by strong anisotropy and non-equilibrium turbulence. As a consequence, the model consistently misrepresents shear stresses, over-diffuses momentum in

separated regions, and fails to capture the directional and history-dependent nature of turbulent transport.

In addition, explicit representation of bed roughness, while physically motivated to better approximate real-world conditions such as agricultural terrain, introduces further complexity in the near-wall region. The interaction between roughness-induced turbulence and panel-generated flow separation amplifies the challenges faced by the model, making the accurate prediction of near-wall momentum transfer more difficult. This highlights that increasing physical realism, although necessary, can expose further limitations of simplified turbulence closures.

On the whole, this study demonstrates that while the $k-\omega$ SST model remains a practical and robust tool for many engineering applications, its predictive capability is fundamentally limited in flows involving strong separation, curvature, and complex momentum redistribution. These findings highlight the need for more advanced modeling approaches, such as Reynolds stress models or Detached-Eddy simulations, when an accurate representation of such flow features is required.

Bibliography

- [1] A. M. Aly and G. Bitsuamlak. «Aerodynamics of ground-mounted solar panels: Test model scale effects». In: *Journal of Wind Engineering and Industrial Aerodynamics* 123 (2013), pp. 250–260. DOI: 10.1016/j.jweia.2013.07.007 (cit. on pp. 1, 3).
- [2] I. Nezu and H. Nakagawa. *Turbulence in Open-Channel Flows*. Routledge, 2017. DOI: 10.1201/9780203734902 (cit. on p. 2).
- [3] Stephen B. Pope. *Turbulent Flows*. Cambridge University Press, 2000 (cit. on pp. 2, 43, 44).
- [4] Bengt Andersson. *Computational Fluid Dynamics for Engineers*. Cambridge University Press, 2012 (cit. on pp. 2, 16, 44).
- [5] Y. Hu, H. Zhang, Z. Luo, Y. Zhou, and G. Yuan. «Critical Wind Direction Angles and Edge Module Vulnerability in Fixed Double-Row Photovoltaic (PV) Arrays: Analysis of Extreme Wind Conditions Based on CFD Simulation». In: *Energies (Basel)* 18.9 (May 2025). DOI: 10.3390/en18092330 (cit. on pp. 3, 10, 11).
- [6] R. L. Simpson. «Turbulent Boundary-Layer Separation». In: *Annu. Rev. Fluid Mech.* (1989) (cit. on pp. 4, 6).
- [7] I. P. Castro and D. A. G. Robins. «The flow around a surface-mounted cube in uniform and turbulent streams». In: *Journal of Fluid Mechanics* 79.2 (1977), pp. 307–335 (cit. on pp. 4–6).
- [8] J. K. Eaton and J. P. Johnston. «A Review of Research on Subsonic Turbulent Flow Reattachment». In: *AIAA Journal* 19.9 (1981), pp. 1093–1100. DOI: 10.2514/3.60048 (cit. on pp. 4, 6).

- [9] A. D. Quinn, C. J. Baker, and N. G. Wright. «Wind and vehicle induced forces on flat plates Part 2: vehicle induced force». In: *Journal of Wind Engineering and Industrial Aerodynamics* 89 (2001), pp. 831–847 (cit. on p. 4).
- [10] W. Rodi. «Comparison of LES and RANS calculations of the flow around bluff bodies». In: *Applied Sciences (Switzerland)* (1997) (cit. on pp. 4, 6).
- [11] M. Tabak and D. J. Peake. «Topology of Three Dimensional Separated Flows». In: *Annual Review of Fluid Mechanics* (1982). URL: www.annualreviews.org (cit. on pp. 4, 5).
- [12] C. C. Chou, P. H. Chung, and R. Y. Yang. «Wind loads on a solar panel at high tilt angles». In: *Applied Sciences (Switzerland)* 9.8 (Apr. 2019). DOI: 10.3390/app9081594 (cit. on pp. 4, 8, 9).
- [13] J. H. Sizmaons, R. B. E. Lee, M. Kiya, and K. Sasaki. *Session 15 Fundamentals of Bluff Body Aerodynamics Free-Stream Turbulence Effects on a Separation Bubble*. 1983 (cit. on pp. 5, 6).
- [14] B. J. C. R. Hunt, C. J. Abell, J. A. Peterka, H. Woo, and J. C. R. Hunt. «Kinematical studies of the flows around free or surface-mounted obstacles; applying topology to flow visualization». In: *Journal of Fluid Mechanics* 86.1 (1978), pp. 179–200 (cit. on pp. 5–7).
- [15] W. J. Devenport and R. L. Simpson. «Time-dependent and time-averaged turbulence structure near the nose of a wing-body junction». In: *Journal of Fluid Mechanics* 210 (1990), pp. 23–55 (cit. on p. 5).
- [16] R. Martinuzzi and C. Tropea. «The Flow Around Surface-Mounted, Prismatic Obstacles Placed in a Fully Developed Channel Flow (Data Bank Contribution*)». In: *Journal of Fluids Engineering* 115.1 (1993), pp. 85–92. URL: <http://fluidsengineering.asmedigitalcollection.asme.org/> (cit. on p. 6).
- [17] C. H. K. Williamson. «Advances in our understanding of vortex dynamics in bluff body wakes». In: *Journal of Wind Engineering and Industrial Aerodynamics* 69-71 (July 1997), pp. 3–32. DOI: 10.1016/S0167-6105(97)00145-1 (cit. on p. 6).

- [18] B. F. Armaly, F. Durst, J. C. F. Pereira, and B. Schönung. «Experimental and theoretical investigation of backward-facing step flow». In: *J. Fluid Mech.* 127 (Feb. 1983), pp. 473–496. DOI: 10.1017/S0022112083002839 (cit. on p. 6).
- [19] J.-Y. Hwang and K.-S. Yang. «Numerical study of vortical structures around a wall-mounted cubic obstacle in channel flow». In: *Physics of Fluids* 16.7 (July 2004), pp. 2382–2394. DOI: 10.1063/1.1736675 (cit. on p. 6).
- [20] D. A. Paterson and C. J. Apelt. «Simulation of wind flow around three-dimensional buildings». In: *Build. Environ.* 24.1 (Jan. 1989), pp. 39–50. DOI: 10.1016/0360-1323(89)90015-2 (cit. on p. 7).
- [21] Y. Tominaga, A. Mochida, S. Murakami, and S. Sawaki. «Comparison of various revised $k\text{-}\varepsilon$ models and LES applied to flow around a high-rise building model with 1:1:2 shape placed within the surface boundary layer». In: *Journal of Wind Engineering and Industrial Aerodynamics* 96.4 (Apr. 2008), pp. 389–411. DOI: 10.1016/j.jweia.2008.01.004 (cit. on p. 7).
- [22] A. M. Aly. «On the evaluation of wind loads on solar panels: The scale issue». In: *Solar Energy* 135 (Oct. 2016), pp. 423–434. DOI: 10.1016/j.solener.2016.06.018 (cit. on pp. 8, 9).
- [23] A. Abiola-Ogedengbe, H. Hangan, and K. Siddiqui. «Experimental investigation of wind effects on a standalone photovoltaic (PV) module». In: *Renewable Energy* 78 (2015). DOI: 10.1016/j.renene.2015.01.037 (cit. on pp. 8, 9).
- [24] D. Mathioulakis, N. Vasilikos, P. Kapisiris, and C. Georgantopoulou. «Ground Clearance Effects on the Aerodynamic Loading of Tilted Flat Plates in Tandem». In: *Fluids* 10.6 (June 2025). DOI: 10.3390/fluids10060155 (cit. on pp. 8, 10, 13).
- [25] M. Trevisson, Y. Akutina, and O. Eiff. «Stereo-PIV measurements within the canopy in open-channel flows». In: *Exp. Fluids* 65.5 (May 2024). DOI: 10.1007/s00348-024-03797-x (cit. on pp. 8, 9).
- [26] M. Shademan, R. M. Barron, R. Balachandar, and H. Hangan. «Numerical simulation of wind loading on ground-mounted solar panels at different flow configurations». In: *Canadian Journal of Civil Engineering* 41.8 (2014), pp. 728–738. DOI: 10.1139/cjce-2013-0537 (cit. on pp. 9–12).

- [27] G. P. Reina and G. De Stefano. «Computational evaluation of wind loads on sun-tracking ground-mounted photovoltaic panel arrays». In: *Journal of Wind Engineering and Industrial Aerodynamics* 170 (Nov. 2017), pp. 283–293. DOI: 10.1016/j.jweia.2017.09.002 (cit. on pp. 11, 12).
- [28] J. Franke, A. Hellsten, H. Schlünzen, and B. Carissimo. *BEST PRACTICE GUIDELINE FOR THE CFD SIMULATION OF FLOWS IN THE URBAN ENVIRONMENT COST Action 732 QUALITY ASSURANCE AND IMPROVEMENT OF MICROSCALE METEOROLOGICAL MODELS*. Tech. rep. COST Office, 2007 (cit. on pp. 12, 13).
- [29] S.-Q. Yang, S.-K. Tan, and S.-Y. Lim. «Velocity Distribution and Dip-Phenomenon in Smooth Uniform Open Channel Flows». In: *Journal of Hydraulic Engineering* 130.12 (2004), pp. 1179–1186. DOI: 10.1061/(ASCE)0733-9429(2004)130:12(1179) (cit. on p. 50).
- [30] R. Absi. «An ordinary differential equation for velocity distribution and dip-phenomenon in open channel flows». In: *Journal of Hydraulic Research* 49.1 (2011), pp. 82–89. DOI: 10.1080/00221686.2010.535700 (cit. on p. 50).
- [31] Zhi-Qiao Wang, Nian-Sheng Cheng, and Charles S Melching. «Velocity distribution in open channel flows». In: *Journal of Hydraulic Engineering* 127.3 (2001), pp. 228–233 (cit. on p. 50).