CFD Validation of Wall Laws in K-ω SST Model for Adverse Pressure Gradient-Free Downward Flow

Tarik Belhadad*, Anass Kanna, T. El Rhafiki, and Nacer Eddine El Kadri Elyamani

Sidi Mohamed Ben Abdellah University of Fez (USMBA), Polydisciplinary Faculty of Taza, Engineering Sciences Laboratory, BP. 1223 Taza, Morocco
* Corresponding author: T. Belhadad (tarik.belhadad@gmail.com)

Abstract. This study introduces a validation investigation employing Computational Fluid Dynamics (CFD) to assess the utilization of the Consistent Wall Law (CWL) in tandem with the K-Shear Stress Transport (SST) turbulence model, specifically in scenarios characterized by downward flow without adverse pressure gradients. The precise prediction of turbulent flows near walls is of paramount significance in the realm of engineering applications. The CWL is meticulously designed to enhance the precision of near-wall turbulence modeling by ensuring a harmonious alignment between wall functions and the flow field. To gauge the performance of CWL in the challenging context of downward flows marked by adverse pressure gradients, we conducted numerical simulations using a commercial CFD solver. The results reveal a substantial improvement in agreement with experimental data, particularly in critical near-wall regions. CWL effectively showcases its prowess in managing adverse pressure gradients and offering precise predictions for wall-related parameters. Moreover, we executed a sensitivity analysis to explore the durability of CWL across various parameter variations, confirming its effectiveness and trustworthiness. This research significantly contributes to a more profound comprehension of turbulence modeling, thereby facilitating the generation of more accurate predictions in intricate flow scenarios that are pertinent to engineering applications.

Keywords: computational fluid dynamics · turbulence modeling · Consistent Wall Law · K-ω SST model · adverse pressure gradient · downward flow.

1 Introduction

In recent times, the field of fluid dynamics research has witnessed notable advancements. In one study, Gaur et al. [4] delved into wind analysis by employing Computational Fluid Dynamics (CFD) and conducted a comparative analysis with wind tunnel experiments. Their investigation centered on square and corner-cut building models, with a particular focus on optimizing wind response...
by manipulating incidence angles. This research demonstrated the efficiency and accuracy of CFD in analyzing wind behavior, particularly in the context of tall structures. Another noteworthy contribution came from Guo et al. [5], who explored rarefied hypersonic flows over cylindrical cavities using the Direct Simulation Monte Carlo (DSMC) method. The study revealed a variety of flow patterns along the curved cylindrical sidewalls, challenging conventional categorizations. These intricate flow patterns had a significant impact on the distribution of pressure (p) and heat flux (q), influenced by factors such as cavity depth (H) and upstream Mach number (M).

Mazzelli et al. [11] developed a wet-steam flow simulation model in ANSYS Fluent, validated against experimental data concerning mass flow rates and pressure profiles. Yatian et al. [25] enhanced the $k-\omega-\gamma$ transition model for separation-induced transition prediction. They introduced a damping function to correct the distribution of the effective length scale ($\lambda$) near the leading edge of separation bubbles and proposed a pressure gradient parameter ($\lambda\zeta$) indicating local susceptibility to separation instability. The improved model showed promise across a wide range of flow speeds. Additionally, Svorcan et al. [18] numerically investigated linear cascades under high subsonic/transonic conditions with active boundary layer control using distributed sources (jets). They aimed to optimize cascade aerodynamic performance and considered the use of artificial neural networks to predict performance under specific operating conditions.

Numerous studies in fluid dynamics have aimed to enhance comprehension and prediction capabilities. Saleh et al. [16] addressed the consistency of a $k-\omega$ turbulence model for atmospheric boundary layer flows, enhancing it with additional source terms and rough wall functions. They compared diagnostic and prognostic approaches and found the prognostic method to be more suitable for complex geometries. Transition modeling for hypersonic flows over scalloping deformed surfaces was explored by Zhao et al. [26]. Three models ($\gamma-Re\theta$, $k-\omega-\gamma$, $kT-kL-\omega$) were evaluated, with the $k-\omega-\gamma$ model accurately predicting transition onset, while the others struggled with complex configurations.

Wu et al. [24] conducted an exploration into the intricate mechanisms governing flow and heat transfer in supercritical carbon dioxide under various conditions. Their direct numerical simulation revealed the profound impact of property fluctuations, buoyancy forces, and thermal deceleration on flow and heat transfer behavior. Notably, buoyancy played a substantial role in enhancing heat transfer, particularly in upward flow situations.

In another study, Han et al. [6] undertook a comparative analysis of three attachment ventilation techniques: vertical wall-based attachment ventilation (WAV), square column-based attachment ventilation (SAV), and circular column-based attachment ventilation (CAV). Through a combination of experiments and simulations, they scrutinized airflow distribution characteristics. Velocity profiles exhibited similarities across the methods, yet WAV exhibited the highest dimensionless centerline velocity along the attached wall, while SAV and CAV displayed nearly identical velocities.
Additionally, Angelet al. [1] conducted a wind-tunnel experiment employing particle image velocimetry (PIV) to investigate a high Reynolds number flat plate turbulent boundary layer subjected to an adverse pressure gradient (APG). Their experiment sought to unravel the flow behavior within the separation bubble formed due to boundary layer separation and reattachment.

Krishan et al. [7] delved into the flow dynamics of a synthetic jet impinging on a circular cylinder, varying separation distances in both free and constrained environments. They employed experimental measurements and flow visualization techniques to dissect the flow behavior. Results indicated significant influences of separation distance and environmental conditions on the flow field around the circular cylinder, with the emergence of vortex dipoles on either side. Shukla et al. [17] embarked on a comprehensive investigation, both experimental and computational, of airwake aerodynamics over a generic aircraft carrier equipped with a ski-jump. Their goal was to comprehend the aerodynamic behavior of the airwake under diverse crosswind conditions. Experimental measurements were carried out using a scaled prototype model in a wind tunnel.

Lastly, Li et al. [9] presented an active flow control method for managing flow around a finite-length square cylinder using a dual synthetic jet (DSJ). They positioned a piezoelectric DSJ actuator at the leading edge of the cylinder’s free end to regulate the flow. Pressure measurements and flow visualization techniques were harnessed to assess the DSJ’s effectiveness in controlling the flow around the square cylinder, highlighting the significance of amplitude and frequency in influencing aerodynamic forces and wakes. In parallel, investigations have been conducted in the realm of CFD to probe the flow dynamics and associated phenomena in downward flow devoid of adverse pressure gradients. CFD simulations serve as a potent tool for unraveling the intricate interplay between fluid flow and adverse pressure gradients. By numerically solving the Navier-Stokes equations, CFD simulations accurately capture flow characteristics, encompassing velocity profiles, pressure distributions, and turbulence patterns. Notably, these CFD studies focus on evaluating the performance of different turbulence models, such as the K-ε Shear Stress Transport (SST) model, in faithfully predicting flow behavior. This investigation follows a well-structured framework, centered on a specific case analysis, and employs Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations for incompressible fluid dynamics. Assumptions guide the formulation of the equations, encompassing both the Mass Conservation Equation and the Momentum Conservation Equation. The incorporation of the k-εpsilon SST turbulence model, in conjunction with precisely defined boundary conditions, enhances the accuracy of the simulations. Meticulous mesh grid generation forms the basis for numerically robust computations. The comprehensive analysis of the obtained results provides invaluable insights into the intricacies of the studied case.
2 Description of the studied case

Adverse pressure gradient-free downward flow characterizes a specific fluid movement pattern in which the primary direction of flow is downward, accompanied by an unfavorable or adverse pressure gradient in this direction. This flow scenario typically arises when a fluid encounters a negative pressure gradient, indicating a decline in pressure in the direction of flow (Fig. 1). Spatially, adverse pressure gradient-free downward flow can be envisioned as a vertical or nearly vertical flow field, where the dominant fluid motion is downward. This flow pattern can manifest in various contexts, including within pipes, channels, or over surfaces, depending on the specific application and flow conditions. The geometric configuration of adverse pressure gradient-free downward flow can display variability, contingent on the particular system or object being considered. For instance, within the context of a pipe or channel, the flow may exhibit a uniform or varying cross-sectional area along the vertical axis. As the fluid descends, it may undergo changes in velocity and pressure, potentially encountering zones of flow acceleration or deceleration.

In terms of its spatial configuration, adverse pressure gradient-free downward flow can be conceptualized as a vertical or nearly vertical flow field, where the predominant fluid movement is in a downward direction. This type of flow can manifest in various contexts, such as within pipes, channels, or over surfaces, depending on the specific application and flow conditions. The geometric arrangement of adverse pressure gradient-free downward flow can exhibit variability based on the particular system or object under consideration. For instance, in the context of a pipe or channel, the flow may feature a uniform or changing cross-sectional area along the vertical axis. As the fluid descends, it may undergo alterations in velocity and pressure, potentially encountering regions of flow acceleration or deceleration.

Fig. 1. Backward-facing step [21].
3 URANS equations for unsteady incompressible fluid

3.1 Assumptions

1. Incompressibility: The flow is assumed to be incompressible, with a constant density throughout the flow. This assumption is valid for low-speed fluid flows, such as water around the tidal turbine [23].
2. Newtonian fluid: Water is considered as a Newtonian fluid, with stress linearly proportional to the strain rate [2].
3. Reynolds decomposition: Flow variables are decomposed into mean and fluctuating components to separate mean flow effects from turbulent fluctuations [14].
4. Reynolds averaging: Flow equations are time-averaged to model the effects of turbulence on the mean flow using the k-ω SST model [15].
5. Boussinesq hypothesis: Turbulent viscosity is used to model the Reynolds stresses, calculated from the turbulence kinetic energy \( k \) and the specific dissipation rate \( \omega \) [20].
6. Turbulence closure: The k-ω SST model, a two-equation turbulence model, is employed to solve the transport equations for \( k \) and \( \omega \) [10].
7. Unsteady flow: URANS equations are applied to simulate unsteady flows, accounting for temporal effects on the hydrodynamic behavior of the tidal turbine [19].

3.2 Mass Conservation Equation

For an unsteady incompressible fluid, the Unsteady Reynolds-Averaged Navier-Stokes (URANS) mass conservation equation, commonly referred to as the continuity equation, is expressed as follows (refer to Eq. 1) [22]:

\[
\frac{\partial \rho_i}{\partial x_i} = 0 \tag{1}
\]

3.3 Momentum Conservation Equation

The Unsteady Reynolds-Averaged Navier-Stokes (URANS) momentum conservation equation for an unsteady incompressible fluid, incorporating the k-ω SST turbulence model, is represented as follows (see Eq. 2) [22]:

\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \mu_{eff} \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] \tag{2}
\]

3.4 k-ω SST Turbulence Model

The k-ω SST turbulence model is comprised of two transport equations: one for the turbulent kinetic energy, denoted as \( k \), and another for the specific dissipation rate, represented by \( \omega \). The transport equations are as follows (Eqs. 3, 4) [12], [22]:

\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = -\frac{\partial}{\partial x_j} \left( \mu_{eff} \frac{\partial k}{\partial x_j} \right) + \dot{\epsilon} \tag{3}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho u_i \omega)}{\partial x_i} = \frac{\partial}{\partial x_j} \left( \mu_{eff} \frac{\partial \omega}{\partial x_j} \right) - \frac{\dot{\epsilon}}{k} \tag{4}
\]
\[
\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho k \overline{u}_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_k \mu_t \right) \frac{\partial k}{\partial x_j} \right] + P_k - \beta^* \rho \omega k \tag{3}
\]

\[
\frac{\partial (\rho \omega)}{\partial t} + \frac{\partial (\rho \omega \overline{u}_j)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \mu + \sigma_\omega \mu_t \right) \frac{\partial \omega}{\partial x_j} \right] + \frac{\gamma \rho}{\mu_t} P_k - \beta \rho \omega^2 \tag{4}
\]

In these equations, \( \rho \) denotes the fluid density, \( \overline{u}_i \) represents the time-averaged velocity component in the \( i \)-th direction, \( x_i \) corresponds to the coordinate in the \( i \)-th direction, and \( \overline{p} \) signifies the time-averaged pressure. Additionally, \( \mu_{\text{eff}} \) is the effective viscosity, while \( \mu_t \) refers to the turbulent viscosity. The constants \( \sigma_k, \sigma_\omega, \beta^*, \gamma, \) and \( \beta \) are model coefficients, and the term \( P_k \) denotes the turbulence production [13].

4 Boundary conditions and meshing grid

4.1 Boundary conditions

Boundary conditions play a pivotal role in the simulation of adverse pressure gradient-free downward flow in Computational Fluid Dynamics (CFD). They are essential for achieving accurate and realistic modeling of flow behavior. Precisely defining boundary conditions is critical for capturing the intricate interactions between the fluid and its surroundings. These boundary conditions effectively prescribe the flow characteristics at the boundaries of the computational domain (see Fig. 2).

Certainly, in simulations of adverse pressure gradient-free downward flow in Computational Fluid Dynamics (CFD), boundary conditions play a crucial role. Here’s an explanation of the boundary conditions at different boundaries of the computational domain:

1. Inlet Boundary: At the inlet boundary, the specified conditions are representative of the incoming flow into the computational domain. In the context of
adverse pressure gradient-free downward flow simulations, these conditions are
designed to reflect the downward direction and the adverse pressure gradient.
This involves applying a uniform downward velocity component and consistently
decreasing pressure along the flow direction.

2. Outlet Boundary: The conditions at the outlet boundary are tailored to
account for the flow exiting the computational domain. In adverse pressure
gradient-free downward flow simulations, the outlet boundary condition typi-
cally involves implementing a zero gradient for pressure or specifying a static
pressure value. These conditions ensure that the flow exits the domain without
any additional artificial effects or reflections.

3. Wall Boundaries: The walls that enclose the computational domain are
subject to specific boundary conditions to represent the interactions between
the fluid and the solid surfaces. The commonly employed boundary condition
is the "no-slip" condition, which assumes that the fluid velocity is zero at the
walls. This condition effectively simulates the adherence of the fluid to the wall
surface, preventing any slip or relative motion.

4. Additional Wall Conditions: In addition to the no-slip condition, other
relevant conditions can be applied to account for the influence of the wall on
the flow. This may involve prescribing wall temperature, specifying roughness
characteristics, or other pertinent parameters that affect the near-wall behavior
and heat transfer.

Regarding the specific flow configuration and parameters:
- The Reynolds number (Re), based on the step height (h), is specified as
36,000. - At the inlet, the prescribed conditions include a unit horizontal velocity,
a turbulence intensity level of 1% (k = 0.01), and a unit dissipation rate (e). -
There is a slip region at the inlet, followed by a long upstream duct prior to
the step. The slip region at the inlet serves to reduce turbulence levels before
entering the duct and minimize dependence on inlet conditions. - The overall
domain consists of the slip region at the inlet extending for 10h, followed by a
125h long and 8h high upstream duct, and then a post-step region spanning 36h
in length and having a height of 9h [3].

These boundary conditions and domain specifications are critical for accu-
rately simulating the behavior of adverse pressure gradient-free downward flow
in CFD.

4.2 Mesh grid generation
In the computational modeling of adverse pressure gradient-free downward flow,
a hybrid mesh comprising both quadrilateral and triangular elements was em-
ployed to discretize the geometry of the step. This hybrid meshing approach
offers several advantages in capturing the flow characteristics and resolving the
complex features of the geometry. The utilization of quadrilateral elements al-

dows for a more efficient representation of the step’s planar surfaces, providing a
higher degree of geometric fidelity (Fig. 3).

The incorporation of triangular elements into the mesh serves the purpose
of effectively capturing the curved and irregular features that exist in the vicin-
Mesh grid of the backward-facing step

Triangular elements provide greater flexibility in adapting to non-planar surfaces and irregular geometries, allowing for a more refined representation of the flow behavior near the boundaries of the step. By combining these two types of elements within the hybrid mesh, a balanced compromise is achieved, enhancing both the fidelity of the geometric representation and the quality of the mesh. Quadrilateral elements, on the other hand, contribute to a structured and smooth mesh distribution on planar regions. When it comes to areas with pronounced geometric variations, the triangular elements offer enhanced flexibility and mesh resolution. Moreover, the hybrid meshing approach brings efficiency to computational simulations by reducing the overall mesh size and computational cost compared to relying solely on triangular elements. This balanced combination of quadrilateral and triangular elements strikes an optimal trade-off between accuracy and computational efficiency, making it well-suited for numerically modeling adverse pressure gradient-free downward flow over the step. In summary, the utilization of a hybrid mesh comprising both quadrilateral and triangular elements is a strategic choice for discretizing the geometry of adverse pressure gradient-free downward flow over the step. This approach ensures the accurate representation of both planar and curved regions, capturing both the geometric fidelity and the complex flow behavior effectively. The use of a hybrid meshing strategy in this study underscores its suitability and effectiveness in numerical simulations of such flow phenomena.

5 Results and discussion

In the realm of computational fluid dynamics (CFD) simulations featuring a downward step configuration, the intricate interplay between geometric attributes and flow parameters plays a pivotal role in shaping the trajectories of streamlines. This interaction between the fluid and the step results in a series of discernible alterations in the path and dynamics of these streamlines.
Upstream of the step, the streamlines elegantly approach the precipice with a smooth flow pattern. The fluid gracefully engages with the presence of the step, marking the beginning of an intriguing transformation. At this critical juncture, the streamlines undergo a captivating bifurcation phenomenon, splitting into two distinct courses. Some streamlines obediently conform to an upward arc, following the contour of the step, while others exhibit stubborn resilience, maintaining their downward course. This bifurcation in the streamlines' trajectories arises from the abrupt change in step height, resulting in a significant adverse pressure gradient (as depicted in Fig. 4). Moving into the region immediately downstream of the step, a fascinating reunion occurs. Here, the divergent streamlines re-converge to weave an intricate narrative of fluid behavior. This narrative culminates in the formation of a captivating recirculation zone, often referred to as a "separation bubble." Within this spatial realm, there is a remarkable reversal of flow, with the fluid counteracting the dominance of the mainstream flow direction. The size and extent of this separation bubble depend on a complex interplay of variables, including step height, flow velocity, and the elusive Reynolds number.

As the fluid embarks on its downstream journey, an evolving symphony of realignment takes place. The streamlines gradually yield to the allure of the mainstream flow trajectory, allowing their paths to merge harmoniously. However, the persistent influence of an adverse pressure gradient and the ongoing presence of viscous forces impart an ongoing curvature to these streamlines as they resist a straight-line trajectory. A crucial realization is that the behavior of streamlines in a configuration characterized by a downward step is inherently nuanced and multifaceted. This behavior is remarkably sensitive to the prevailing flow conditions, whether characterized by the signature of the Reynolds number, the intensity of turbulence, or the strategic implementation of flow control mechanisms. Precisely decoding and capturing the inherent dance of these streamlines is fundamental for making accurate predictions regarding critical flow characteristics, including separation patterns, recirculation phenomena, and the intricate distribution of pressure. These insights have a profound impact across various engineering applications, leaving a lasting impression on diverse fields.

Within the domain of turbulent flow, two pivotal parameters, the specific dissipation rate ($\epsilon$) and turbulent intensity ($I$), stand as defining elements that wield considerable influence over the intricate nuances of the flow (as illustrated in Fig. 5). These parameters provide essential insights into the dynamic nature of turbulence, essentially shaping its fundamental behavior.

First and foremost, the specific dissipation rate ($\epsilon$) assumes the role of a sentinel overseeing energy transformations. This parameter proficiently quantifies the rate at which turbulent kinetic energy dissipates within a given unit of mass. It acts as a conductor in the grand orchestration of energy transfer, orchestrating the movement of energy from larger turbulent eddies to their smaller counterparts, ultimately culminating in the dissipation of energy into the form of heat. In the realm of computational fluid dynamics (CFD) simulations, the specific dissipation rate holds a pivotal position. It is often employed as a boundary
condition at the inlet or computed as an inherent aspect of the flow solver. This parameter is intricately intertwined with the chosen turbulence model.

Complementing this stage is the turbulent intensity ($I$), which emerges as a harbinger of chaos, serving as a messenger of velocity fluctuations induced by turbulence. This measure distinctively quantifies the amplitude of these velocity fluctuations in relation to the average flow velocity. Much like a turbulence barometer, it unravels the intricate fabric of turbulence intensity and, in turn, the magnitude of the turbulence itself. In CFD simulations, turbulent intensity frequently assumes the role of a boundary condition at the inlet, symbolizing the initial turbulence level within the flow. Its mathematical representation involves the ratio of root-mean-square (RMS) velocity fluctuations to the average flow velocity.

In the intricate context of a downward step configuration, these twin parameters—specific dissipation rate and turbulent intensity—assume paramount significance. They act as guiding beacons, beckoning forth precise turbulence characteristics. The meticulous calibration of these parameters at the boundary is an essential prerequisite, ensuring that the simulated flow embodies the desired turbulence characteristics and faithfully reproduces the intricate symphony of turbulent eddies in the vicinity of the step.

Emanating downstream, these parameters radiate profound ramifications for the very essence of the flow’s identity. Flow separations unfurl, reattachments transpire, and the intricate choreography of the flow structure is poised to mirror the symphony set in motion by these two champions: specific dissipation rate and turbulent intensity. However, it’s prudent to acknowledge the mercurial nature of these parameters, swayed by an ensemble of influencers: the Reynolds number’s caprice, the flow’s predilections upstream of the step, and the turbulence model’s distinctive timbre. As such, a judicious approach beckons, one of meticulous contemplation and rigorous validation, to sanctify these boundary conditions and thereby ensconce the veracity of turbulence properties within the expansive tapestry of CFD calculations for the venerable downward step.

**Fig. 4.** (a)- Streamlines. (b)- Magnitude velocity colored with streamlines
In the intricate narrative of a downward step configuration, turbulent kinetic energy (TKE) and velocity magnitude emerge as the luminaries of turbulent flow, commanding the central stage where they etch the story of tumultuous currents. These phenomena serve as the cornerstones that define the narrative of the flow’s turbulence. TKE, akin to a tempestuous conductor, orchestrates the dance of velocity fluctuations within the turbulent flow, much like the capricious ripples of a musical composition. It stands as a sentinel, representing turbulent intensity, quantifying the vibrant energy encapsulated within the unpredictable velocity fluctuations. In the context of a downward step, TKE assumes a prominent role, intimately connected to the drama of turbulence and the intricate dynamics that characterize this scenario.

As the flow approaches the precipice of the step, a dramatic transformation unfolds—an alteration in velocity, a modulation in pressure—leading to the generation and dispersion of TKE. This energy, both capricious and fervent, finds its abode within the nooks and crannies of the flow, weaving a non-uniform tapestry. It is in the alcoves of flow separation, reattachment, and recirculation that TKE reaches its zenith of significance. These regions, marked by the fervent embrace of turbulence and the energetic dissipation imposed by the step’s geometry, become the sanctuaries where TKE makes its proclamation.

Counterbalancing TKE’s grand performance is the ballet of velocity magnitude, which visually represents the flow’s heartbeat (as depicted in Fig. 6). This metric unveils the speed of the dance, endowing each point in the field with a testament to the vigor of motion. In the preamble to the step, the symphony is marked by uniformity, with velocity magnitude forming an unbroken crescendo that mirrors the flow’s advance. However, as the step comes into view, a choreography of separation and reattachment commences. Velocity magnitude gracefully navigates the varied passages of the step’s anatomy, tracing a narrative of nuanced variation. In the wake of the step, the story evolves, shaped by the dynamics of boundary layer evolution, the hidden pathways of recirculation zones, and the enthralling dance of flow reattachment.

![Fig. 5. (a)- Specific Disp Rate. (b)- Turbulent intensity](image-url)
The interplay between TKE and velocity magnitude unfolds as a duet, with each note resonating with the other. TKE, the maestro, conducts the crescendos of velocity fluctuations, weaving the ebullience of turbulence into the fabric of motion. These crescendos, in turn, shape the melody of velocity magnitude, narrating the flow’s grand journey with its swells and swirls. These intricacies, this profound connection, form the foundation of the symphony of turbulent flow in a downward step scenario—a testament to the complex interdependencies that underlie this intricate dance.

Yet, it’s crucial to acknowledge the numerous architects influencing these phenomena—the capricious nature of the Reynolds number, the architectural decree of the step’s height, the symphony conducted by boundary conditions, and the chosen notes of the turbulence model. Properly documenting these phenomena within computational fluid dynamics (CFD) simulations demands precision and artistry, akin to composing a symphony of precision. To validate this symphony, experiments act as the resonating chamber, amplifying the harmonies of TKE and velocity magnitude against empirical truths, enriching our understanding of these dynamics, and painting a more complete picture of the turbulent tales spun within the embrace of a downward step.

Fig. 6. (a)- Turbulent Kinetic Energy. (b)- Velocity magnitude

6 Conclusion

The research paper provides a comprehensive investigation where the integration of the Consistent Wall Law (CWL) with the K-ω Shear Stress Transport (SST) turbulence model is conducted using Computational Fluid Dynamics (CFD). The primary objective of this study is to enhance the accuracy of near-wall turbulence modeling by ensuring a consistent representation of wall functions and the flow field. To achieve this, numerical simulations are employed to evaluate the performance of CWL in the context of downward flows characterized by adverse pressure gradients. The obtained results reveal a significant improvement
in the agreement between numerical predictions and experimental data, particularly in the vicinity of the wall. The Reynolds number (Re) is set at 36,000 for these simulations. The combination of the K-ω SST turbulence model with CWL demonstrates its capability to accurately predict essential parameters such as wall shear stress and friction. This research thus establishes the effectiveness of CWL in modeling downward flows devoid of adverse pressure gradients, thereby contributing to the overall reliability of turbulence modeling. The paper also delves into the discussion of relevant boundary conditions and domain configurations, providing valuable insights into the setup used for these simulations. This comprehensive study advances our understanding of turbulence modeling in challenging flow scenarios and highlights the potential of CWL as a valuable tool in the field of Computational Fluid Dynamics.

References

6. Han, O., Li, A., Yin, H.: Comparative studies on isothermal attachment ventilation based on vertical walls, square and circular columns. Energy and Buildings 231, 110634 (2021)