

# **Establishing Fluid Dynamics Scales Critical to Dynamic Interface Applications and their Impact on Handling Qualities**

FY 2022 Report

Prepared by:

G. R. Whitehouse, A. Sharma and J. D. Keller  
Continuum Dynamics, Inc.  
34 Lexington Avenue  
Ewing, NJ 08618-2302

Prepared for:

David Gonzalez  
Office of Naval Research  
875 North Randolph  
Street  
Arlington VA 22203

Under Contract No. N00014-21-C-1044



---

Glen R. Whitehouse

August 26, 2022

## **Abstract**

The goal of this effort is to develop a better understanding of the relationship between the length and time scales typically present in the disturbance fields experienced by Naval aviators (i.e. ship airwakes, wing wakes etc.) and rotorcraft flight dynamics and flying qualities when the fully-interacting fluid dynamics of the disturbance and rotor wake and flight mechanics are accounted for. With this information in hand, the engineering community would be better able to:

- Understand the relationship between aircraft type, ride quality, flight dynamics and handling qualities during flight conditions where disturbance fields are encountered.
- More accurately define trainer requirements, minimum experimental campaign requirements, minimum CFD modeling requirements and consequently establish a benchmark to evaluate CFD predictive capability.
- Better define the requirements for new aircraft given the direct correlation between flight dynamics and handling qualities.

The effort, which commenced in March 2021 is undertaking the research required to develop such an understanding, along with disseminating the observations and conclusions of the work to the Navy and the broader flight dynamics and handling qualities community. The effort is structured using a build-up approach that first focuses on defining relevant disturbance fields, followed by predicting and understanding the fundamental aeromechanics response (e.g. aerodynamic forces and moments and rotor dynamics), and will culminate in predicting the flight dynamics and flying qualities for realistic, but generic, helicopter and tiltrotor configurations. To date, the effort has focused on investigating the type of disturbance fields experienced by Naval aviators, and how to define the relevant primary flow structures and time scales.

## **Table of Contents**

|                                                                                                        |     |
|--------------------------------------------------------------------------------------------------------|-----|
| Abstract .....                                                                                         | 2   |
| Table of Contents .....                                                                                | 3   |
| Summary .....                                                                                          | 4   |
| Introduction .....                                                                                     | 6   |
| Rotorcraft Aeromechanics and Interactions with Disturbance Fields .....                                | 7   |
| Methods, Assumptions, and Procedures .....                                                             | 12  |
| Technical Objectives .....                                                                             | 12  |
| Task Overview .....                                                                                    | 13  |
| Definition of Disturbance Field Fluid Dynamics Scales Relevant to Naval Rotorcraft<br>Operations ..... | 13  |
| Generation of Synthetic Disturbance Fields .....                                                       | 14  |
| Definition of Aeromechanics and HQ Metrics .....                                                       | 15  |
| Generic FD Model Assembly .....                                                                        | 16  |
| Fundamental Helicopter Response .....                                                                  | 16  |
| Fundamental Tiltrotor Response .....                                                                   | 17  |
| Helicopter Flight Dynamics Response .....                                                              | 17  |
| Tiltrotor Flight Dynamics Response .....                                                               | 17  |
| Results and Discussion .....                                                                           | 18  |
| Disturbance Field Characterization .....                                                               | 18  |
| Flow Characteristics .....                                                                             | 18  |
| Influence of Ship Motion on Ship Airwake .....                                                         | 57  |
| Wake Parametrization .....                                                                             | 79  |
| Simplified Wake Representation .....                                                                   | 81  |
| CFD Simulation and Validation .....                                                                    | 81  |
| Vortex Model Fitting .....                                                                             | 86  |
| Conclusions .....                                                                                      | 117 |
| References .....                                                                                       | 118 |

## Summary

One of the most demanding tasks for naval aviators is landing on a moving flight deck in high sea-states (i.e. the dynamic interface (DI) problem - see Figure 1). This task is made even more difficult by aerodynamic disturbances at the landing spot from the flow around the ship's bow, superstructure and deck edges. This highly unsteady ship airwake can lead to significant pilot workload. Flight simulation has long been recognized as a valuable tool for augmenting engineering development and pilot training in DI operations, however, it is most effective when the underlying simulation model has appropriately characterized the complex aerodynamic interactions between the rotorcraft and ship airwake. Given the complexity of the problem, a fundamental difficulty when assessing the simulation approaches is the quantification of “good enough”, both from the standpoint of understanding and predicting the underlying physics and also with regards to trainer fidelity (i.e. can the pilot feel/tell a difference). The premise of the proposed effort is to quantify “good enough” with regards to understanding the fundamental aero-physics of a rotorcraft interacting with an external disturbance field to quantify which length and time scales - such as those present in a ship airwake or in the wake of a an upstream aircraft during formation flight/refueling - directly impact the aircraft's fundamental response and flying qualities (FQ)<sup>1</sup> along with the aeromechanics modeling fidelity required to simulate interactions adequately.



*Figure 1: DI scenarios: landing on a moving deck (left), night operations (center) and approaching a pair ships (right)*

Several ONR funded efforts that seek to quantify “good enough” have recently completed, or are underway, at the University of Maryland, Georgia Institute of Technology and NAVAIR, where the academic researchers are focusing on understanding the response of a wing to the wake shed by canonical structures [1] and the latter (Generalized Airwake Goodness Evaluation) program seeks to represent the entire DI scenario with a variety of methods. This effort does not seek to duplicate that work, rather to complement it by focusing on several fundamental unknowns in the rotorcraft community, namely:

1. What spatial and temporal scales, present in a disturbance field, matter from Flight Dynamics (FD) and FQ standpoints? Beyond the obvious constraints (i.e. larger than a

---

<sup>1</sup> It is worth noting that there is a trend in the community to generalize and over use the term “handling qualities”, and not be specific. David Key, one of the primary creators/instigators of ADS-33, liked to differentiate between flying qualities, and handling qualities. In his definitions, flying qualities are how the dynamics of the vehicle respond to control inputs. Handling qualities relates to the pilot opinion and the pilot interface (sticks, displays, force and displacement gradients) to how the aircraft responds. This effort focuses on what Key refers to as “flying qualities” given the intention to not develop full aircraft specific control systems and pilot models for the assessment process. By focusing on “flying qualities”, the resultant trends and conclusions should be applicable to an entire class of rotorcraft rather than a specific aircraft and flight dynamics model. Furthermore, the proposed work can generate guidance on operating UAS rotorcraft (i.e. FireScout and successors) to establish acceptable Ship-Helicopter Operating Limits since there is no pilot on board to distinguish between flying and handling qualities.

rotor radius and those that induce velocities larger than the wake induced velocity), how does a rotor respond to different size disturbances and how does the rotor type influence this? For example, in general, a rotor acts to filter the influence of a disturbance field, but the level of filtering will vary significantly between an articulated flexible rotor (H-60) and a gimbaled stiff one (V-22). Moreover, rotors typically respond  $\sim 90^\circ$  out of phase of the disturbance, whereas fixed aerodynamic surfaces response directly in-phase, which results in a fundamentally different response between rotorcraft types (i.e. helicopter vs. tiltrotor).

2. How do these scales vary when the effects on the full aircraft are accounted for? The primary response from the helicopter will be dominated by that of the rotor, though there may still be significant response associated with the disturbance induced flow on the fuselage, empennage and tail rotor.
3. How do these scales vary with aircraft configuration/type (i.e. conventional helicopter vs. tiltrotor)? The primary response from the helicopter will be dominated by that of the rotor, whereas the tiltrotor may respond primarily in a fixed wing manner associated with the induced flow on the wing. Of course, the tiltrotor's response will also vary with nacelle angle, and it is well known that the V-22's response to wake disturbances (i.e. during formation flight) is quite different to other rotorcraft in the Navy's inventory [2].
4. How do these scales vary with aircraft flight condition, and can valid modeling simplifications be made (i.e. distorting vs. classical frozen disturbance field)? Work by Whitehouse and Brown for helicopter rotors [3-7] suggests that for high speed flight, the traditional frozen field (superposition) assumption may be adequate, but that at the low speeds associated with DI operations, a distorting disturbance field and wake are required because the response is critically different.

Developing an understanding of fundamental aircraft aeromechanic response, FD, and FQ to the unknowns outlined above would provide great utility to the community with regards to establishing the level of modeling fidelity required to accurately simulate disturbance interactions, the level of fidelity required to be output by CFD simulation generated ship airwake databases for training scenarios, and the quantification and specification of handling qualities to types of disturbance fields that can be used to define future and ongoing training and aircraft performance requirements and specifications, such as ADS-33 [8].

## **Introduction**

Rotorcraft flight dynamics models are typically based on simplified component aeromechanics, requiring tuning to capture realistic response characteristics. DI introduces other difficulties such as partial ground effect and interactions between the rotorcraft wake and ship airwake. Reliance on overly simplified modeling and limited bodies of experimental data undermines the realism necessary for investigating the aeromechanics of new configurations and for effective training. For example variations on linear superposition with a frozen disturbance field are still used to represent the influence of the ship airwake and other flow-field effects on the aerodynamics, and thus flight dynamics of the aircraft even though such approaches were shown to be invalid for rotorcraft interacting with external disturbances more than 15 years ago [3-7]. An example of the types of distortions experienced by the rotor wake and the disturbance field (in this case an idealized wake vortex trailed from an aircraft wing tip) is shown in Figure 2. As will be discussed below, this prior work is significant because it attempted to breakdown the response<sup>2</sup> of a rotorcraft interacting with an external flow field to the most fundamental level by idealizing both the aircraft and the disturbance field; however, given the level of fidelity available at that time no attempt was made to investigate the full aircraft 6 degree-of-freedom FD response, FQs or HQs. Only recently, in work by Continuum Dynamics, Inc. (CDI) in collaboration with NAVAIR Flight Vehicle Modeling and Simulation Branch personnel, have evaluations of fully-interacting DI simulations been undertaken in a research trainer [9] (see Figure 3) and a production fleet trainer [10].

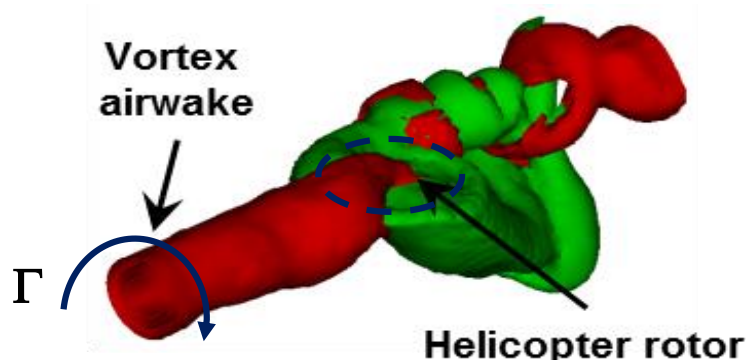


*Figure 2: Predictions of interacting rotor wake (green) and airwake (red) with direction of rotation of the vortex and the location of the rotor disk in blue*

Over the past few years, CFD has been used to improve the modeling fidelity of ship airwakes [11], with recent success in developing airwake databases for integration into fleet pilot trainers [12]. Development of such a database is a substantial undertaking, and may provide a higher fidelity model – subject to resolving the length and time scales that impact aircraft handling qualities. From a fundamental fluid mechanics standpoint, prediction of these flows will always require more fidelity until we can simulate the Kolmogorov scale directly; however, from FD and HQ standpoints a much coarser representation may be sufficient, or at least the fidelity of the stored airwake database can be coarser, rather than the resolution of the original CFD prediction. Exactly what that coarser representation looks like is unknown and current airwake database requirements [12] are an ad hoc compromise between perceived numerical storage availability, feasibility and intuition rather than a formal quantification based upon

<sup>2</sup>individual component response subject to changes in modeling assumptions and fidelity

aeromechanics, FQs and HQ.<sup>3</sup> Indeed, CDI, the Navy and others have already begun investigating methods to reduce the storage and computational cost of accessing these databases. This effort seeks to address this unknown directly from a physical insight standpoint, since once the scales of the disturbance that impart significant influence on the aeromechanics are known, the specification of an airwake database, and indeed the level of fidelity of the trainer, can be defined exactly. Finer resolution will always provide more information with regards to high frequency vibration (assuming that the trainer has sufficient predictive fidelity), but at a minimum, the scales that influence flight dynamics and FQs must be present. It is worth noting that independent of the complicated DI problem, the fluid dynamic scales present in atmospheric turbulence, formation flight wake interactions and nearfield operations (wake vortices and flow around/near to buildings) that influence realistic FD and FQs are also unknown beyond qualitative measures associated with low fidelity models and anecdotal evidence from pilots.

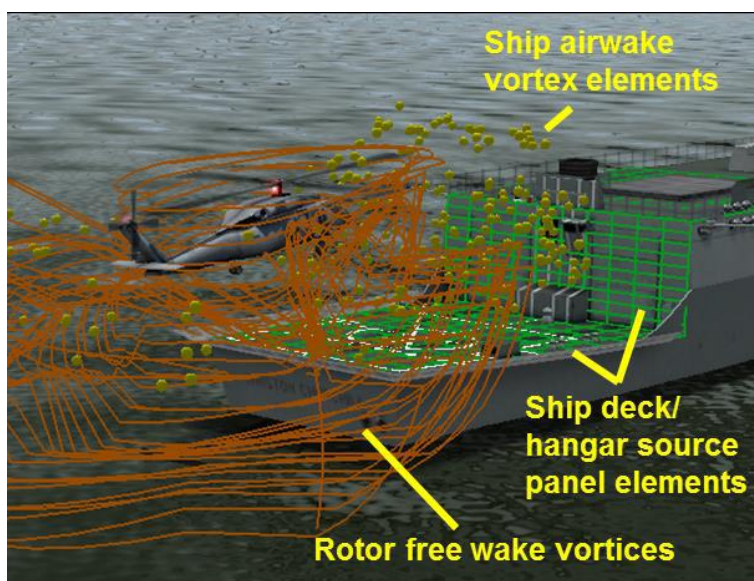


Figure 3: CASTLE / ExHel / CHARM visualization of shipboard operations

### *Rotorcraft Aeromechanics and Interactions with Disturbance Fields*

Analysis of the response of rotorcraft to gusts, atmospheric turbulence and disturbance fields, such as ship airwakes, has generally followed practices derived from the fixed-wing community, focusing on frozen gust profiles superimposed onto aircraft aeromechanics [13] - i.e. the fields are fixed in space and do not distort/respond to the aircraft. The underlying assumption is that the time scales of the interaction are too short for such a mutual fluid dynamic interaction to occur [6], and indeed, ONR sponsored work at UMD/GIT [1] is investigating exactly this for canonical fixed-wing configurations.

Unlike their fixed-wing cousins, the response of a rotorcraft to external disturbances can be broken into two parts, namely, the response of the rotor, and the response of the airframe. When integrated together the, net response of the complete rotorcraft can be determined. Historically, the influence of gusts on the rotor has centered on ensuring the dynamic stability of the rotor system [14] and making first order estimates of the heave response to sharp edged gusts in order

<sup>3</sup> Many ship airwakes are stored at frequencies up to 2Hz because this is the perceived bandwidth for pilot response, not aircraft or rotor response.

to quantify ride quality [13]. Rotor blade loading is usually much larger than the wing loading of conventional aircraft, and thus, since the heave damping derivative is inversely proportional to the blade loading, helicopter heave sensitivity is much lower than that for fixed-wing aircraft [13]. From a flight dynamics and handling qualities standpoint, estimates of the full six degree of freedom (6-DOF) derivatives can be similarly made, however, care must be taken to ensure that only the aerodynamic derivatives are used to determine “gains” related to gust response [13].

Such a derivative-based approach was a good fit with early rotorcraft flight dynamics simulations that applied simple stability derivative-based models. However, as rotorcraft became more complicated, the need to develop more sophisticated models for control law, control system development and training led to the adoption of a component build-up approach for aeromechanics models and FD simulations. In these simulations, each individual rotorcraft system/component is represented by a model that can be validated independently, and examples include GenHel [15] and the “example helicopter” model in NAVAIR’s CASTLE environment.

From an aerodynamics perspective, most modern rotorcraft FD models employ a blade element rotor model, with lifting-line/strip theory local aerodynamics coupled to a finite-state wake model; fuselage and empennage aerodynamics use a lookup table. The effective state-of-the-art in FD simulation employs a lifting surface blade aerodynamics model coupled to an unsteady vortex wake and panel method for the airframe [9, 10, 16]. CFD methods have been coupled to FD models of isolated rotors to study maneuvers (i.e. UTTAS) and DI scenarios [17], but full vehicle simulations using consistent (aerodynamic) fidelity across the aircraft components have not been performed due to computational cost.

The approach used to generate and represent the source of the rotorcraft gust model, or atmospheric disturbance, has taken a similar development route, with initial models following the fixed-wing paradigm of applying a frequency-domain linear statistical representation of atmospheric turbulence (determined by taking a PSD or Fourier transform of a suitable turbulence field – often the von Karman spectra or the Dryden “white noise” model [18]) as a frozen field. Extensions of this approach, such as Jones’ Statistical Discrete Gust method, enable more detailed representation of the structure of a gust since some of the gust’s phase information can be retained through the use of wavelets [13]. With the adoption of blade-element based aerodynamics models by the FD community, more sophisticated spatially accurate models of disturbance fields have been adopted by implementing analytical representations and 3D and 4D (space and time) look-up tables. Examples range from discrete vortices representing the wake (i.e. tip vortex) from a large fixed-wing aircraft implemented in research simulators [19, 20], to steady (3D) and unsteady (4D) ship airwake databases derived from CFD predictions implemented in operational flight trainers [12].

Despite significant advances in the level of realism in the disturbance field, and the acknowledgement that the assumption is fundamentally flawed when applied to rotorcraft [13], the vast majority of simulations still apply the frozen field model. Reliance on this assumption is predicated by the level of fidelity offered by contemporary analysis tools (i.e. they do not employ aerodynamics models capable of distorting the flow field)), but also by the prevalence of linear superposition in text books and papers when discussing the theoretical underpinnings of rotorcraft aeromechanics (i.e. [13, 21, 22]). Nevertheless, work over the past 15 years undertaken by the key personnel for this effort, and others, has shown a path forward that will form the basis of the work described here.

In the early 2000s, there was a resurgence of interest in more tightly integrating fixed- and rotary-winged aircraft into the civil airspace, and research in Europe investigated the impact of fixed-wing wake vortices on helicopter flight dynamics [3-7, 20, 23-25]. Padfield and co-workers [20, 23-25] employed a conventional 6-DOF FD model of a Westland Lynx-like helicopter, with finite-state-wake, fuselage aerodynamic lookup table and frozen disturbance field, to determine that the helicopter's handling qualities could be significantly degraded and that the fuselage response could be large and important. Whitehouse and Brown [3-7] investigated the impact of the level of modeling fidelity used to represent the wake and the disturbance field on the aeromechanics of a generic isolated rotor interacting with a tip vortex from a large aircraft. The level of modeling fidelity ranged from analytical idealizations and traditional frozen field and finite state wake models, to fully distorting CFD models solving the Eulerian (grid-based) form of the incompressible Navier-Stokes equations in vorticity-velocity form. To quantify the effects of modeling fidelity in a consistent and rigorous manner, the flight dynamics of the model and the controls were constrained and the fundamental rotor aerodynamic and dynamic responses were calculated. Whitehouse and Brown concluded that when the wake and disturbance field interact over relatively long timescales, such as during low speed flight, the validity of the frozen field assumption must be called into question [6]. Predictions revealed that at low forward speeds the distortion of both the rotor wake and the disturbance field could be significant (see Figure 2), and that the severity of the rotor's response is reduced when compared to the frozen field case (see Figure 4). At high speeds, where the interaction time scale between the rotor wake and the disturbance field is much shorter, the frozen field assumption appears to be more appropriate; nevertheless, differences in the predicted thrust coefficient could vary by as much as 10% of the trim value.

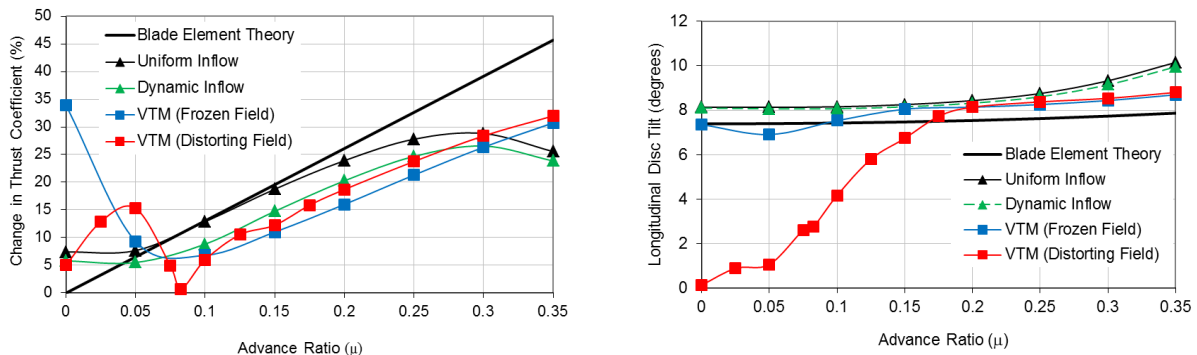


Figure 4: Mean perturbations to rotor thrust and longitudinal disc tilt for an isolated rotor interacting with a vortex aligned with the flight path. Adapted from [6, 7]

Around the same time in the United States, CDI started to use a distorting Lagrangian wake model to investigate multiple aircraft wake interactions in the terminal area for NASA [26]. This model built upon CDI's extensive history modeling aircraft wake vortices [27-32], and included FD models for a variety of configurations for preliminary safety assessment. This work was later extended under Navy sponsorship to investigate multiple aircraft shipboard operations [33]. More recently, this model was integrated into the Navy's CASTLE environment, and has been used by the Navy to plan and diagnose V-22 formation flight operations [2].

In recent years, several efforts of significance have demonstrated proof-of-concept for undertaking fully-coupled FD and disturbance field fluid dynamics predictions [9, 10, 17]. In

[17], researchers at NAVAIR integrated the Kestrel CFD solver with CASTLE to simulate a helicopter landing on a ship. Due to computational cost considerations, the CASTLE blade element rotor model was used for the blade aerodynamics and to determine the strength of appropriate momentum sources that should be inserted into the CFD grid to represent the wake. Kestrel then calculated the evolution of rotor wake and the ship airwake, including their mutual interference, and provided induced velocities to the blade model. No attempt was made to represent the helicopter fuselage or to undertake blade-resolved CFD; only qualitative results were presented. More recently, CDI, in collaboration with NAVAIR, interfaced the CHARM unsteady aerodynamics module (lifting surface blade aerodynamics model coupled to an unsteady vortex wake, panel method airframe and vortex particle disturbance field) with CASTLE and undertook fully-coupled simulations of a helicopter approaching and landing on a ship deck in both a research simulator [9] and a production (fleet) pilot trainer [10]. In the CDI approach, a ship-airwake database is used to initialize a vortex particle representation of the disturbance field. The Lagrangian vorticity-velocity form incompressible Navier-Stokes equations - the vorticity transport equations - are then solved to determine the mutual interaction of the rotorwake, the ship airwake and the fuselage. Like the work in [17], only limited qualitative metrics were presented, with comparisons made only to the baseline CASTLE model. While the work in [9, 10, 17] represents a significant advance in demonstrating what is feasible with regards to FD simulation for DI applications, these efforts did little to answer the key fundamental questions about aircraft response outlined above.

Unfortunately, while there have been significant strides in advancing aeromechanics modeling in recent years, the fundamental understanding of the response of rotorcraft to disturbance fields has advanced little since the early 2000s. The following quote from Whitehouse and Brown [6], dated 2004, remains just as pertinent today as it did then.

“Unfortunately, the wide variation in modeling techniques employed in the various published studies and the absence, in most cases, of any systematic variation in model parameters has made it difficult to organise the literature into a mutually-consistent body of evidence from which any general conclusions regarding the safety implications of helicopter encounters with aircraft wakes can be inferred. The situation is compounded by a singular lack of data against which these computations can be validated, and thus the possibility of model-dependence in some of the published results cannot be excluded. In particular, in most previous studies the mutually-induced distortion of the wakes of the rotor and the interacting aircraft has been neglected, yielding the so-called ‘frozen vortex’ assumption. This assumption has generally been adopted as a matter of computational expediency, but where the wakes interact over relatively long timescales, its validity must be called into question. Such is the case when the helicopter is operating at the low forward speeds that are characteristic of terminal flight operations.”

Indeed, it is exactly this need that the proposed effort seeks to address. Now, with rotorcraft aerodynamics prediction methodology in a flight dynamics environment at the point where both the disturbance field, the aerodynamic/wake model and the rotorcraft configuration can be varied in a systematic manner, courtesy of the recent CDI ONR effort (N00014-14-C-0014 – see [34,

35]), we are in a position to finally answer many of the fundamental handling qualities and flight dynamics questions outlined above.

## **Methods, Assumptions, and Procedures**

### *Technical Objectives*

The goal of the effort is to develop a fundamental understanding of the relationship between the length and time scales typically present in disturbance fields experienced by Naval aviators (i.e. ship airwakes, wing wakes etc.) and rotorcraft FD and FQ when the fully-interacting fluid dynamics of the airwake and rotor wake and flight mechanics are accounted for. With this information in hand, the engineering community would be better able to understand the relationship between aircraft type, ride quality, FD and FQ during flight conditions where disturbance fields are encountered. The community would be able to more accurately define trainer requirements, minimum experimental campaign requirements, minimum CFD modeling requirements and consequently establish a benchmark to evaluate CFD predictive capability. Furthermore, the conclusions of this work would also directly impact the development of requirements for new aircraft given the direct correlation between FD and FQ. The proposed effort would undertake the research required to develop such an understanding, with disseminating the observations and conclusions of the work to the Navy and the broader FD, FQ and handling qualities communities - a key objective from the outset. The effort will be structured using a build-up approach that first focuses on defining relevant disturbance fields followed by predicting and understanding the fundamental aeromechanics response (i.e. aerodynamic forces and moments and rotor dynamics). The effort would culminate in predicting the flight dynamics and handling qualities for realistic, but generic, helicopter and tiltrotor configurations that include representations of flight controls, propulsion system and cross-coupling characteristics. The key objectives for the effort roughly form the main tasks and are as follows:

1. Define the spatial and temporal fluid dynamic scales present in relevant disturbance fields and develop numerical representations for testing (Year 1).
2. 6-DOF generic model assembly and shakedown testing to ensure correct operation and functionality.(Year 1)
3. Define aeromechanics performance and HQ metrics along with a detailed simulation test matrix that includes systematic and consistent model fidelity build-up (Year 1-2).
4. Undertake simulation of a generic helicopter interacting with frozen and distorting disturbance fields to establish fundamental response characteristics (Year 2).
5. Undertake simulation of a generic tiltrotor interacting with frozen and distorting disturbance fields to establish fundamental response characteristics (Year 2).
6. Develop a realistic full helicopter model and undertake simulations of interactions with frozen and distorting disturbance fields to establish flight dynamics and handling qualities response (Year 3).
7. Develop a realistic full tiltrotor model and undertake simulations of interactions with frozen and distorting disturbance fields to establish flight dynamics and handling qualities response (Year 3).
8. Documentation and dissemination of observations and conclusions to the Navy and the wider FD/HQ community (Years 1-3).

The intention is that this effort be a natural, albeit rotor-centric, follow-on to the ONR sponsored work of Jones and Smith at the University of Maryland and Georgia Institute of

Technology respectively [1]. In their work, Jones and Smith are undertaking a combined experimental and numerical investigation of the aerodynamic loads induced on a wing by the wake shed from different canonical structures. The proposed effort seeks to leverage and complement that work by extending it to rotor systems and focusing on several fundamental unknowns related to how disturbance field spatial and temporal scales alter rotorcraft aeromechanics. A numerical approach is proposed because we feel that it is the most cost effective way to interrogate the underlying physics so that critical understanding of both the rotor response and the level of modelling fidelity required can be established. With this information in hand, an understanding of the flight dynamics and flying qualities (gust suppression) can be developed that will impact simulator development, dynamic interface training, operations and ship-aircraft suitability.

As summarized earlier, a rotor system responds to a disturbance field in a fundamentally different manner to a wing, with the typical response being  $\sim 90^\circ$  out of phase to the disturbance and the amplitude filtered somewhat. There is a relative dearth of literature, and reproducible experimental data, related to the fundamental response of a rotor system to a large disturbance field, with studies either investigating full-up DI type scenarios (where the problem is so complicated that one is unable to unpack the fundamental physics from configuration specific nuances) or the type of disturbance field is generated by a relatively simple structure, but is difficult to model confidently and accurately in numerical methods, which results in an inability to differentiate the underlying physics from deficiencies/differences in the modelling choices made. For example, it would be convenient to use a cylinder, a cube or a backwards step to generate a disturbance field in a wind tunnel, but predicting/replicating the flow fields associated with these structures with CFD, even in the absence of a rotor, is technically challenging. Add in the need for an adequately instrumented rotor<sup>4</sup> and the prospect of undertaking a high quality experimental test that can produce reproducible and archival quality data is potentially expensive and daunting.

## *Task Overview*

### Definition of Disturbance Field Fluid Dynamics Scales Relevant to Naval Rotorcraft Operations

The temporal and spatial fluid dynamics scales associated with disturbance fields experienced by Naval aviators during both land-based and shipboard operations will be quantified. Characteristic properties will be determined in order to assist in the development of synthetic “surrogate” disturbance fields. The following discussion presents an overview of the types of flow structures present in a ship airwake that this task will seek to characterize and task 3 will represent.

It is well known that the ship airwakes contain a wide spectrum of spatial and temporal scales, with structures as large as the beam of the ship all the way down to very fine scale high frequency turbulent structures (see Figure 5). Far downwind of the ship, the airwake is composed of relatively weak chaotic structures associated with the turbulent breakdown of the vortical structures as they age. Near to the ship, and most notably for DI operations, near to landing spots, the flow can be critically different.

---

<sup>4</sup>Minimum required measurements are integrated loads at the hub, the zeroth and 1<sup>st</sup> harmonic blade flapping, in addition to flow diagnostics (flow visualization and possibly PIV)

The flow near to and over the hangar of a small-deck ship is dominated by the roll-up and shedding of a strong horseshoe-like vortical structure from the sides and roof of the hangar [36]. The induced flow over the deck causes the boundary layer near to the stern to roll-up into a pair of counter-rotating vortices aligned with the flow [36, 37]. A pressure difference between the sides and top of the ship can cause the flow around the bow of the ship to form a “wing vortex” like structure that wraps around the fore of the ship and becomes aligned with the flow downstream along the sides of the ship. In some cases, these vortices can be entrained into the flow-aligned pair of counter-rotating vortices trailed from the ship deck.

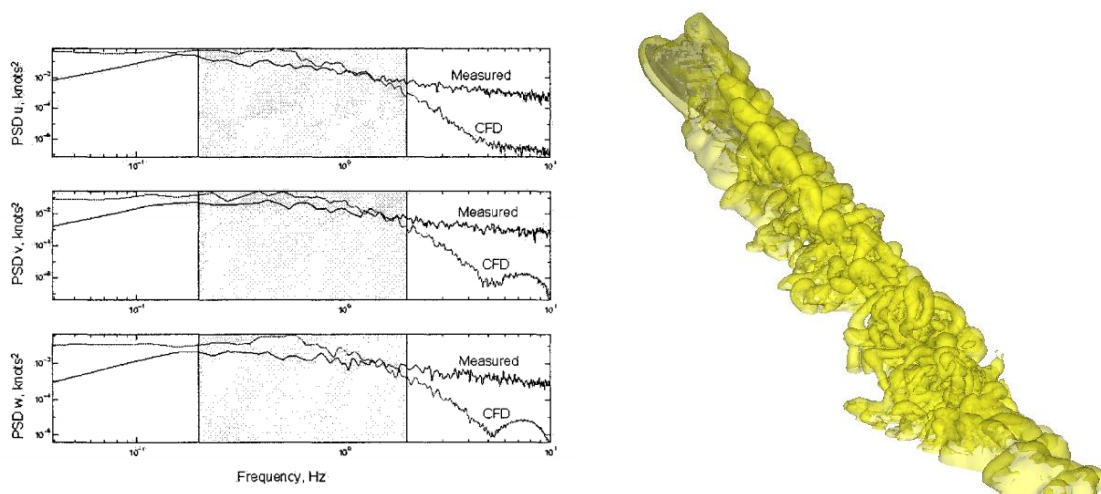


Figure 5: PSD of measured and computed velocity spectra for a ship airwake from [38] (left) and CDI CFD prediction of the wake behind a LPD-4 from [39]

The flow around flat-topped ships, such as aircraft carriers and amphibious assault ships, includes regions behind the superstructure that are grossly similar to those just described for small-deck ships, but also include “wing vortex” like structures trailed from the upstream deck edges. At zero wind over deck, the flow wraps up and over the leading edge of the deck forming a large separation bubble that periodically sheds vortical structures. Around the deck edge, and deck notches in the case of a carrier, wind-aligned “wing vortices” are trailed downstream. At non-zero wind over deck, the leading edge separation bubble is greatly diminished, though the “wing vortices” become more pronounced and can trail across the deck (and landing spots).

It should be noted, that the descriptions of the flow just given present a time-averaged perspective, and the actual flow field is highly unsteady with many of the afore-mentioned flow structures being shed from the deck at frequencies related to the flow speed and the characteristic dimension of the shedding structure (i.e. the ship’s beam, hangar height etc.). “Wing vortex” like structures, such as those trailed from the edge of a flat-top ship along the deck, are likely to be more stable given the tight, concentrated vortical structure.

#### Generation of Synthetic Disturbance Fields

A set of representative “synthetic” disturbance fields will be created. To aid in undertaking a systematic variation of the flow field in order to decompose the fundamental aeromechanics, FD and FQ responses, these fields could be as simple as a single (“wing” or “deck edge”) vortex, a set of vortices, or as sophisticated as a generic turbulent field with given statistical properties (i.e. atmospheric turbulence) or a CFD-generated ship airwake. It is anticipated that the numerical representations of the disturbances will be formulated as both velocity fields and distorting

vorticity fields compatible with the basic FD model, as well as CHARM Lagrangian and VTM2 grid based representations of the simulation model described below. Full 4D representations will be developed for the appropriate length of time. Solutions will be initialized with the full 3D disturbance field, and advanced by using the field as an upstream boundary condition. In the CHARM-based model (see discussion below), the field will be represented as vortex particles and will evolve according to the Lagrangian form of the vorticity transport equations as in [9, 10]. In the VTM2-based model [34, 35], the flow field will be represented on a 3D grid and will evolve according to the Eulerian form of the vorticity transport equations as in [3-7] (Figure 2), albeit now on an adaptive grid.

In many DI simulations the resolution of ship airwake disturbance fields focuses on the frequency range most associated with pilot workload (see shaded region on Figure 5 left), which is typically 1-10rad/s (0.2-2Hz). This is the frequency range at which pilots typically close the loop around the aircraft/flight control system, though there may be certain circumstances, like station keeping, where a pilot may be able to respond faster. Below this range, the response does not impact workload, and manifests as an essentially steady input. Providing loop closure at higher frequency is not physiologically possible, though such high frequency content will manifest as vibrations transmitted to the fuselage and thus ride quality.<sup>5</sup> In this effort, we seek to distinguish between the fundamental aeromechanics response of the rotor system, the aircraft and what the pilot can respond to; thus we will not filter any of the “synthetic” disturbance fields.

#### Definition of Aeromechanics and HQ Metrics

This task will see the definition of aeromechanics performance and FQ metrics along with a detailed simulation test matrix. Emphasis will be on establishing appropriate and reproducible metrics with which to evaluate the influence of a disturbance field on the aircraft’s force and moment response (Year 2) and FD and FQ (Year 3). Both can be related back to the ADS-33 design standard, though subjectiveness introduced by the pilot ratings, and indeed nuances associated with aircraft specific control systems and stability augmentation systems, can be difficult to isolate. The approach considered here is to rely upon the fundamental performance metrics, as in prior work in the context of multiple aircraft interactions [20, 23-25, 33], where the aircraft is trimmed and then the controls are held fixed. The resultant response is compared directly to the design standard specifications to quantify the severity of the response and the ability of the aircraft to reject the disturbance. Given the complexity of modern aircraft control systems and the role of the pilot, careful attention must be made to distinguish between fundamental aircraft performance (response) and subjective metrics that are aircraft and/or pilot specific.

One could develop single/multi-axis controllers that would function as a regulation/tracking “pilot” to adjust the controls to maintain a given maneuver, though while the control variation required to reject the disturbance could be compared to the design standards, this approach is a better test of the controller than of FD and FQ. Similarly, development of a formal pilot model with appropriate gains and crossovers to mimic a real-world pilot would require extensive validation and would still only produce subjective assessments.

In the analysis to be undertaken in the second year of the effort, only the aircraft’s fundamental response (aerodynamic forces and moments along with rotor dynamic response)

---

<sup>5</sup> Loads in flap are transmitted to the fuselage a multiples of N/rev (i.e. N/rev, 2N/rev, 3N/rev etc.), where N is the number of blades. Chordwise loads, that couple with the fuselage body modes, are transmitted at multiples of  $N\pm 1$ /rev (i.e.  $(N\pm 1)$ /rev,  $(2N\pm 1)$ /rev,  $(3N\pm 1)$ /rev etc.).

will be investigated to look at pure aeronautics without the complications of aircraft specific properties such as stability augmentation etc. since ultimately, it is the fundamental response that drives the aircraft's FD. In the third year of the effort, a more sophisticated engineering model will be developed to represent specific aircraft characteristics, such as cross-couplings between propulsion system and the rotor, as well as control systems. With this model in hand, the work undertaken in year 2 will be extended to include flying qualities.

#### Generic FD Model Assembly

In conjunction with the development of aeromechanics, FD and FQ metrics, the FD model for undertaking the simulations will be assembled and tested. Component system models will be validated as they are developed. For example comparing to Carpenter and Fridovich's data [40] for a rotor's response to a ramp collective input would demonstrate that both the rotor dynamics and the unsteady aerodynamic feedback between the impulsively shed wake from the change in collective, are adequately modelled). It is currently anticipated that the simulation will be based around the Navy's CASTLE 6-DOF FD framework (CDI will request permission from NAVAIR for its use in this project) coupled to the CHARM rotor, fuselage and ship airwake model – as in [9, 10] – and the VTM2 model developed under prior ONR sponsorship [34, 35].

This suite of FD models will provide the ability to take a systematic step-wise increase in capability within the same analysis to investigate the role that modeling fidelity plays on FD and HQ during an interaction with an external disturbance field. The anticipated levels of fidelity offered by the model are, in order of increased fidelity,

1. Finite state wake model coupled to a frozen disturbance field
2. Free wake model coupled to a frozen disturbance field
3. CFD wake model coupled to a frozen disturbance field
4. Free wake model coupled to a distorting (particle method) disturbance field
5. CFD wake model coupled to a distorting (CFD) disturbance field

While the VTM2 model, derived from the same model used by Whitehouse and Brown [3-7], and CHARM can be thought of as being comparable from a FD perspective, VTM2 is able to predict more accurately the natural coalescence and rupture of vortical structures in the wake and disturbance field than the hybrid filament-particle method in CHARM.

The test matrix to be investigated will be developed based upon analysis of the literature as well as discussions with ONR and Navy personnel. A limited number of relevant flight conditions will be selected, and the level of modeling fidelity used to represent the wake and the disturbance field will be varied, as above. At the very least, steady level flight, as investigated by Padfield and co-workers [20, 23-25], will be studied as it is convenient for mapping back to the ADS-33 standards; however other conditions, such as a generic approach may be considered.

#### Fundamental Helicopter Response

The analysis of the response of a conventional helicopter configuration to the series of disturbance fields will commence in the second year of the effort. This task will start with the assembly of a generic helicopter model, possibly based upon the "example helicopter" included with CASTLE, and the level of modeling used to represent the aircraft will be varied. It is currently anticipated that the following list, in order of increasing fidelity, will be considered:

1. Isolated main rotor (no representation of the airframe, and no tail rotor for the conventional helicopter)

2. Main rotor and tail rotor
3. Rotors and fuselage (lookup table)
4. Rotors and fuselage (panel method for the fuselage)

Given the 4 permutations of helicopter model and the 5 permutations of wake/disturbance field modeling to be evaluated for each wake disturbance field and flight condition of interest, the test matrix gets quite large, and thus data analysis will be an extensive part of this task. A detailed breakdown of the influence of each parameter on aeromechanics will be documented, and conclusions about aeromechanics response, the level of modeling fidelity required and ultimately which length and time scales associated with the fluid dynamics of disturbance fields influence aircraft aeromechanics will be made.

#### Fundamental Tiltrotor Response

This task will commence with the assembly of a suitable tiltrotor model and will culminate with undertaking the same series of investigations described above for a conventional helicopter. It is currently anticipated that significant differences in the aircraft response - most notably in roll - will be observed when compared to the helicopter investigated earlier.

#### Helicopter Flight Dynamics Response

This task will build upon the conventional helicopter configuration modeling investigated in year 2 to assemble a FD model of a complete aircraft. Specific properties, such as weight, control systems etc., will be selected based upon interactions and possible collaboration with Navy personnel. Once the model has been assembled, the test matrix of cases investigated earlier will be repeated. In addition to predictions and comparisons of the fundamental aerodynamics response, additional FD and FQ metrics will be evaluated. A detailed breakdown of the influence of each parameter on aeromechanics will be documented, and conclusions about flight dynamics, flying qualities, the level of modeling fidelity required and ultimately which length and time scales associated with the fluid dynamics of disturbance fields influence aircraft aeromechanics will be made.

#### Tiltrotor Flight Dynamics Response

The effort will culminate in the development of a complete FD model of a tiltrotor, complete with control system models and cross-couplings etc<sup>6</sup>, based upon interactions and possible collaboration with Navy personnel. The test matrix evaluated previously will be repeated for the tiltrotor and conclusions about FD/FQ, modeling fidelity and the length and time scales critical to FD/FQ will be made.

---

<sup>6</sup>Such as the cross-coupling between the throttle command, the engine, transmission and resultant collective in the V-22

## **Results and Discussion**

Work to date has focused on investigating the type of disturbance fields experienced by Naval aviators, and how to define the relevant primary flow structures and time scales. Initial work was summarized in the first annual report and below. Recent work has investigated how to parameterize the wake structures and how to use simpler models to represent them.

### *Disturbance Field Characterization*

#### Flow Characteristics

Properties of flow over two-dimensional obstacles are described first, followed by a description of characteristics over 3D obstacles. For 3D geometries, the direction of the incoming wind plays an important role on flow characteristics [41-43], and this influence will be discussed. In the context of DI environments, flat-topped ships have unique wake structures, and receive special attention in this document. The influence of ship motion on airwake features will be discussed as well.

#### *Backward-Facing Step (2D)*

Figure 6 illustrates the general flow features associated with a backward-facing step [44], where an unstable shear layer is formed when the upstream boundary layer separates at the sharp corner [45, 46]. This shear layer consists of concentrated vorticities with fluctuating velocity gradients that become increasingly more turbulent as the shear layer propagates downstream with increasing curvature towards the reattachment zone, where part of it is deflected upstream towards the recirculation zone (where the pressure is lower), and part of it is entrained downstream up to as much as 50 step heights [45].

In the recirculation zone, a primary larger “bubble” can be identified, where flow velocities directed upstream can reach up to 20% of freestream velocities, and where negative skin-friction coefficients have been measured in tests [45]. A secondary smaller recirculation zone can also result when 3D effects due to the presence of side walls are accounted for – see discussion below and in [47, 48].

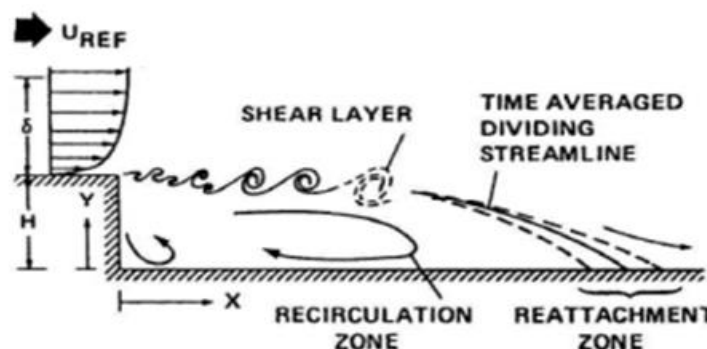


Figure 6: Flow topology over backward-facing step [4].

Flow in the reattachment zone is highly unsteady with coherent structures with length scales at least as large as the step height passing through this region [45, 49]. Coherent structures are large scale vortex structures that conserve their spatial features for prolonged durations. The location where the shear layer impinges on the wall fluctuates, shifting the reattachment zone upstream and downstream at low frequency [45, 46, 50, 51], with the primary recirculation bubble shrinking and expanding accordingly. Driver et al [44] suggest that this “flapping” motion results when part of the shear layer having greater momentum entrains more of the flow

downstream, resulting in less flow being diverted towards the recirculation region. This causes the recirculation region to shrink and increases the shear layer curvature, which, in turn, increases the adverse pressure gradient in the reattachment zone, forcing flow back into the recirculation region. The amplitude of this flapping motion is estimated to be approximately 20% of the shear layer width  $b$ , which is given by

$$b = \Delta U / (d\bar{u}/dy)_{max}, \quad (1)$$

where  $U$  is the mean velocity in the stream wise direction,  $\bar{u}$  is the streamwise velocity fluctuation, and the  $y$  coordinate is as defined in Figure 6 [44]. Driver et al further note that this low frequency motion does not have a strong impact on flow turbulence [44]. It should be noted that there is no consensus on the origin of the flapping motion in literature, for example see [49].

The mean reattachment length  $x_R$ , which represents the point of zero skin friction, is generally used to characterize the reattachment zone and can vary from  $4.9H$  to  $8.2H$ , where  $H$  is the step height, depending on the following parameters:

- a) Reynolds number of the initial boundary layer,
- b) boundary layer thickness,
- c) free stream turbulence, and
- d) blockage effects (for wind tunnel tests) [45].

In general,  $x_R$  is sensitive to the boundary layer Reynolds number only in the laminar and transition regimes (see Figure 7), and the effect of boundary layer thickness  $\delta$  is noted to be more pronounced when  $\delta/H \ll 1$  [45]. Eaton and Johnston [45] suggest that  $x_R$  decreases with high levels of freestream turbulence, and that its effect would likely be dependent on the spectrum of the turbulence.

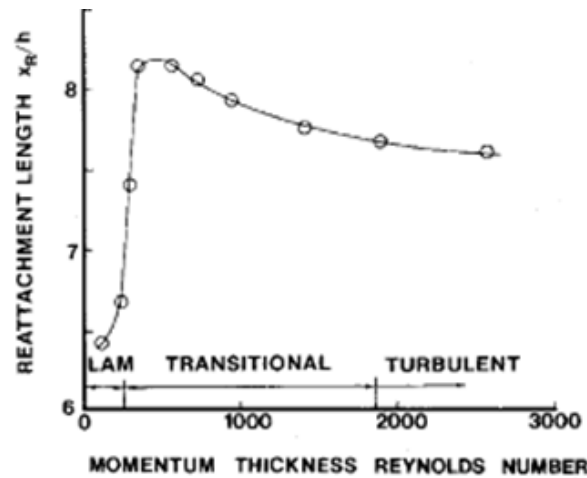


Figure 7: Influence of Reynolds number on reattachment length [45].

Eaton and Johnston [45] further show that maximum turbulence intensity  $I$ , defined as the ratio of root-mean-square of turbulent velocity fluctuations to mean absolute velocity

$$I = \frac{\bar{u}}{U_0}, \quad (2)$$

tends to be close to  $I = 20\%$ , and is typically observed approximately  $1H$  upstream of the mean reattachment point. Beyond  $1H$ , the maximum turbulence intensity decays rapidly.

Distribution Statement A

Approved for public release: distribution unlimited

Vortex shedding is characterized by the Strouhal number which is defined as

$$St = \frac{fb}{U}, \quad (3)$$

where  $f$  is the vortex shedding frequency and  $b$  is the characteristic length. Through spectral analysis of the pressure fluctuations at two streamwise locations  $x/H = 5.5$  and  $x/H = 8.0$ , Driver et al. [44] identify a dominant frequency of  $St = 0.2$ , which is comparable to the characteristic frequency noted in the spanwise vortical structures of free shear layers, and is consistent with shedding from a flat plate aligned perpendicular to the flow [52].

#### *Rectangular Prism (2D)*

In 2D flow over a rectangular prism, free shear layers are shed from the sharp edges of the cross-section as depicted in Figure 8 [53], where streamlines over four rectangular prisms with different width-to-length aspect ratios are shown. It is evident that aspect ratio influences the mean reattachment length, and for aspect ratios 1:3 and 1:4, the mean flow reattaches to the lateral sides of the rectangle. The resulting mean (time averaged) flow features are also symmetric. The Strouhal number for vortices shed by rectangular prisms can vary from 0.1 to 0.18 depending on the aspect ratio and Reynolds number [54].

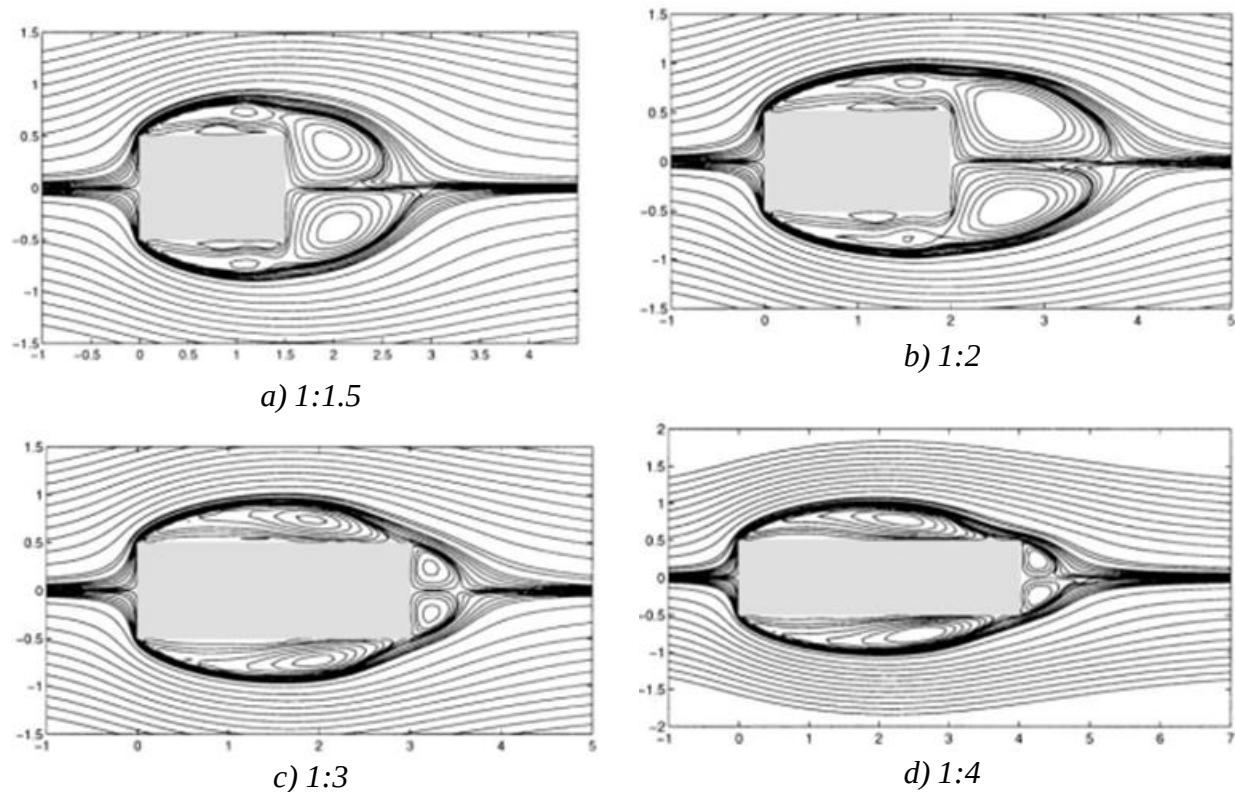


Figure 8: Streamline of mean flow around rectangular prisms with different aspect ratios [53].

#### *Surface-Mounted Cuboid (3D)*

Flow over 3D obstacles, such cuboids, introduce additional complexity relative to the 2D case as end and side wall effects become important [55, 56]. Flow features around a surface-mounted cuboid are depicted in Figure 9 and Figure 10 [55]. Upstream of the obstacle, horseshoe vortices form that extend over the sides of the obstacle and interact with the

downstream separation region, resulting in high flow unsteadiness [57]. An arch vortex forms on the leeward side of the obstacle, and flow detaches at the top edge of the wind-facing surface of the obstacle and reattaches downstream of the arch vortex. The mean reattachment length in 3D is shorter than that noted for 2D backward facing step due to side wall effects. Note from Figure 9 that time-averaged flow features are symmetric.

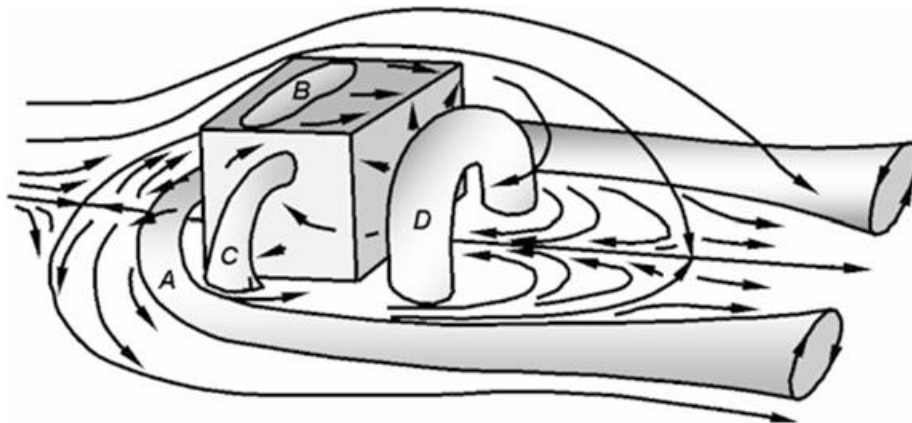


Figure 9: Schematic of principal flow features around surface-mounted obstacle [55].

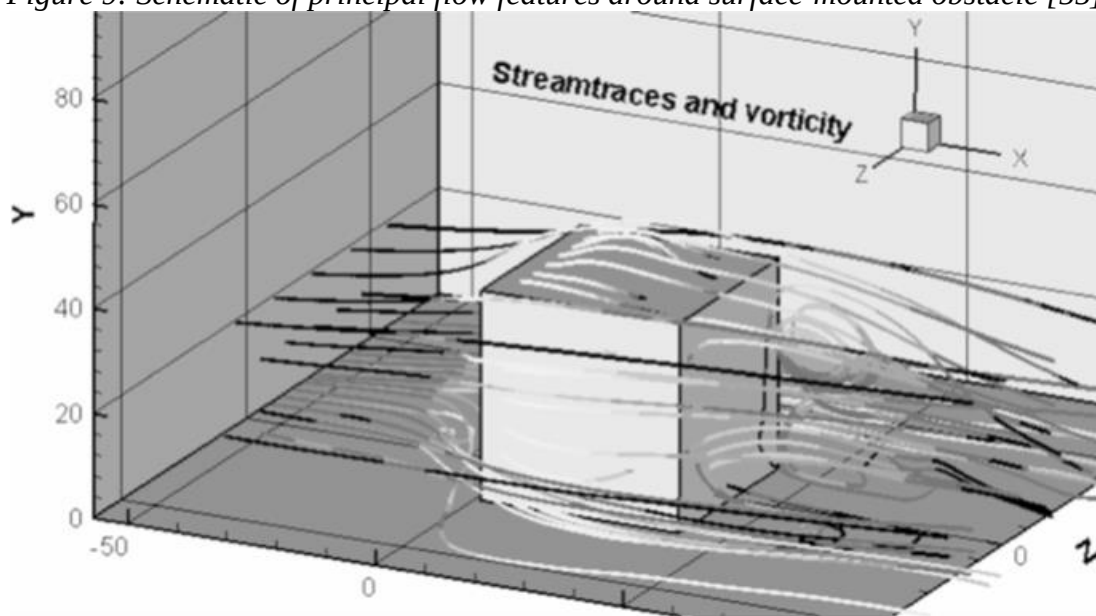


Figure 10: Perspective view of streamlines around surface-mounted obstacle [55].

Similar flow features are noted with 3D rectangular prisms [58], though they are sensitive to aspect ratio [37]. If the height is less than the length and width, the flow reattaches on the top surface of the prism, and up to three recirculation regions are observed, as shown in Figure 11 [58]. The location of the reattachment point on the top surface and behind the prism varies in the spanwise direction as depicted in Figure 12 [58]. Herry [37] notes that the location of the reattachment point is sensitive to the obstacle length-to-height aspect ratio, width-to-height aspect ratio, and Reynolds number. There is a minimum Reynolds number  $Re_c$  above which flow features become independent of Reynolds number for bluff bodies with sharp edges [42, 59, 60], however, a general rule for obtaining  $Re_c$  based on geometrical properties does not seem to have been established [59]. Not surprisingly, however, the mean velocity and turbulence of any

upstream atmospheric boundary layer also influences downstream flow turbulence levels [42, 60, 61].

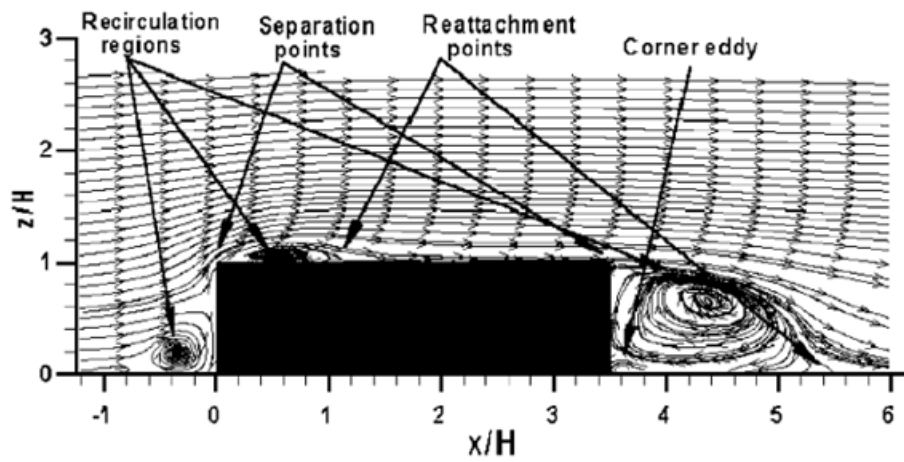


Figure 11: PIV derived streamlines at center plane around surface-mounted obstacle [58].

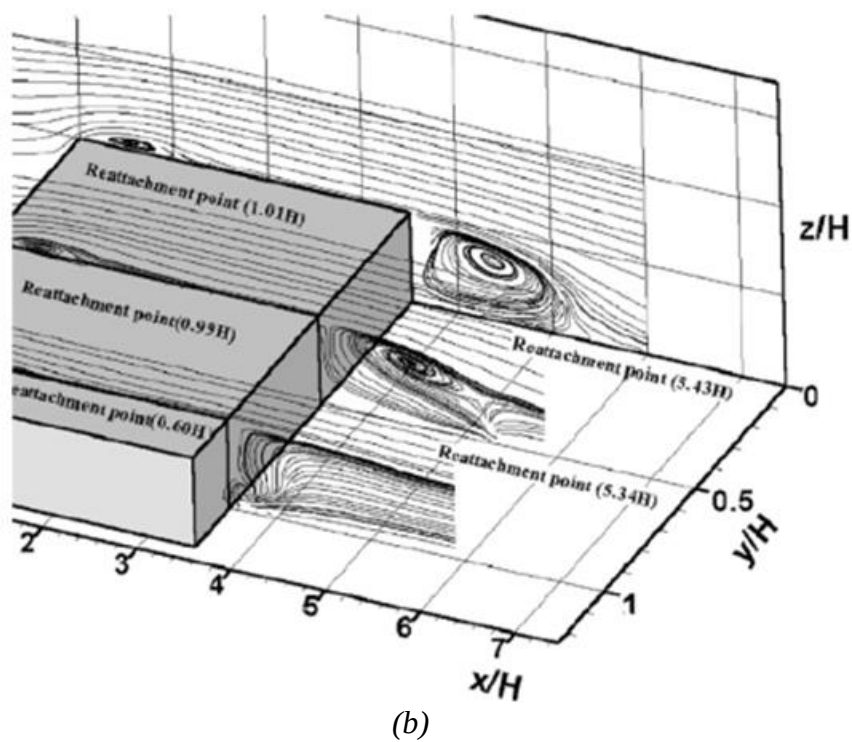
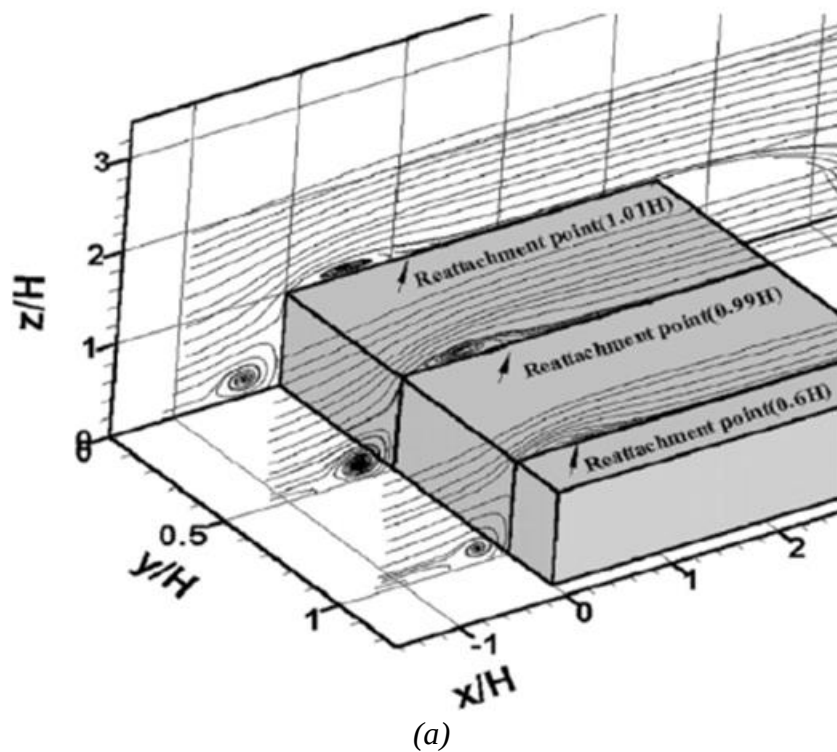


Figure 12: Streamlines of flow over surface mounted obstacle seen from (a) windward and (b) leeward directions [58].

An important parameter used to quantify flow turbulence is the turbulent kinetic energy (TKE) defined as [58]

$$TKE = \frac{1}{2}(\overline{u^2} + \overline{v^2} + \overline{w^2}), \quad (4)$$

where,  $\bar{u}, \bar{v}, \bar{w}$  represent fluctuating streamwise, spanwise and vertical velocity components. TKE can be related to turbulence intensity, introduced previously, as:

$$I = \sqrt{\frac{2}{3}TKE} \quad (5)$$

Kim et al. [58] observed peak TKE levels on the top surface of a  $3.5H$  long rectangular prism at  $x \approx 0.5H$ , see Figure 13. Becker et al. [42], on the other hand, examined flow over a rectangular prism with a shorter length of  $0.25H$  and observe peak turbulence levels at  $x = 3H$ , as depicted in Figure 14, demonstrating the influence of aspect ratio on flow characteristics. Their spectral analysis further indicates dominant frequencies at  $St_H = 0.15$ , with little sensitivity to Reynolds number and the width-to-height aspect ratios (1:2, 1:3, 1:4 were considered) [42]. Note that this Strouhal number is higher than  $St_H = 0.104$  noted for flow over cubes [57]. The subscript “H” indicates that the obstacle height was used as characteristic length in computing the Strouhal number, see Eq. (3).

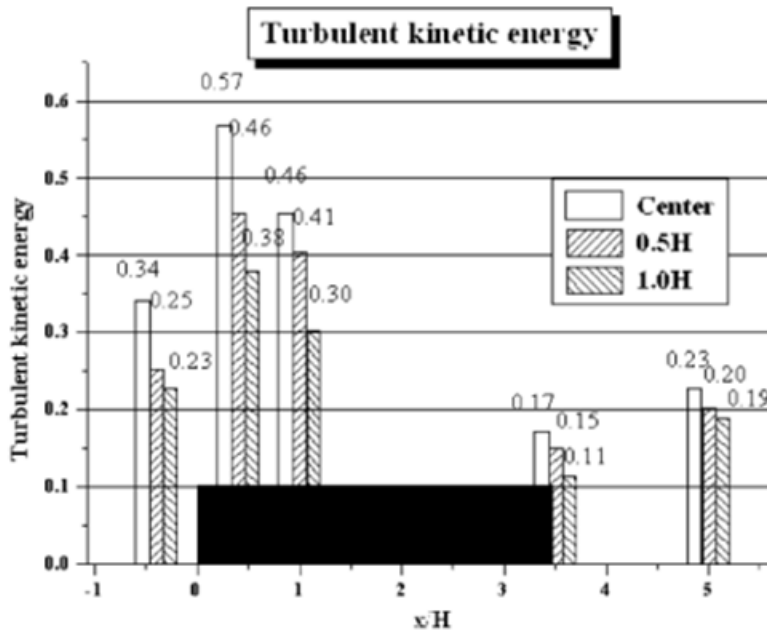


Figure 13: Turbulent kinetic energy peaks at different locations for flow over a 3D rectangular prism from Ref. [58] (boundary layer thickness of  $\delta = 0.06H$  and Reynolds number (based on step height) of  $7.9 \times 10^3$ )

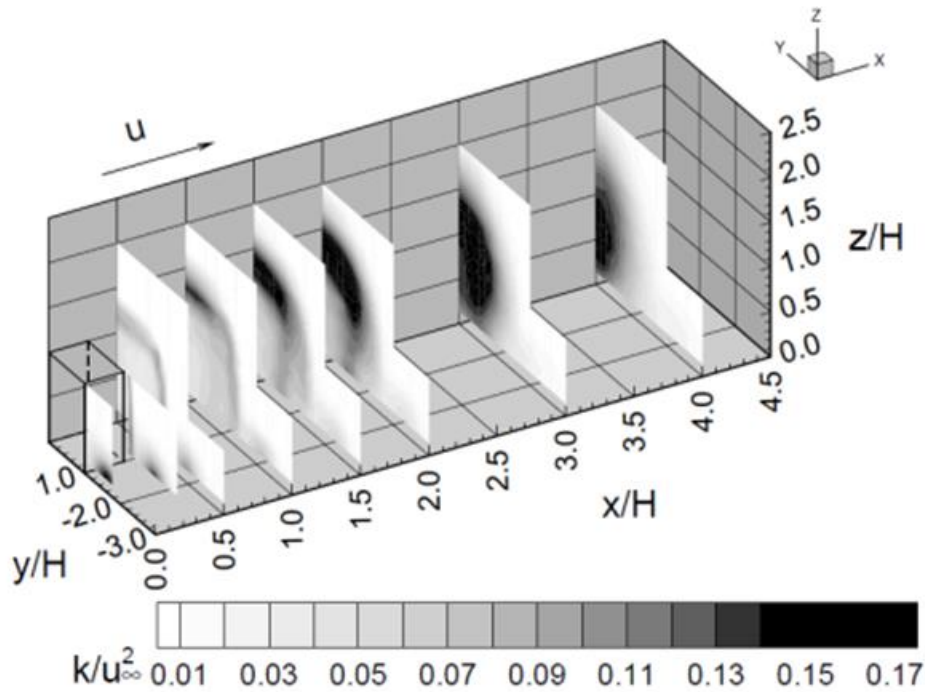


Figure 14: Turbulent kinetic energy at different locations for flow over a 3D rectangular prism from Ref. [42].

#### Small-Deck Ships (3D)

The double-backward facing step geometry, depicted in Figure 15 along with prevalent flow features [36], closely matches the aft region of typical small-deck aviation ship geometries and is thus of particular interest. Tinney and Ukeiley [36] derived the flow features in the figure using both PIV and oil-flow visualizations at  $Re = 9000$  for the ship geometry shown in Figure 16. The flow features in Figure 15 are similar to those for flow over a surface mounted cube shown in Figure 9, with some notable differences. In the flow aligned case, the upstream horseshoe vortex seen in Figure 9 is no longer present in Figure 15 due to flow reattaching on the walls of the structure before the first step. Streak lines that start along the edges of the ship structure from the nose curl up into vortex tubes, denoted as “wing vortex” in Figure 15, due to pressure gradients between the side and top surfaces of the structure and interact with flow downstream of the reattachment region to produce counter-rotating vortex tubes on both sides of the streamsurface bifurcation point on the figure.

Although Figure 15 depicts a symmetric upside-down arch vortex aft of the first step, this structure is unsteady and at any instant in time is asymmetric, see Figure 17 from [37]. This is a consequence of bi-stability of the flow, which causes the resulting asymmetric flow features to switch intermittently between portside and starboard locations. The influence of this asymmetry on helicopter-shipboard operations remains to be examined.

Tinney and Ukeiley [36] observe peak longitudinal turbulence intensity near the recirculation zone at  $1H \leq x \leq 2H$  downstream of the first step (see Figure 18a) and peak vertical turbulence intensity closer to the reattachment zone in the  $2H \leq x < 4H$  region (see Figure 18b). Turbulent kinetic energy is dominated by longitudinal velocity fluctuations, and peak TKE is observed in  $1H \leq x < 2.5H$  region in Figure 18d. Note that the TKE was computed using only longitudinal

$\bar{u}$  and vertical  $\bar{v}$  components. The longitudinal and vertical turbulent intensities in the figure are noted to be greater than those observed with 2D obstacles [36]. The dominant frequency was characterized by a Strouhal number of  $St_H = 0.17$  [36].

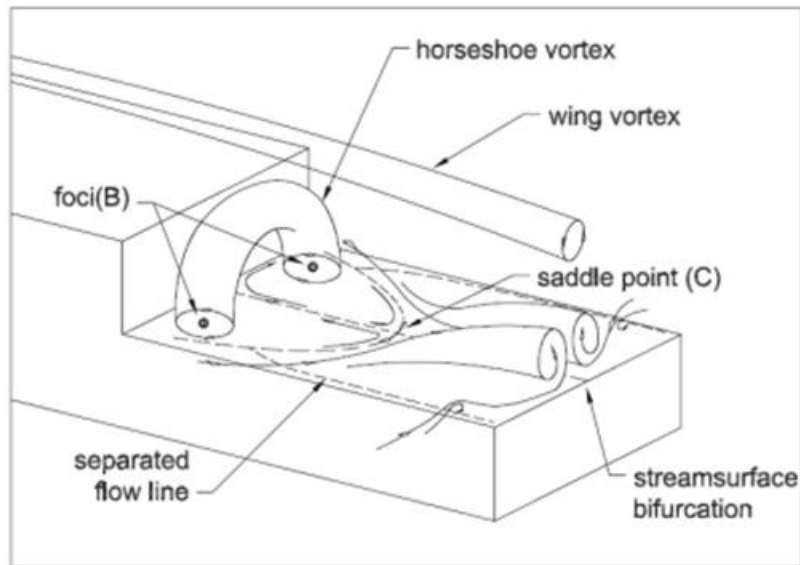


Figure 15: Flow feature for a 3D double-backward facing step [36].

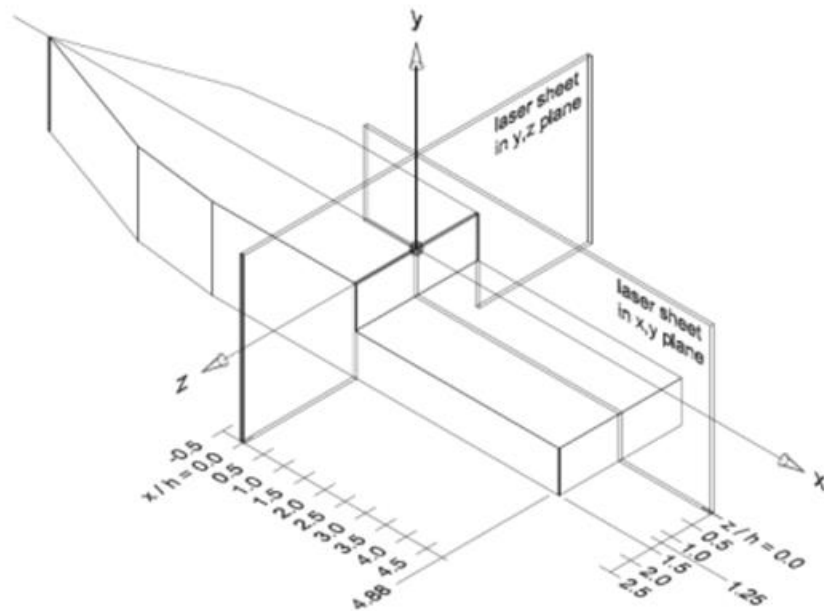


Figure 16: Ship geometry used to derive flow features for a 3D double-backward facing step [36].

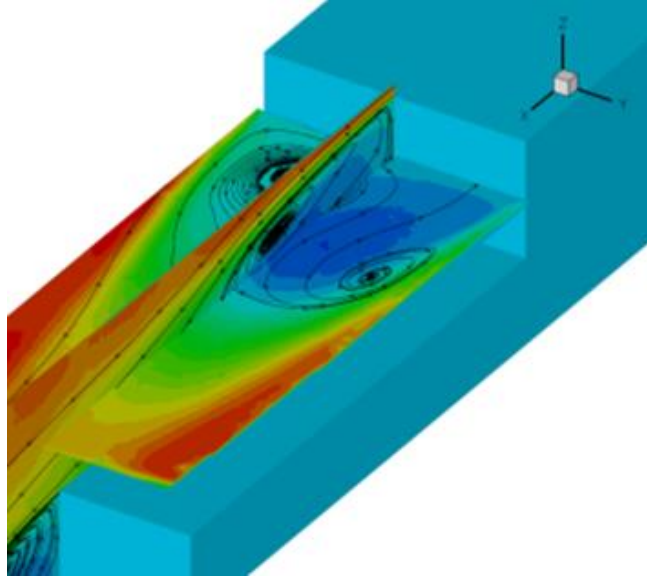


Figure 17: Asymmetry in arch vortex behind 3D double-backward facing step [37].

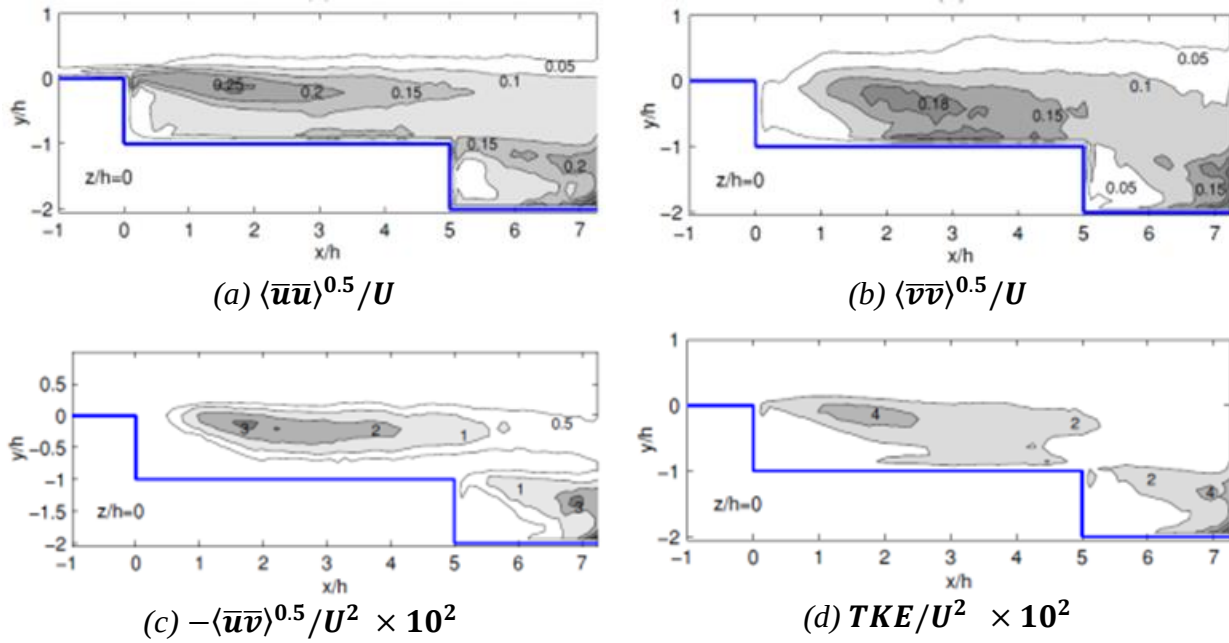


Figure 18: Contours of turbulence parameters for a 3D double-backward facing step [36].

### *Flat-Topped Ships (3D) and Wind-Over-Deck*

Flat-topped ships, such as aircraft carriers and Landing Helicopter Assault (LHA) ships (e.g. Figure 19), are characterized by sharp-edged flat decks and superstructures located towards the starboard side of the ship. While helicopter landing spots are distributed across the entire deck, the fixed wing landing zone typically takes up most of the port side of the deck.

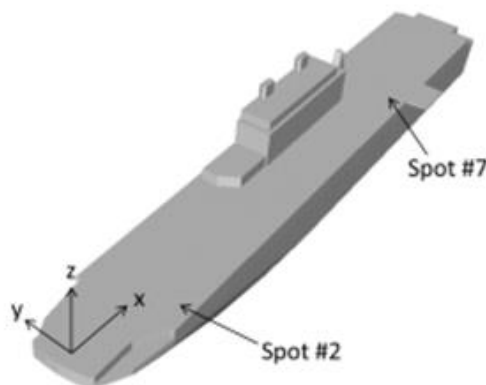


*Figure 19: LHA ship geometry with landing spots #2 and #7 highlighted, adapted from Fig. 1b in Ref. [62].*

A unique aspect of the flow characteristics around such configurations is the existence of “wing-vortex” like structures that originate and propagate from the edges of the ship structure, denoted as “deck-edge vortices” in Figure 20 [62]. From Figure 20a, headwinds produce a large separation at the bow, though strong coherent wind aligned deck-edge vortices also form. These vortices are generated due to pressure differences between the side and top surfaces of the deck. Vortical structures denoted as “bubbles” in Figure 20a shed periodically from the bow towards the superstructure at frequencies of 0.2 – 0.3 Hz [62]. The interaction between the superstructure, bubbles, freestream, and deck-edge vortices produces the highly turbulent region over and aft of the superstructure.

The airwake in the lee side of ship is highly unsteady and separated, producing downwash fields that can cause fixed-wing aircraft to bolter during landing, resulting in a touch-and-go maneuver on the deck and requiring an additional landing attempt. This region of high turbulence is known as the “bubble,” (not to be confused with the bubble structures shed from the bow) and the associated loss of altitude and deviation from desired glideslope experienced by pilots during landing is known as the “bubble effect” [63]. The ambient wind interacting with the ship structure, combined with the ship velocity, is referred to as “wind-over-deck” (WOD). For a starboard WOD condition at 30° (measured clockwise relative to the ship longitudinal axis), no bow separation or bubble structure is present in Figure 20b. The starboard deck-edge vortex rolls onto the deck and spreads spanwise over the deck while the portside one detaches from the ship and is blown over the ocean, with significant flow separation over the superstructure.

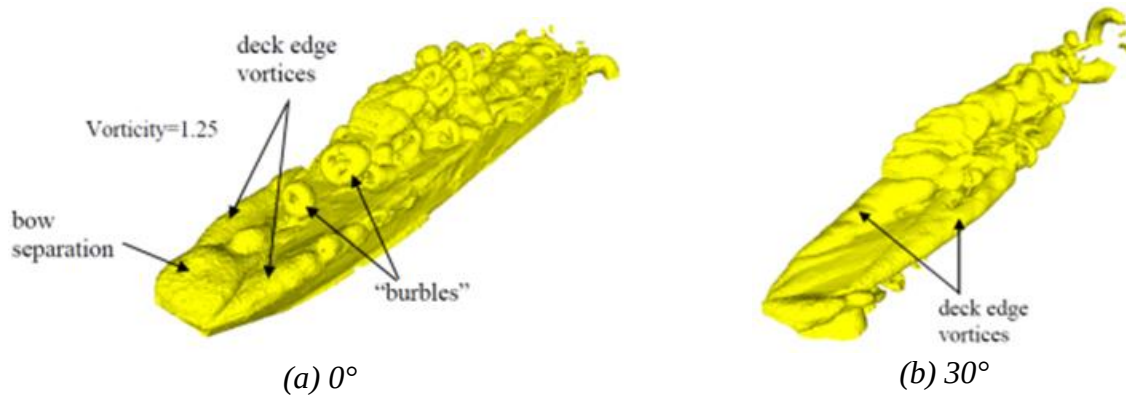


Figure 20: Snapshot of vorticity iso surfaces for a 30 kts WOD from Ref. [62]

Figure 21 shows instantaneous iso-surfaces for the headwind and 30° starboard WOD conditions from Ref. [64], where both time-accurate and time-averaged flow conditions over a full-scale LHA ship geometry were examined using the PUMA2 solver. In Figure 21b, a vortical coherent structure from the starboard edge that runs diagonally across the fore of the ship is evident for the 30° WOD condition. This vortical structure results in lateral velocities of approximately -11 m/s over spot #2 (see Figure 22a [64]), which is 40% greater than the freestream value of -7.7 m/s ( $-15.43 \sin 30^\circ$  m/s). The largest velocity fluctuations were noted in the time-histories ~6 m above spot #7 for the 30° starboard WOD condition, see Figure 22b [64]. From spectral analysis of longitudinal and vertical velocity time-histories, Sezel-Uzol et al. note peak frequencies at 0.1 Hz ( $St = 1.3$ ) and 0.2 Hz ( $St = 2.6$ ) over spot #2, and a peak frequency of 0.5 Hz ( $St = 6.5$ ) over spot #7 for both headwind and starboard 30° conditions. Note that Strouhal numbers were not computed in Ref. [64] and are provided here using the full-scale beam width of 31.8 m as characteristic length and the free stream velocity magnitude of 15.43 m/s.

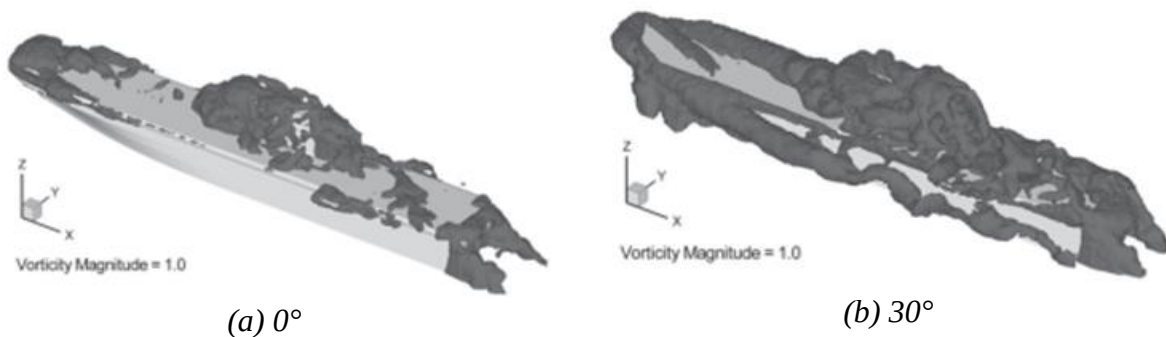


Figure 21: Snapshot of vorticity iso surfaces for a 30 kts WOD from Ref. [64]

The influence of WOD angle on longitudinal  $u$  and vertical  $w$  velocity airwake components are shown on the time-averaged flow contour plots for a plane 6 m above the deck in Figure 23 and Figure 24, respectively [64]. From the figures, oblique and beam WOD conditions produce the greatest fluctuations in velocity components. For beam winds, regions with negative longitudinal velocities are noted in the fore of the ship in Figure 23. The sharp edges of the deck and the superstructure generate regions with reduced  $u$  velocity magnitudes in the oblique conditions. The contours in Figure 24 reveal regions with significant negative vertical velocities (up to -6 m/s) in the fore of the ship, which may adversely influence rotor inflow during piloted

operations. These contours also indicate large upwash velocities of approximately 15 m/s on the windward edges of the ship, as flow rolls over the edges. It should be noted that the study by Sezer-Uzol et al. [64] did not account for atmospheric boundary layer (ABL), which is needed for better predictions of turbulence intensities at full-scale [65], however, it is not anticipated that including the ABL would fundamentally alter the primary flow structures in the wake. Rather, the ABL would serve to introduce additional frequency components, and likely accelerate the breakdown of bluff structures. In an accompanying paper [66], instantaneous vorticity magnitude plots for the PUMA2 based simulation of flow of the LHA ship were provided (see Figure 25). The deck-edge vortices are noted to have magnitudes in the  $0.7 - 2.0 \text{ sec}^{-1}$  range for the headwind condition and peak magnitudes of  $3.0 \text{ sec}^{-1}$  in the starboard  $30^\circ$  WOD case.

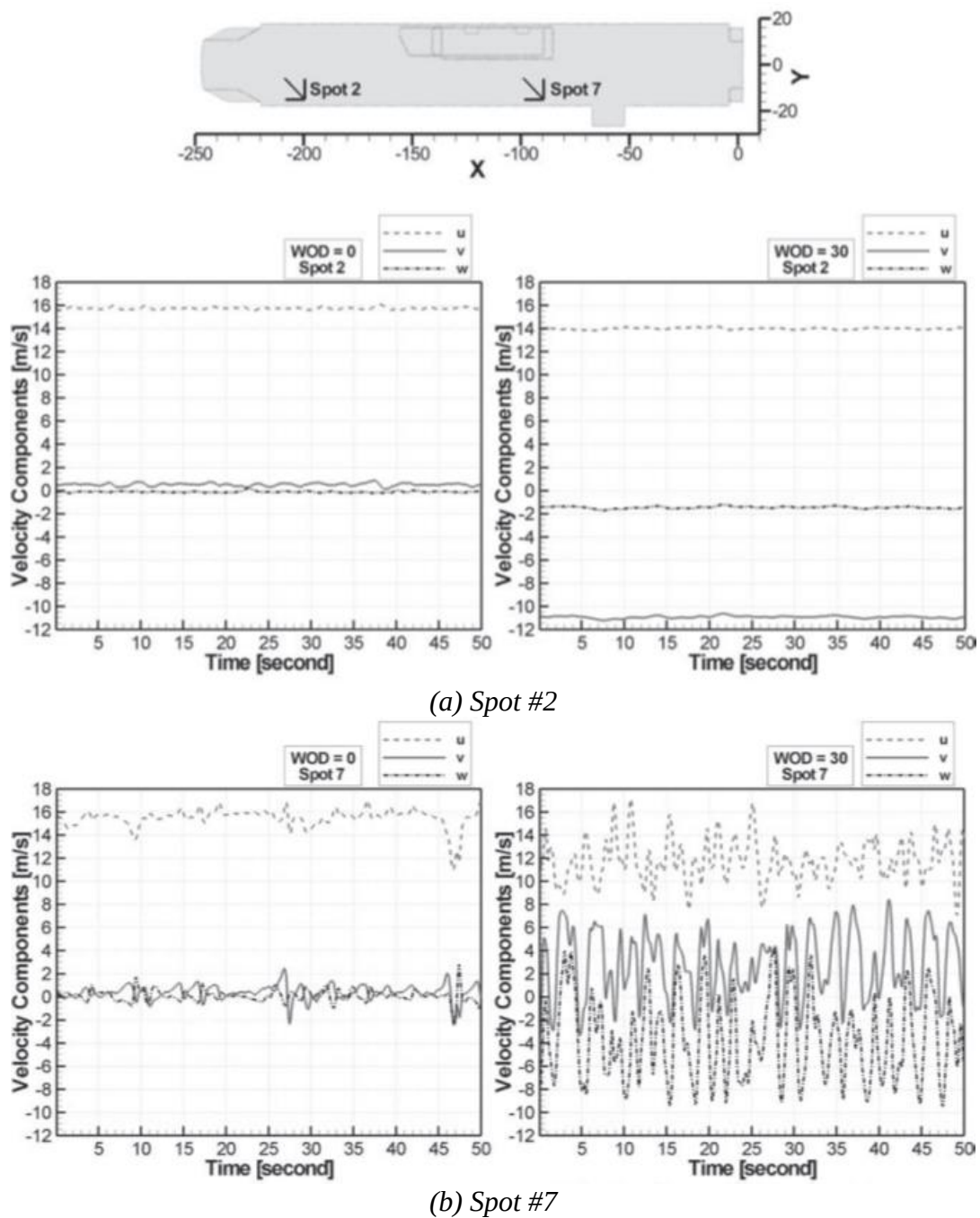


Figure 22: Time histories of velocity components for 30 kts WOD 6m above landings spots #2 and #7 [64]

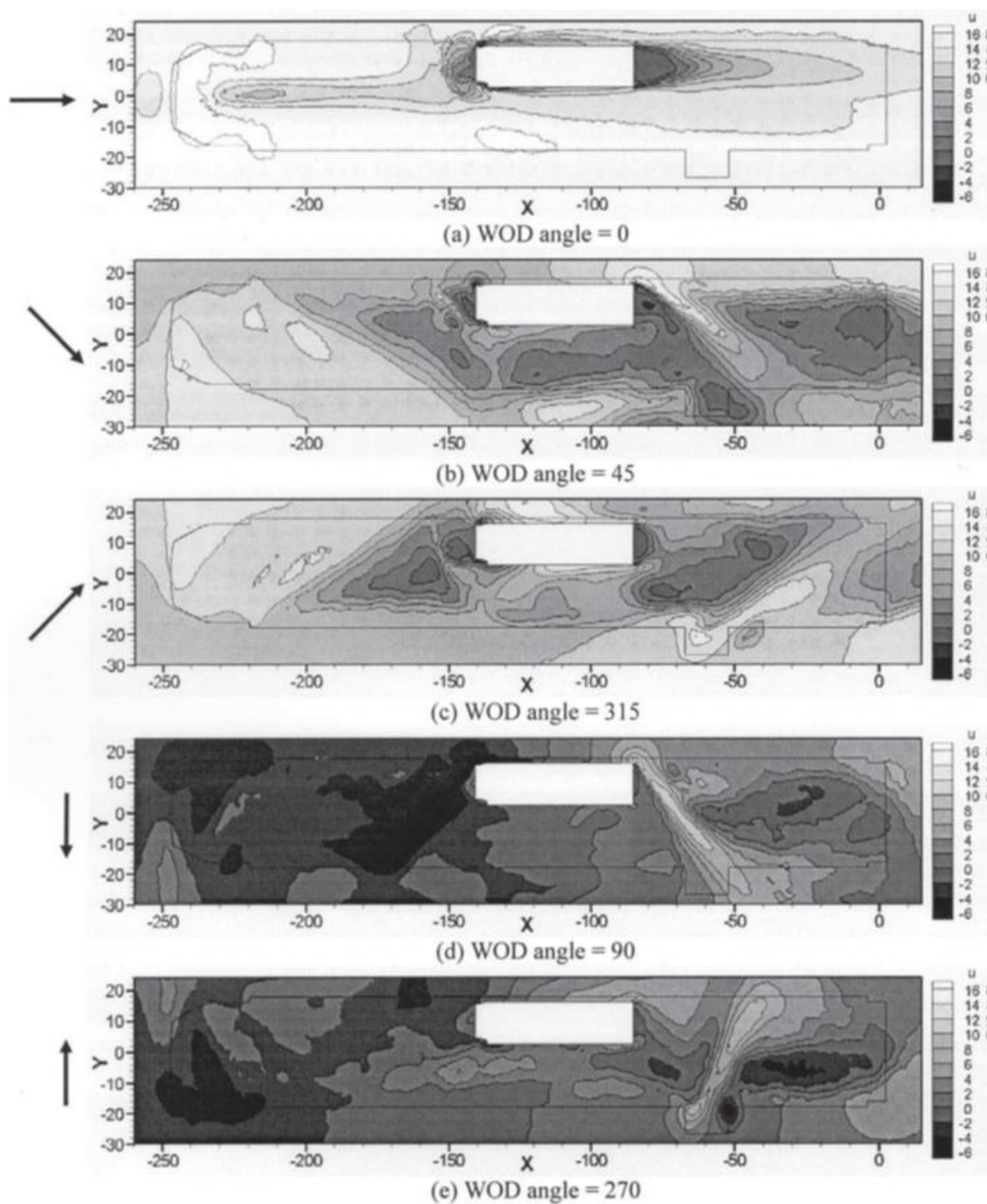


Figure 23: Time-averaged contours of longitudinal velocity for different WOD conditions [64]

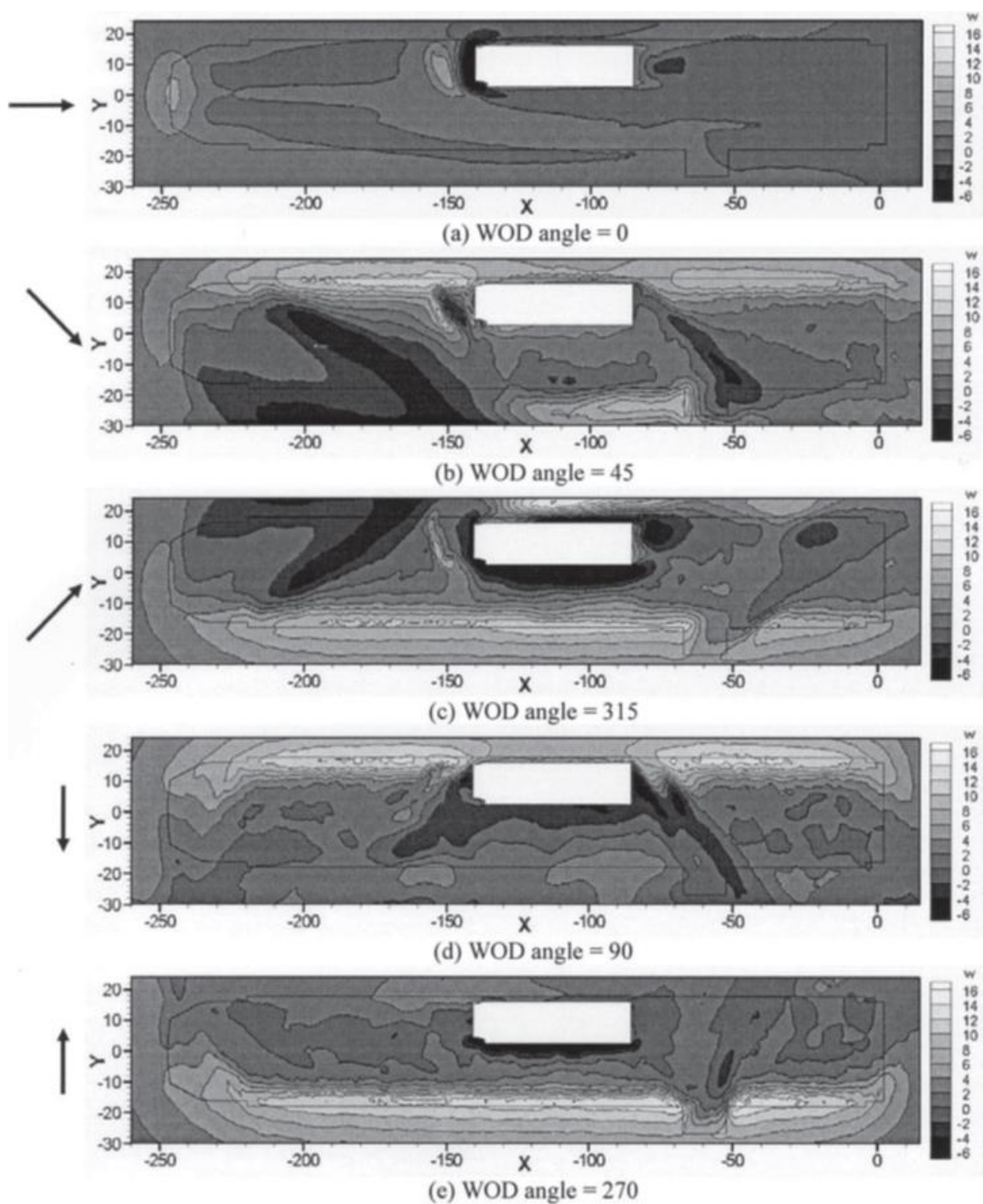


Figure 24: Time-averaged contours of vertical velocity for different WOD conditions [64]

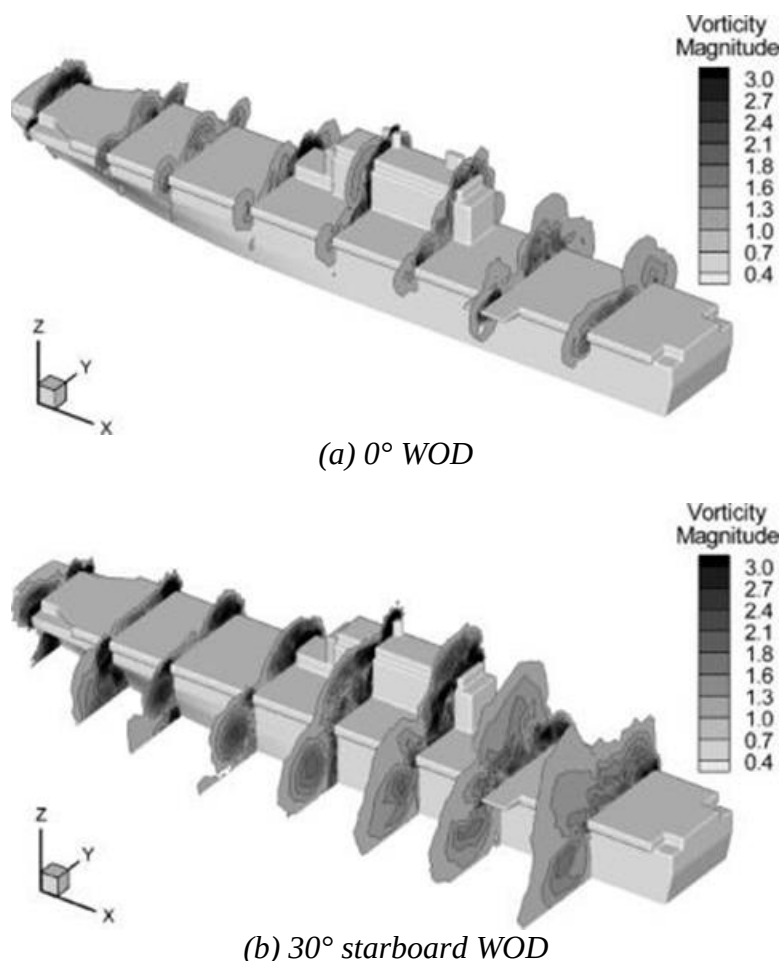


Figure 25: Instantaneous vorticity contour magnitude across ship deck [66]

Although CFD predictions provide valuable insight into ship airwake characteristics, they can only be considered reliable when validated against experimental data (model or full-scale) since factors such as solution type (RANS, DES, LES), time step size and mesh resolution can influence predictions [67]. Wind tunnel experiments are easier to setup, control and interrogate than full-scale tests, which are sensitive to changes in at-sea conditions, but usually fail to replicate full-scale Reynolds number due to tunnel limitations. This has been generally considered acceptable since the wakes shed from the sharp edges of ship decks and superstructures tend to be Reynolds number independent [62]. Turbulence in the wake at wind-tunnel scale experiments also tend to occur at much higher frequencies than full-scale tests [62], but the cause of this is not entirely clear. Polsky and Bruner in Refs. [62] and [68] compare time-averaged and steady-state CFD solutions to wind tunnel data obtained using a  $1/120^{\text{th}}$  scale model of the LHA ship for a headwind condition. Only time-averaged measurements were made during the experiment. Velocities along the data plane shown in Figure 26 are shown in Figure 27 for  $z = 0.2315$  in (equivalent to  $z = 2.315$  ft. at full-scale) [68]. Time-averaged predictions from CFD were obtained from time-accurate simulations. Corresponding standard deviation bounds are also plotted in Figure 27 to highlight velocity fluctuations in the time-accurate solution.

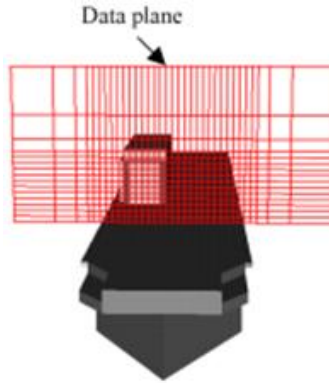


Figure 26: Plane considered for CFD comparison with experimental data in Ref. [62]

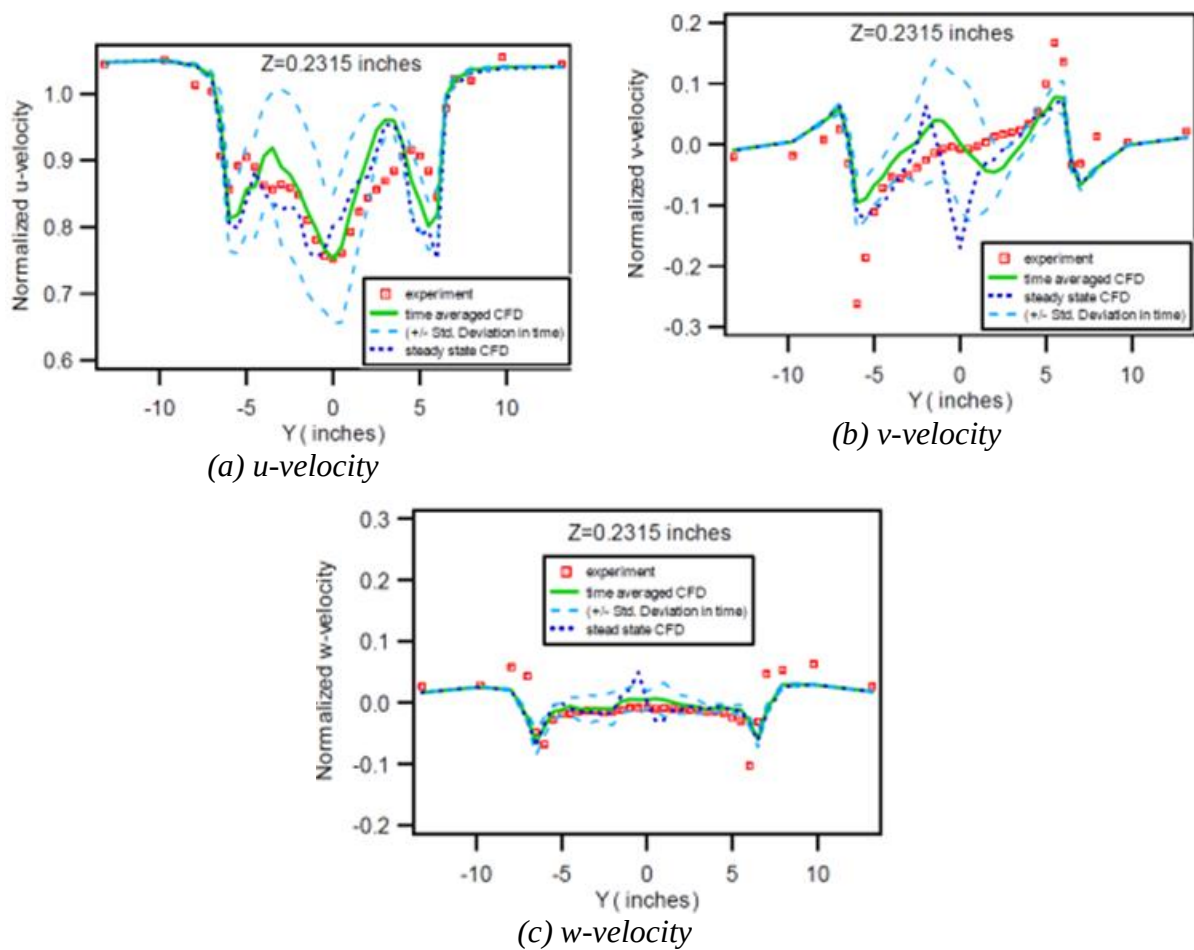


Figure 27: Comparison of velocities from CFD with wind tunnel data,  $z=0.2315$  in [68]

Discrepancies between CFD predictions (with standard deviation) and experiments are noted near the edges of the deck at  $y = \pm 5$  inches in the lateral and vertical velocity components, potentially due to deck-edge vortex effects not being modeled properly in CFD. Time-averaged velocity predictions for  $z = 0.7315$  in (equivalent to  $z = 7.315$  ft. full-scale) are shown in

Figure 28 [62]. Similar offsets are evident in the lateral and vertical velocity components near the edge locations at  $y = \pm 5$  in.

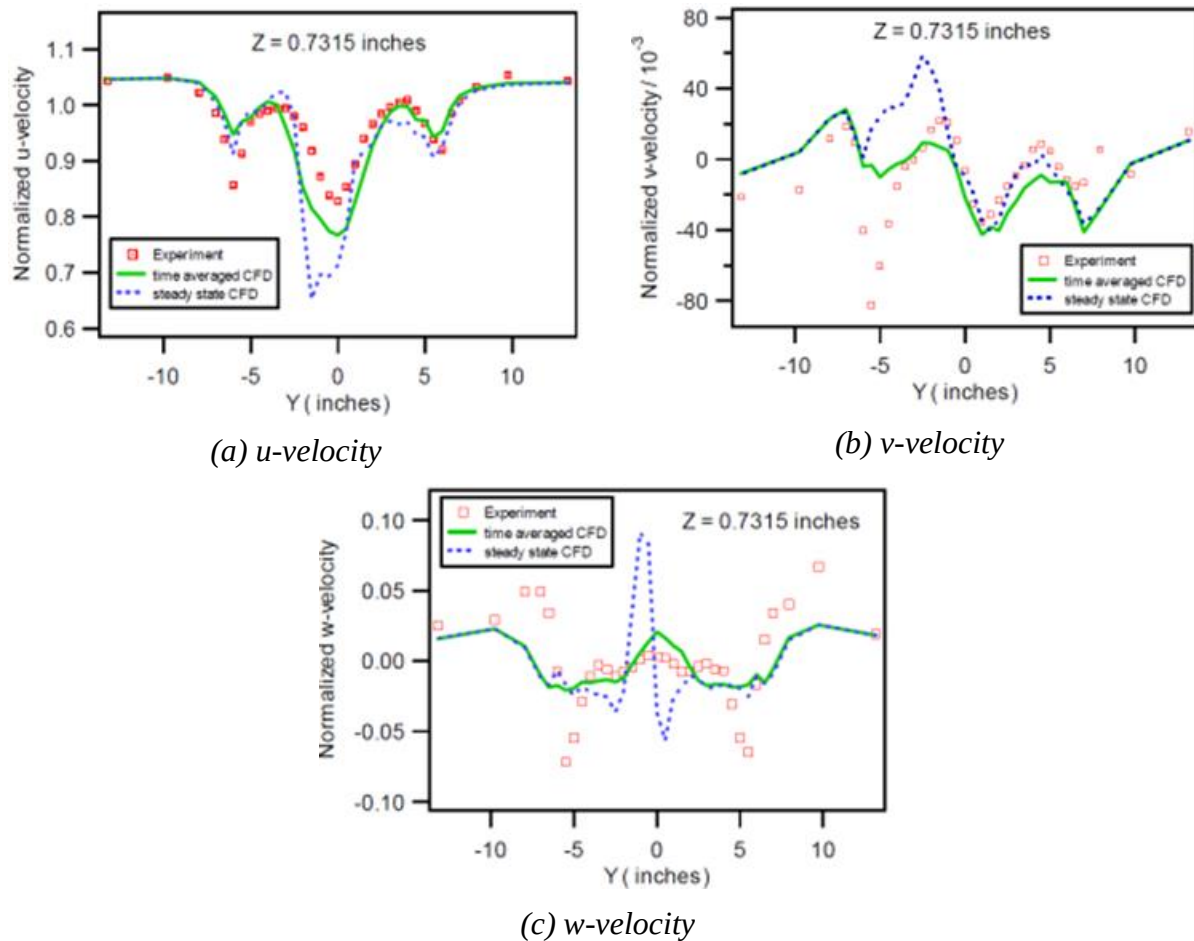


Figure 28: Comparison of velocities from CFD with wind tunnel data,  $z=0.7315$  in [62]

The discrepancy between CFD predictions and experimental data was also noted in a study by Rajagopalan et al. [69], where mean velocity predictions from the Rot3DC unsteady, incompressible flow solver were compared to PIV data obtained for a  $1/48^{\text{th}}$  scale model of the LHA ship for  $0^\circ$  and starboard  $15^\circ$  WOD conditions. This study was part of a comprehensive effort by NAVAIR to better understand ship-aircraft dynamic interface after an on-deck V-22 aircraft experienced an uncommanded roll due to the combined upstream airwake from the ship and a CH-46 aircraft. Figure 29 shows streamwise velocity contours across landing spots #2, #4, #7 and #8. The deck-edge vortices are evident at spots #2 and #4. The influence of the superstructure is shown as well, resulting in velocities with magnitudes of up to 20% of freestream near spot #7. Comparisons of predicted and measured mean vertical velocities at the four landing spots at  $z = 5.206$  in (equivalent to  $z = 20.83$  ft. at full-scale) are shown in Figure 30. Note that this height corresponds to the height of V-22 rotors when the wheels are on the deck. Figure 30 shows that CFD fails to capture the peaks at  $y = \pm 0.6$  near the edges of the ship (spot #4), similar to Ref. [68], but this discrepancy does not seem to have been discussed in literature.

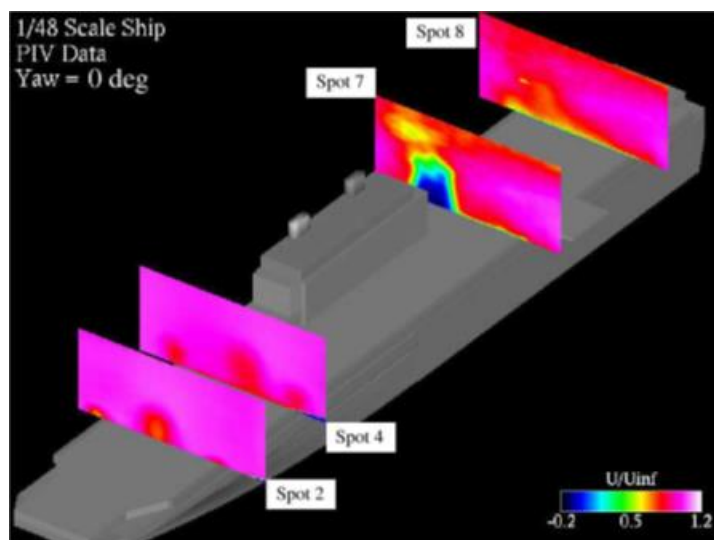


Figure 29: Streamwise velocity contour plots as determined from PIV data for headwind condition [69]

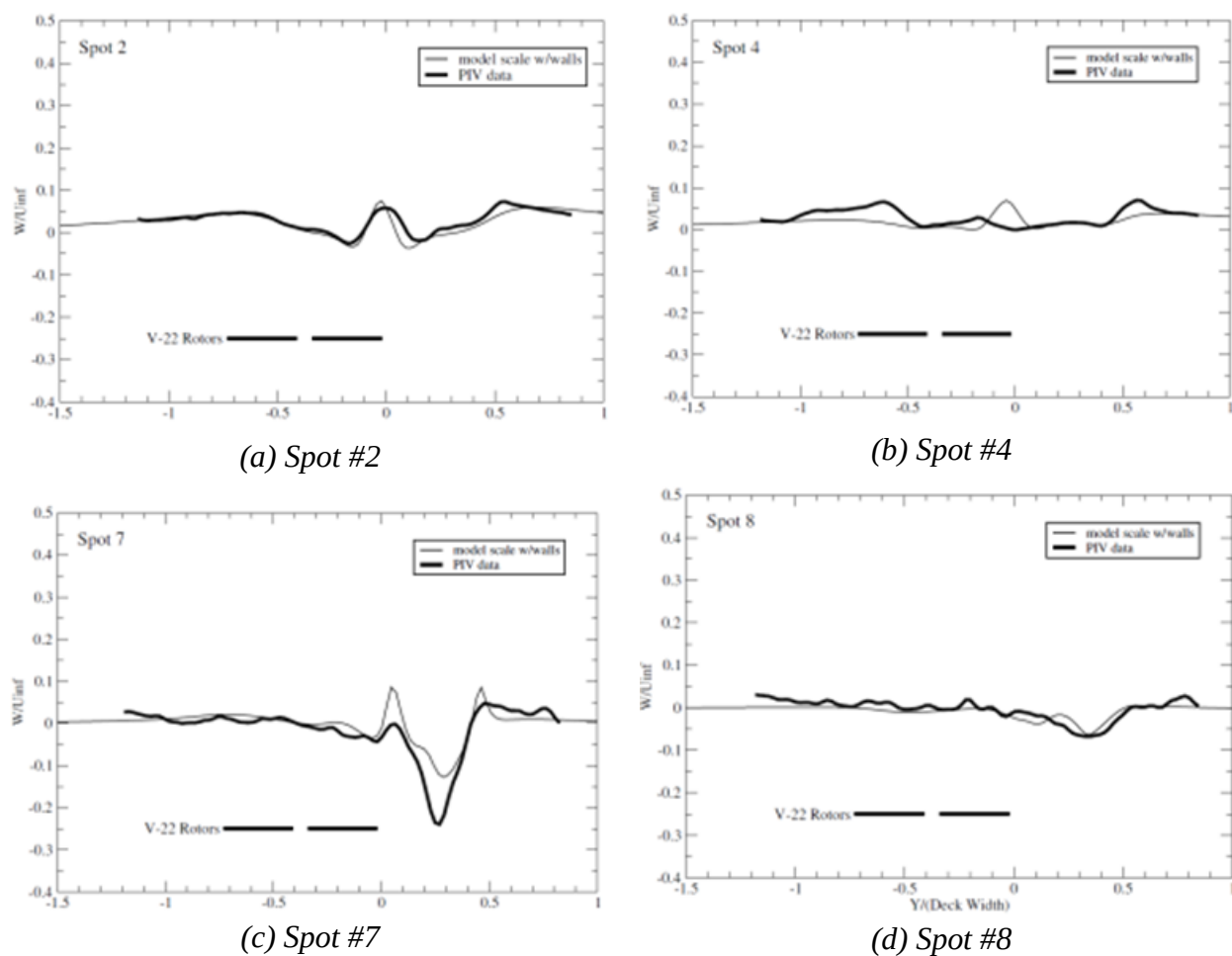


Figure 30: Mean vertical velocities at transverse planes across four landing spots for  $z=5.206$  in [69]

In recent years, the need for a more systematic approach to studying airwake characteristics for flat-topped ships has been highlighted [70], and more recent studies have attempted to provide quantification metrics such as turbulence intensities [71-73] to enable direct comparison between tests and simulations.

An extensive CFD investigation of flow over a LHA ship was performed using OpenFOAM in Ref. [71], and qualitative agreement with results from [64] was noted. Mean velocity magnitudes across the ship beam at spots #2, #4, #7 and #8 are shown in Figure 31 for  $z = 3$  m (9.8 ft.) and  $z = 6$  m (19.7 ft.). The large drop in velocities at spots #2 and #4 are due to bow separation while those at spots #7 and #8 are due to the shedding downstream of the superstructure. It is interesting to note that the influence of deck-edge vortices is more pronounced closer to the deck at  $z = 3$  m at spot #4. Corresponding turbulence intensity plots are shown in Figure 32, where bow separation produces the greatest turbulence at spots #2 and #4. Unfortunately, the definition of “turbulence intensity” is not specified by the author, and thus these figures should be viewed from a qualitative perspective. Streamwise turbulence intensity plots across the ship centerline and a line through the portside landing spots are shown in Figure 33 for two mesh resolutions. At the centerline, a peak in turbulence is noted due to bow separation and predictions from the two mesh resolutions show good agreement. Agreement in predictions deteriorates slightly on the portside line with differences of up to  $\sim 7\%$  evident at  $z = 3$  m.

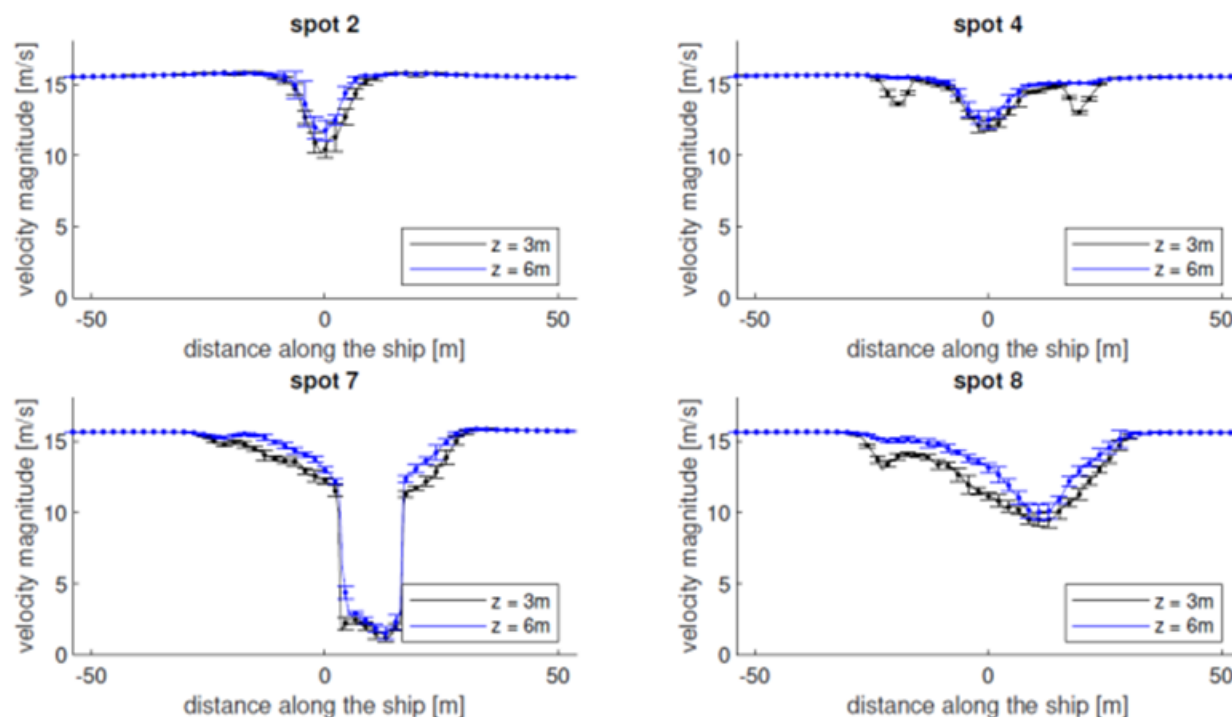


Figure 31: Mean velocity magnitude across landing spots of the LHA ship for a headwind condition [71]

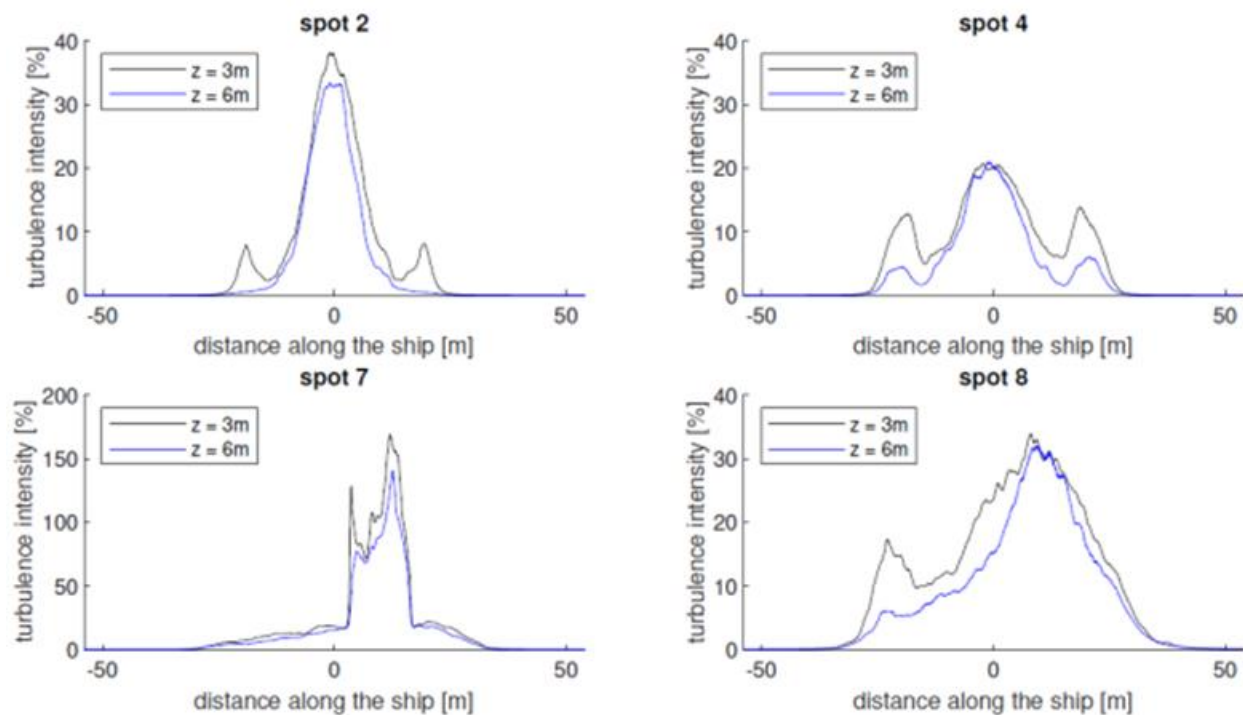


Figure 32: Turbulence intensities across landing spots of the LHA ship for a headwind condition [71]

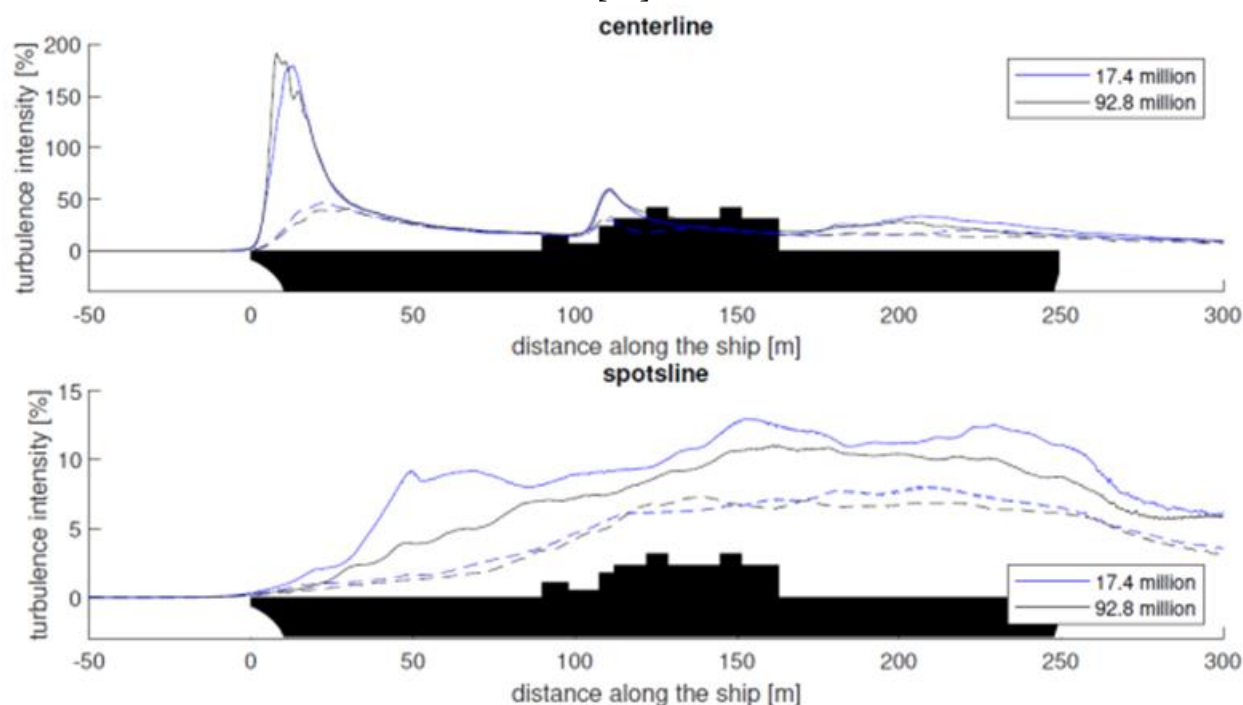
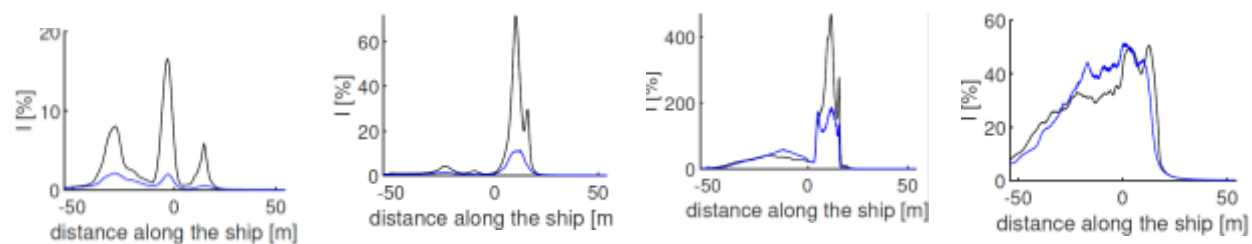


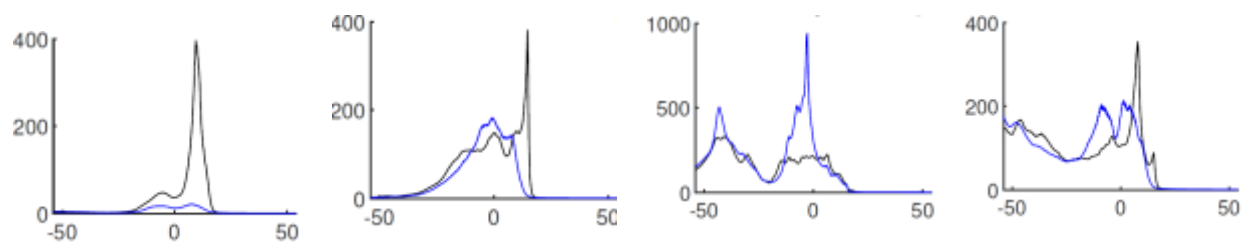
Figure 33: Turbulence intensities across LHA ship length for headwind condition; full line corresponds to  $z = 3\text{ m}$  and dashed line to  $z = 6\text{ m}$  [71]

The influence of WOD direction was assessed also in Ref. [71]. Figure 34 shows turbulence intensity plots for 6 WOD conditions over spots #2, #4, #7 and #8, where “green” winds denote winds from the starboard of the ship and “red” winds from the portside. As the wind conditions

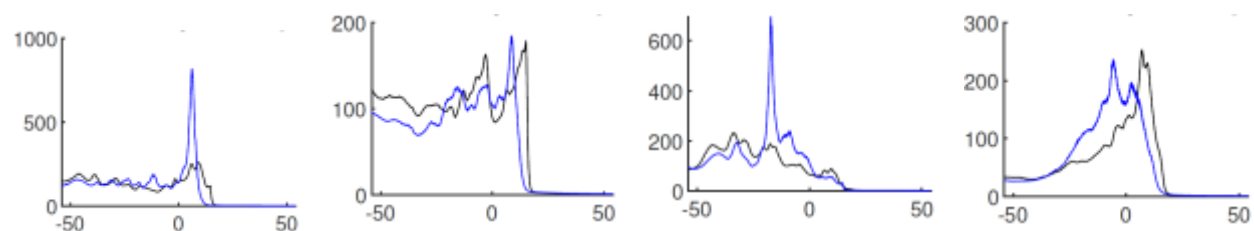
become more oblique, turbulence intensity over the landing spots increases significantly, with beam winds resulting in peak turbulence intensity values of  $\sim 1000\%$ . For turbulence intensities across the ship length shown in Figure 35, the magnitudes of the turbulence intensities for  $60^\circ$  and  $90^\circ$  winds are similar, with the latter condition producing greater turbulence over the entire flight deck including the fore section. Note that no validation against experimental or full-scale test data was performed in this study [71].



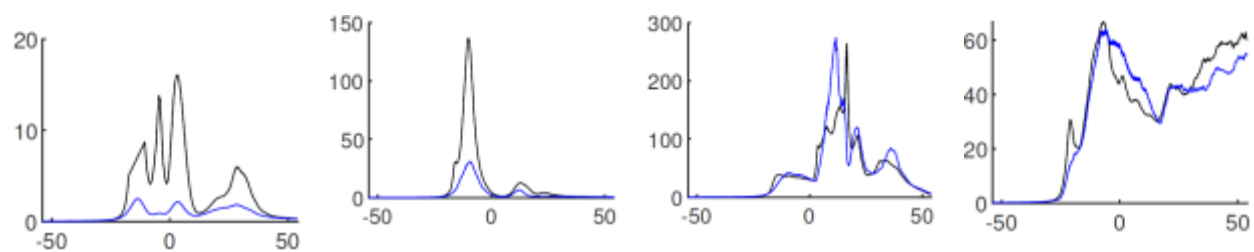
a) Green 30°



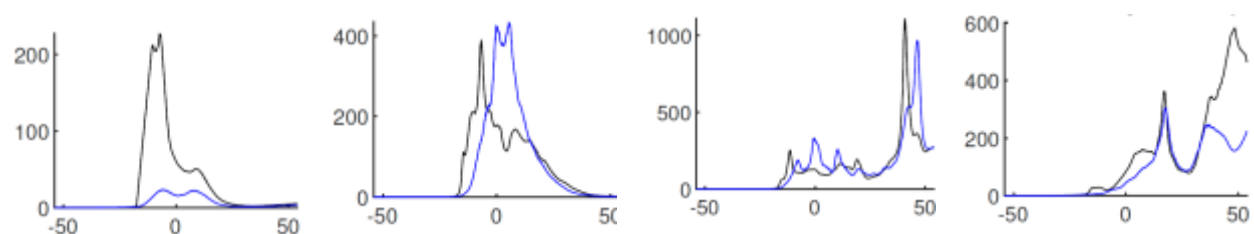
b) Green 60°



c) Green 90°



d) Red 30°



e) Red 60°

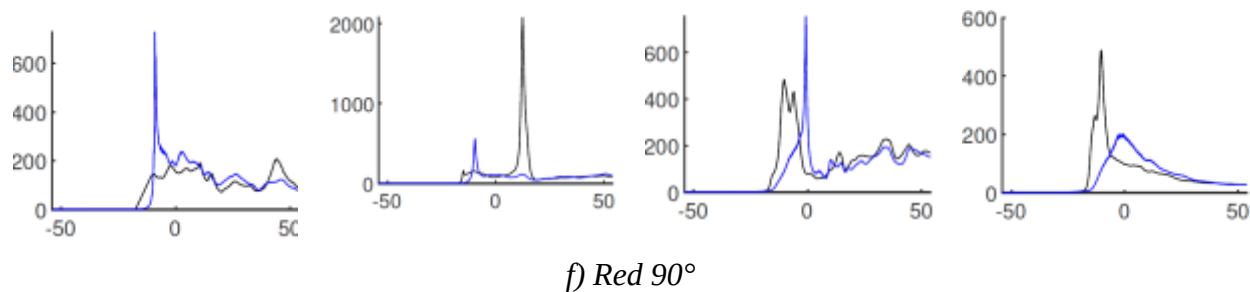
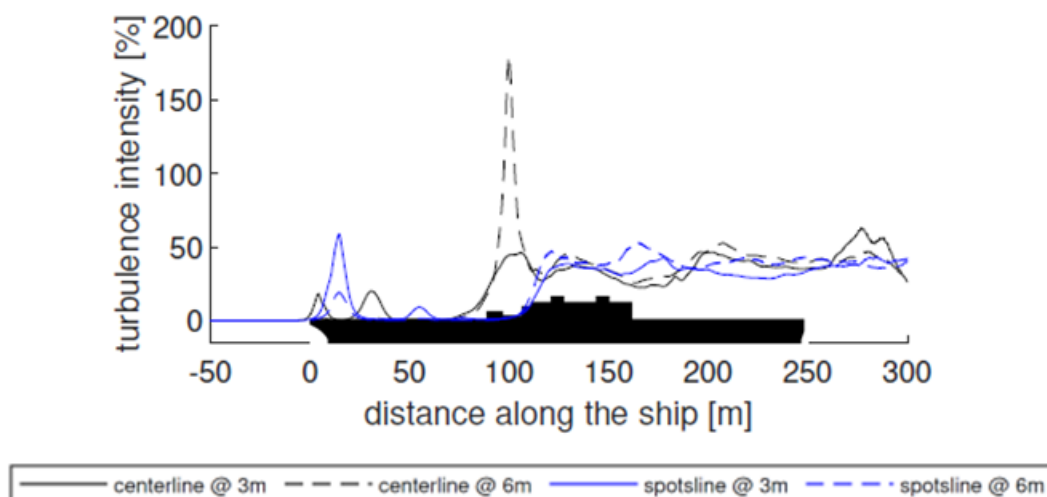
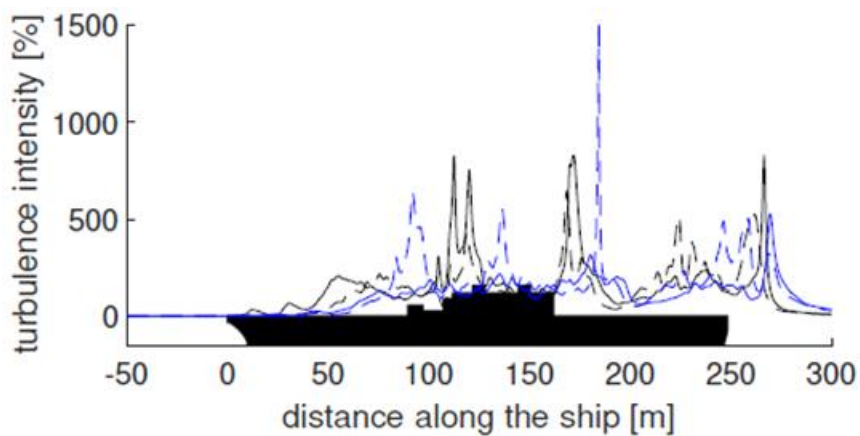


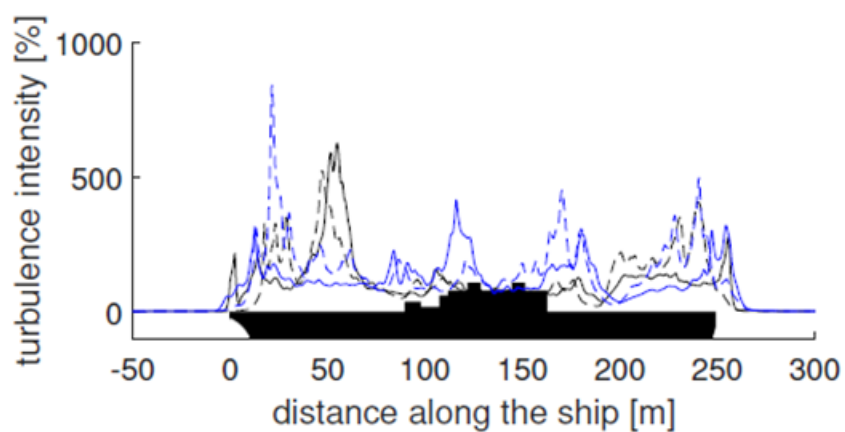
Figure 34: Turbulence intensities across landing spots #2, #4, #7 and #8 (in that order from the left) for various WOD directions; black line corresponds to  $z = 3$  m while blue line corresponds to  $z = 6$  m; “green” winds are from the starboard side of the ship and “red” winds from the portside [71]



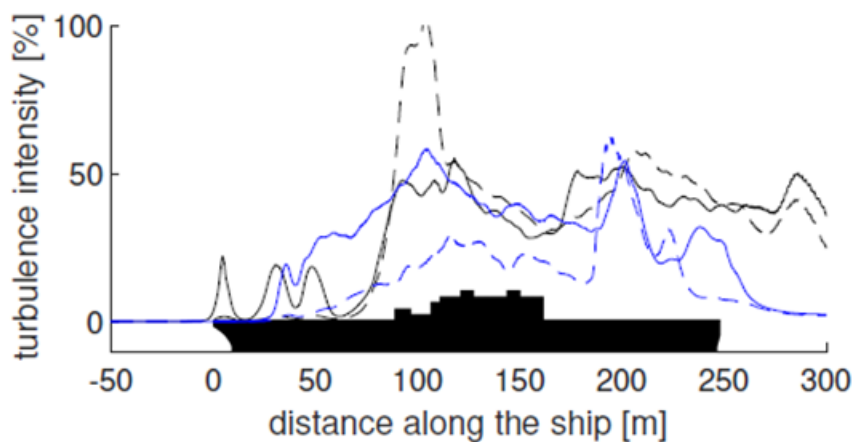
*a) Green 30°*



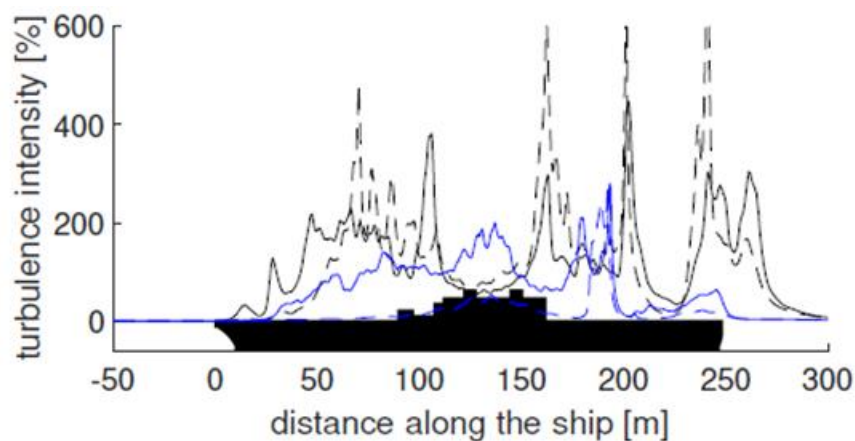
*b) Green 60°*



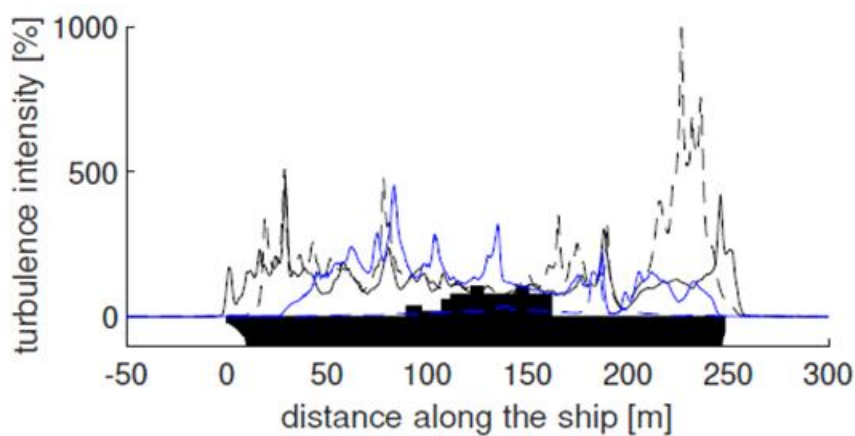
*c) Green 90°*



d) Red 30°



e) Red 60°



f) Red 90°

Figure 35: Turbulence intensities across LHA ship length for various WOD conditions; “green” winds are from the starboard side of the ship and “red” winds from the portside [71]

CFD and experimental investigations of flow over Her Majesty’s Ship (HMS) Queen Elizabeth Class (QEC) aircraft carrier shown in Figure 36 were performed in Refs. [72, 73].

Distribution Statement A

Approved for public release: distribution unlimited

HMS QEC provides a 4-acre flight deck to support fixed- and rotary-wing aircraft, and includes a take-off ramp or “ski jump” to support short take-off, specifically for Lockheed Martin F-35B Advanced Short Take-Off and Vertical Landing (ASTOVL). A notable feature of this carrier is the twin island configuration with the front island serving as base for ship control and navigation and the rear one as base for flight operations. This twin island configuration results in greater turbulence over the deck span when compared to LHA class ships which have one island [72]. Watson et al. validated Delayed Detached Eddy Simulation (DDES) of flow over HMS QEC with measurements obtained from water-tunnel experiments of a 1/200<sup>th</sup> scale model [73]. Two Acoustic Doppler Velocimetry (ADV) probes were used to measure mean velocity components and lateral and vertical velocity fluctuations. Water-tunnel experiments were performed to match Reynolds number to full-scale tests. CFD predictions of turbulence intensity, defined as

$$I = \frac{\left[ \frac{1}{3} (\overline{u'^2} + \overline{v'^2} + \overline{w'^2}) \right]^{\frac{1}{2}}}{U_{\infty}}, \quad (6)$$

for the 1/200<sup>th</sup> scale model showed good agreement with experiments, as evident in Figure 37 [73].



Figure 36: HMS Queen Elizabeth aircraft carrier [73]

Subsequently, full-scale CFD simulations were performed with the atmospheric boundary layer modeled using surface roughness properties typical of oceanic surfaces [73]. Streamlines for headwind condition with turbulence intensity (see Eq. (6)) across a horizontal plane at  $z = 10$  m are shown in Figure 38. Turbulence intensities of ~12% are noted due to the portside deck-edge vortex. Not surprisingly, peak turbulence occurs downstream to the superstructures, and similar plots for oblique WOD conditions are shown in Figure 39. For the green and red 25° and 45° WOD conditions, deck-edge vortices have turbulence intensities of ~20% across the deck. The greatest turbulence across the entire flight deck is evident in the green 90° condition.

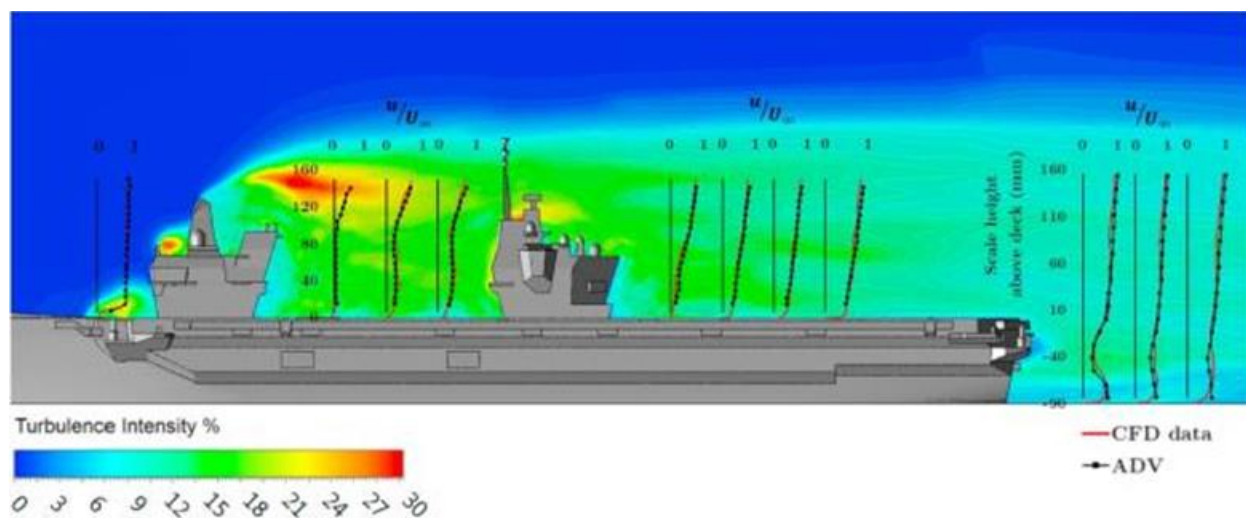


Figure 37: Comparison of experimental and CFD u-velocity components in a plane through center of islands [73]

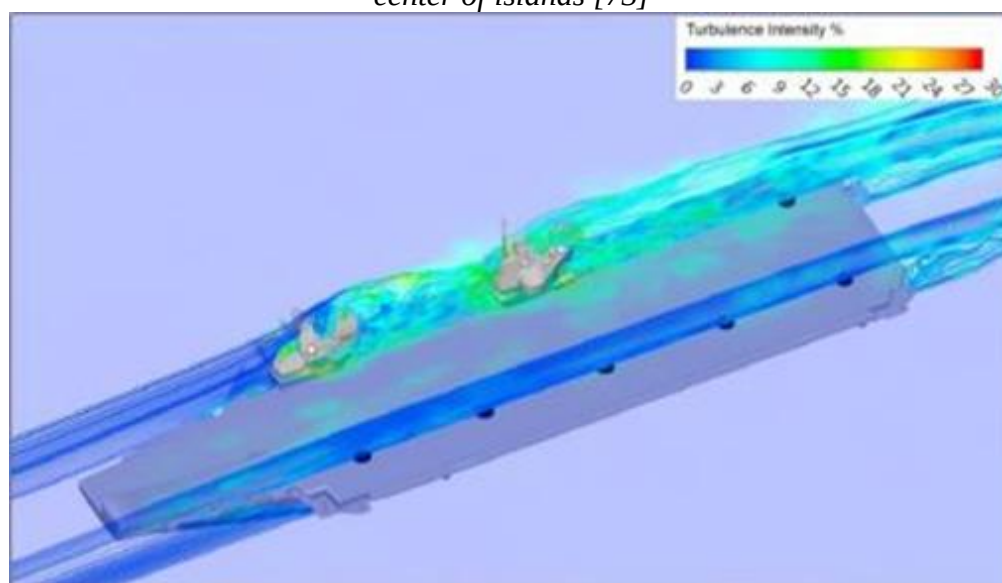


Figure 38: Streamlines for headwind condition in vertical planes through landing spots (black dots) and turbulence intensity contours in a horizontal plane 10 m above deck [73]

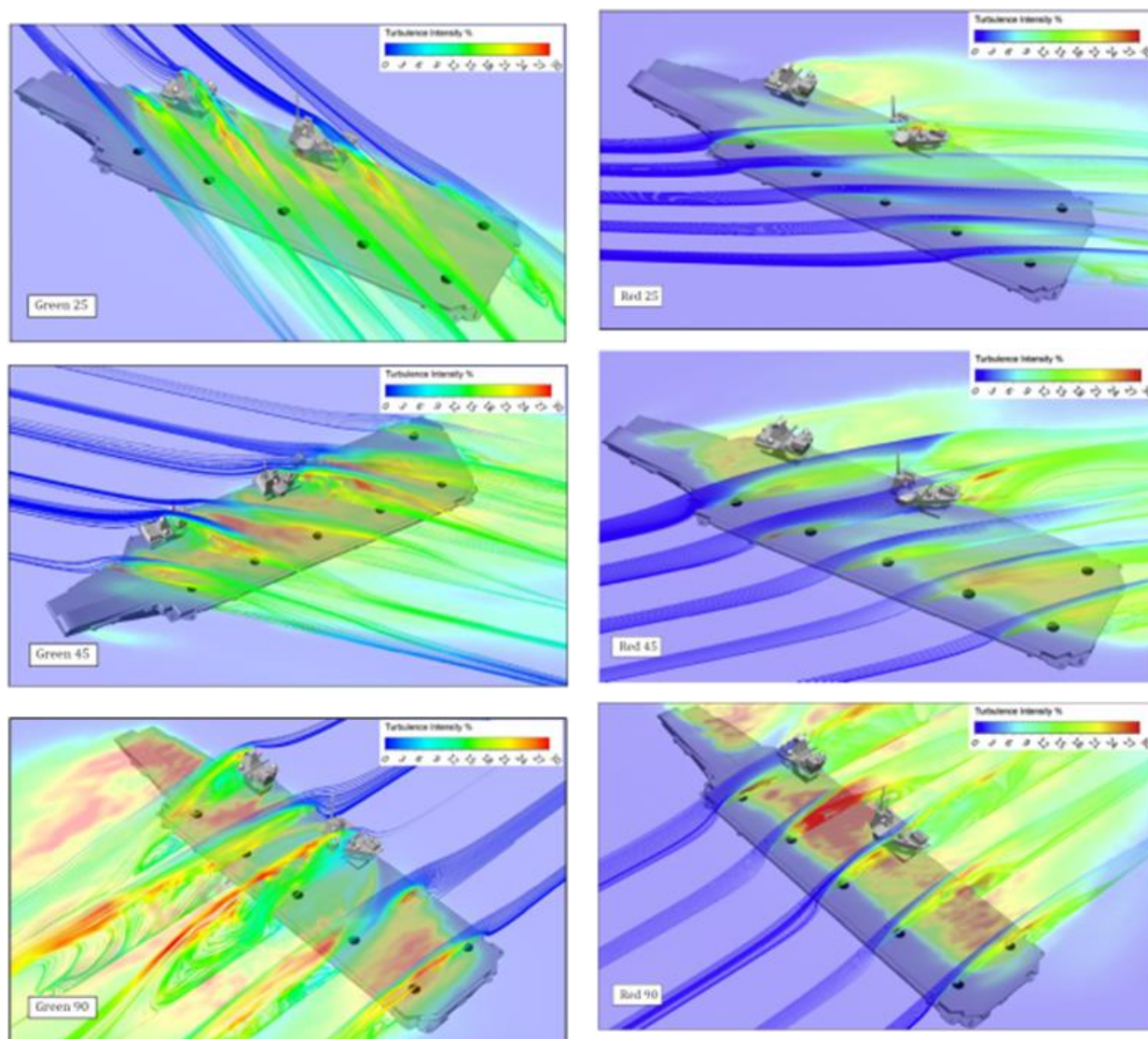


*Figure 39: Streamlines in vertical planes for various WOD directions and turbulence intensity contours in a horizontal plane 10 m above deck [73]*

From the preceding discussions, it is evident that the geometry of a ship superstructure greatly affects the flow patterns observed in the aft of ship. A comparison of the mean flow characteristics around the windward vertical edge of the hangar for an oblique WOD condition is shown in Figure 40 for the Simple Frigate Shape 2 (SFS2) and Type 23 (T-23) Frigate ship geometries [65]. The ship geometries are shown in Figure 41 for reference. The superstructure of the T-23 Frigate does not cover the entire width of the deck as that of the SFS2 and includes a “notch”. Geometrical differences such as these greatly influence the resulting flow characteristics; the interaction of wind, the deck-edge vortex, and the superstructure results in a high vorticity region around the notch, which is not noted with the SFS2 in Figure 40.

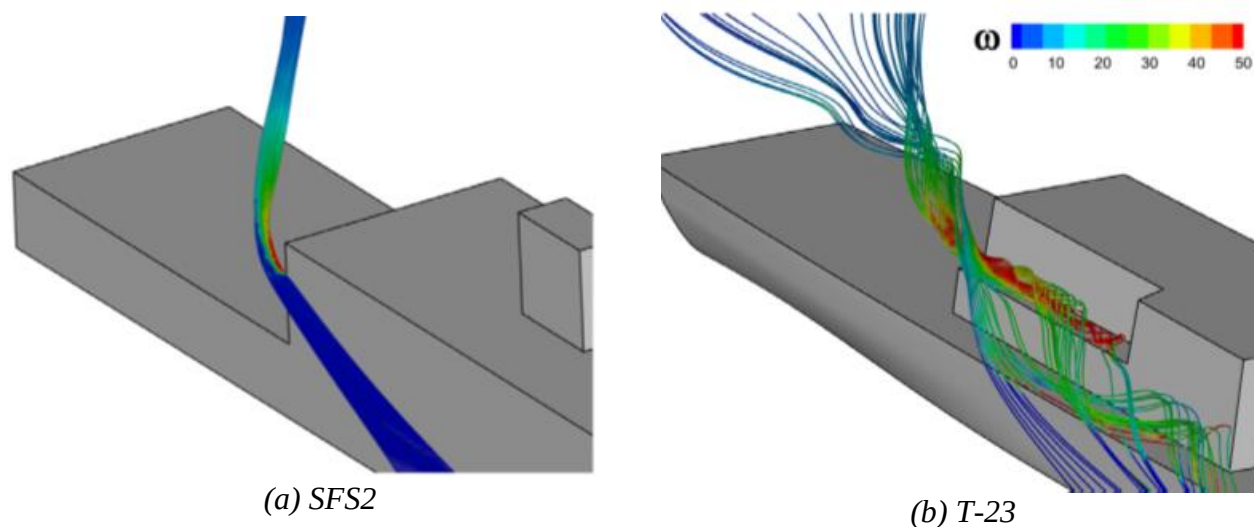


Figure 40: Mean path lines colored by vorticity magnitude for starboard 45° WOD condition highlighting separation characteristics from windward hangar edge [65]

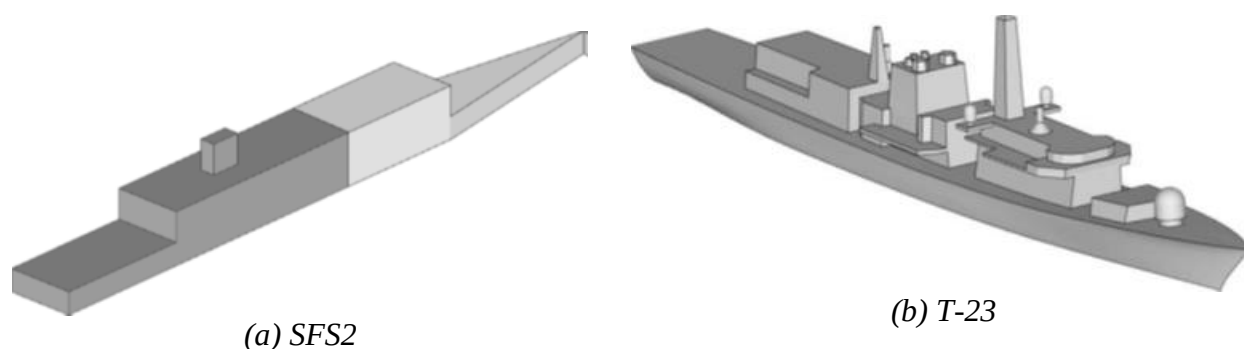


Figure 41: SFS2 and T-23 ship geometries [65]

#### Characterization of Corner Vortex Flow

Deck-edge vortices on a ship originate from corners and propagate along edges of the deck and superstructure. In oblique WOD conditions, relatively strong coherent vortical structures that can influence aircraft dynamics and handling qualities are shed from the corners. Characterization of flow features resulting from corner winds is lacking in ship airwake literature, but such phenomena have been looked at extensively by the wind engineering community given that vortices generated by corner winds can generate high suction regions on roofs of buildings, which, in turn, can lead to severe structural damage [74-77]. The large suction pressure regions are associated with two counter-rotating deck edge vortices that originate from the corner, as illustrated in Figure 42. In a similar manner to shear layers on backward facing steps, these vortices generate fluctuating pressure gradients by flapping over the top surface [74]. The size of each vortex depends on building dimensions, wind speed, wind incidence angle with the corner, and upstream (atmospheric) turbulence levels. Each vortex grows and decays alternatively (driven by the asymmetric shedding from the body), producing asymmetric instantaneous pressure distributions over the roof [78, 79]. Flow perpendicular to an edge produces separation bubbles over the edge and flow at an incidence angle of 45° results in

symmetric vortices along the edges. The magnitude of the peak suction pressure over the top surface tends to be greater for flow with low turbulence [74].

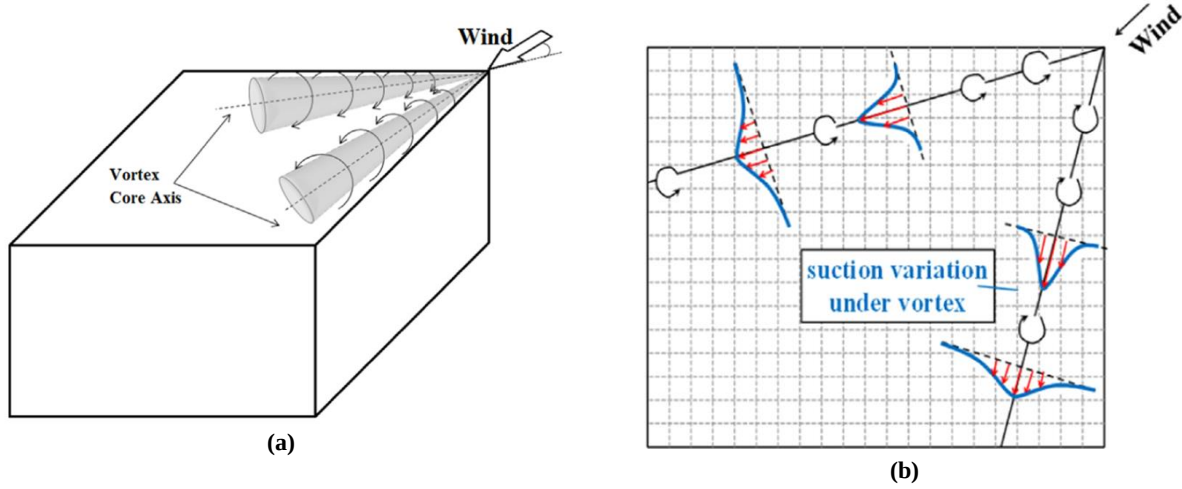


Figure 42: (a) Illustration of counter-rotating vortices generated, and (b) variation of suction pressure due to counter-rotating vortices over a flat roof from a corner wind [76].

Liu et al. in Ref. [75] performed wind tunnel experiments to assess the efficiency of passive vortex generators (PVGs) in mitigating pressure suction effects of the counter-rotating vortices on the top surface of a 1:244 scale model of a building that is 110 m long, 65.8 m wide, 22 m high and has a 5% roof slope—the centerline along the length of the roof is 1.64 m higher than the edges (5% of half of 65.8 m). An atmospheric turbulent boundary layer was simulated in the tunnel using the power law wind profile

$$U(z) = U_{ref} \left( \frac{z}{z_{ref}} \right)^{\alpha}, \quad (7)$$

where the power index  $\alpha$  is set equal to 0.16, the roof height is taken as reference height  $z_{ref}$  with a corresponding reference speed  $U_{ref}$  of 8.6 m/s. Pressure measurements were undertaken following the pressure tape layout in Figure 43. Given the interest in flow phenomenology, only the baseline results without PVGs are of interest here.

Peak pressure coefficient  $C_p$  for various incidence angles are shown in Figure 44, indicating that the highest suction occurs at  $\beta = 40^\circ$  and tap #21, close to the left edge in the blue corner zone in Figure 43. In the context of shipboard landing, the vortex along this edge is similar to a deck-edge vortex on an aircraft carrier that can influence aircraft that are on or near landing spots close to the edge of the ship.

Liu et al. [75] also quantify the size of the vortices for the  $\beta = 40^\circ$  incidence condition, as shown in Figure 45. The vortex center axis (red line on the figure) is determined using the position of maximum mean suction pressure on cross-sections perpendicular to the wind direction. The radius of the vortex (black line on the figure) is estimated using the position of maximum root-mean-square (RMS) of pressure on the cross sections. The vortex along the longer edge of the building continually expands and moves away from the edge, while the vortex along the shorter edge expands slightly and remains closer to the edge. In general, the vortex along the edge that is at a smaller angle relative to the incoming flow is larger in size [74].

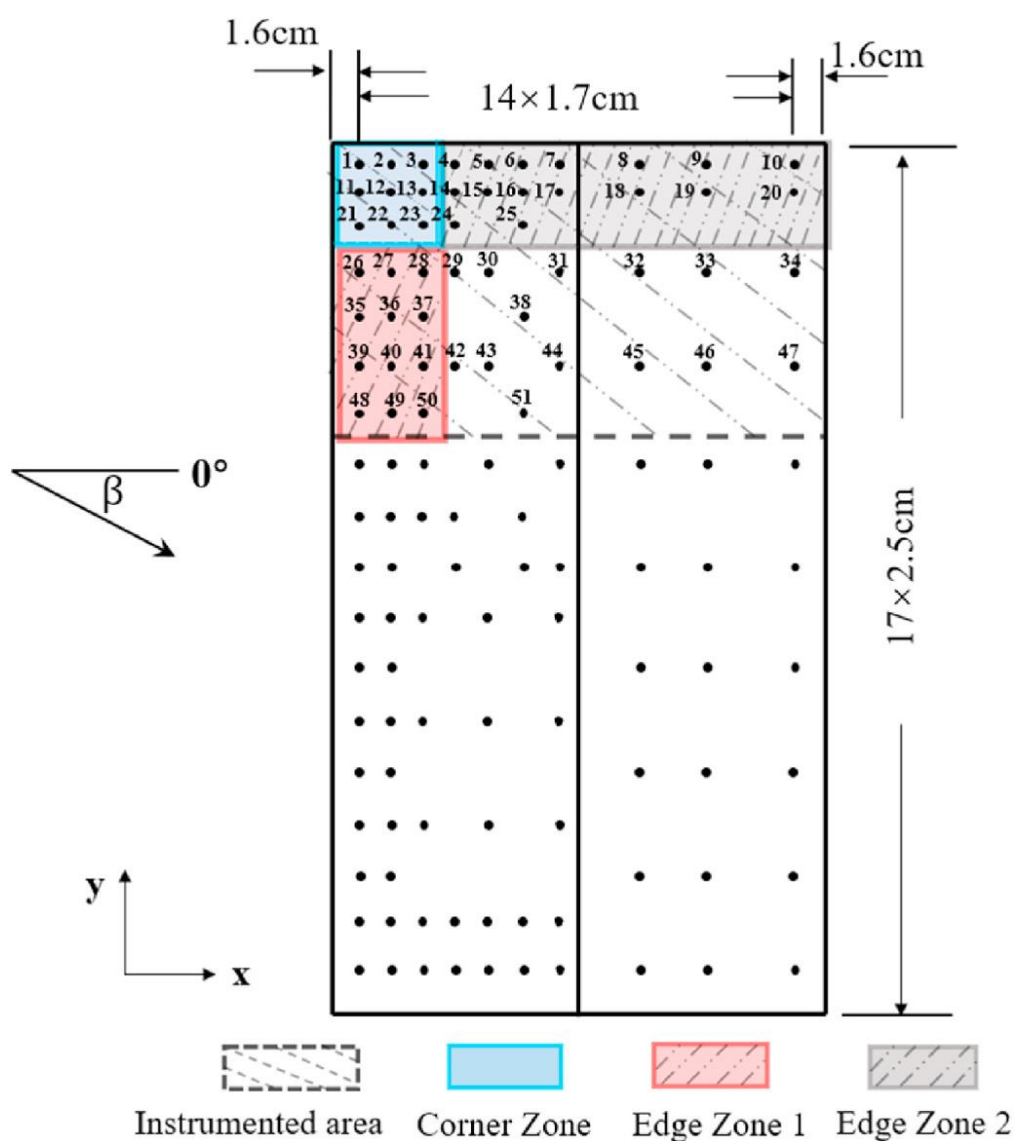


Figure 43: Pressure tap arrangement on top surface of building in Ref.[75].

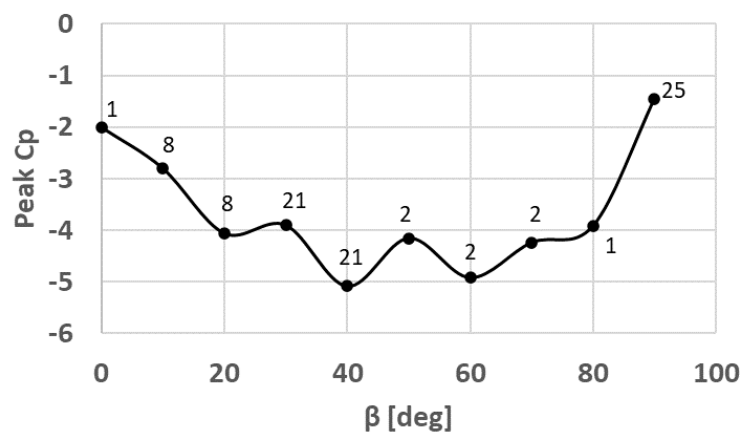


Figure 44: Peak  $C_p$  measured for different incidence angles; tap number indicated [75].

Distribution Statement A

Approved for public release: distribution unlimited

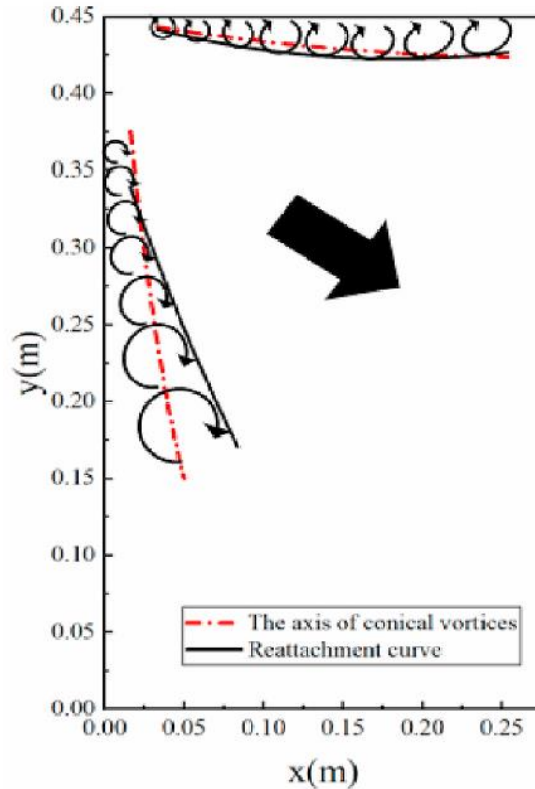


Figure 45: Range of counter-rotating vortices for  $\beta = 40^\circ$  condition. Adapted from [75]. Please note that there is an error in the original paper that shows the sign of the vortex along the shorter edge pointing in the opposite direction.

Power spectral density (PSD) plots of pressure measurements at tap #21 for  $\beta = 40^\circ$  and tap #2 for  $\beta = 70^\circ$  are shown in Figure 46 and Figure 47 respectively, where  $f \cdot H/U$  is the Strouhal characterization frequency with building height  $H$  as the characteristic dimension, and  $f \cdot S(f)/\sigma^2$  is the dimensionless power spectrum with  $S(f)$  being the measured pressure spectrum and  $\sigma^2$  is the variance of the measurements [75]. It should be noted that pre-multiplying the frequency by  $S(f)/\sigma^2$  always results in peaks in the spectrum, even if the pure signal does not contain any dominating periodicity [74]. The “NO PVG” plots in Figure 46 and Figure 47 are of interest here, and show that for  $\beta = 40^\circ$ , the most energetic flow feature occurs at a Strouhal frequency of 0.07 with additional peaks at higher values such as 0.15 and 0.17. For  $\beta = 70^\circ$ , the most energetic flow features occur at Strouhal frequencies of 0.07, 0.13 and 0.19.

Kozmar in Ref. [74] experimentally investigated the pressure distribution and fluctuations over the top surface of a 0.2 m wall-mounted cube due to corner winds for three simulated atmospheric boundary layers corresponding to low- (LT), mid- (MT) and high-turbulence (HT) levels. The power index  $\alpha$  in Eq. (7) was set to 0.17, 0.21 and 0.35, for values of  $U_{ref}$  of 14.97, 13.48 and 10.14 m/s, respectively with flow incidence angle of  $40^\circ$ . Contours of the steady pressure distribution over the cube are shown in Figure 48 showing that the magnitude of the suction peak on the top surface decreases with increasing upstream turbulence level.

The root-mean-square (RMS) of the pressure coefficient ( $C_{p,rms}$ ) along three lines on the top surface of the cube are shown in Figure 49. Figure 49(a) shows  $C_{p,rms}$  underlying the conical vortex along the leading edge. The pressure peaks close to the corner of the cube and decreases

downstream as the vortex tube expands. In Figure 49(b), a decrease in suction pressure is noted closer to the edge, where the boundaries of the two counter-rotating vortices interact. The line considered in Figure 49(c) starts at a region outside the vortex, progresses towards the center, and ends in a region underlying it. Suction pressure underneath the vortex is approximately 3-4 times higher than that outside the vortex, indicating the potential adverse impact these vortices can have on the surface and on aircraft operating within it. For all three profiles, increasing the upstream turbulence level results in greater flow diffusion over the top surface. Greater fluctuations in  $C_{p,rms}$  were observed, but it is not clear whether this is a consequence of unsteadiness in the inflow, or the vortices moving more.

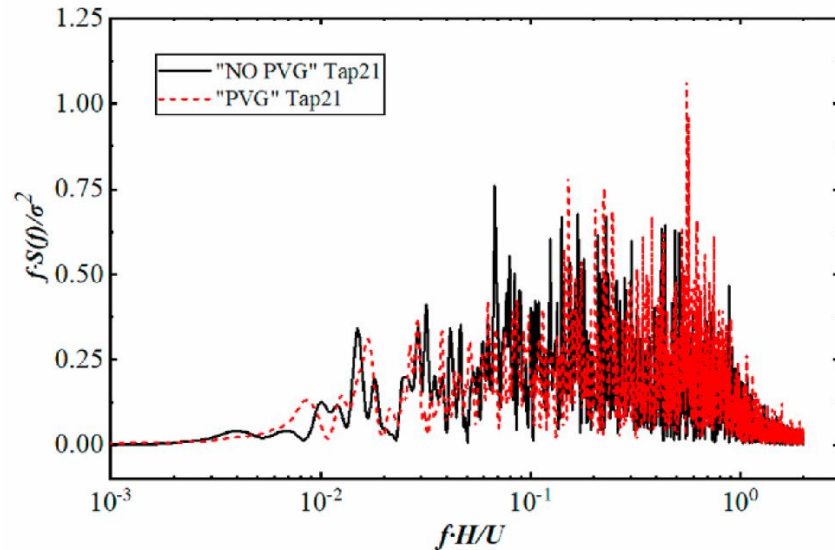


Figure 46: Power spectrum of pressure measurements at tap #21 for  $\beta = 40^\circ$  [75].

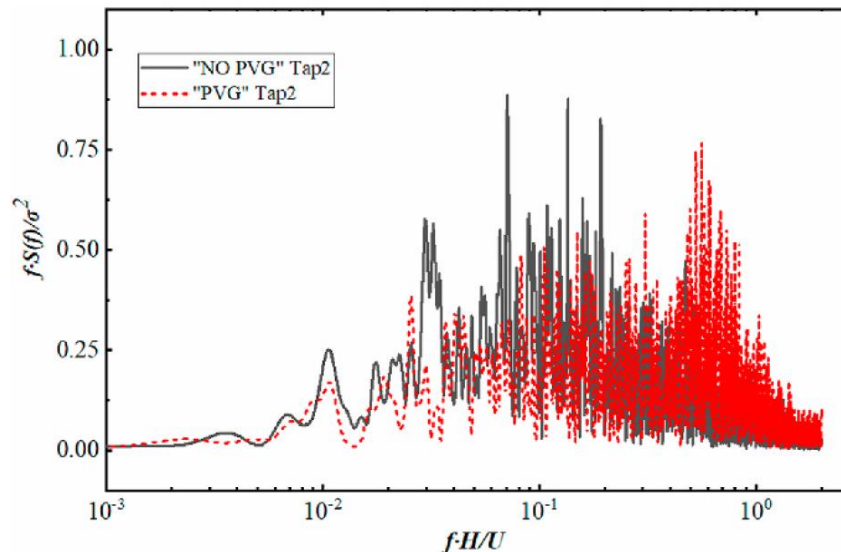


Figure 47: Power spectrum of pressure measurements at tap #2 for  $\beta = 70^\circ$  [75].

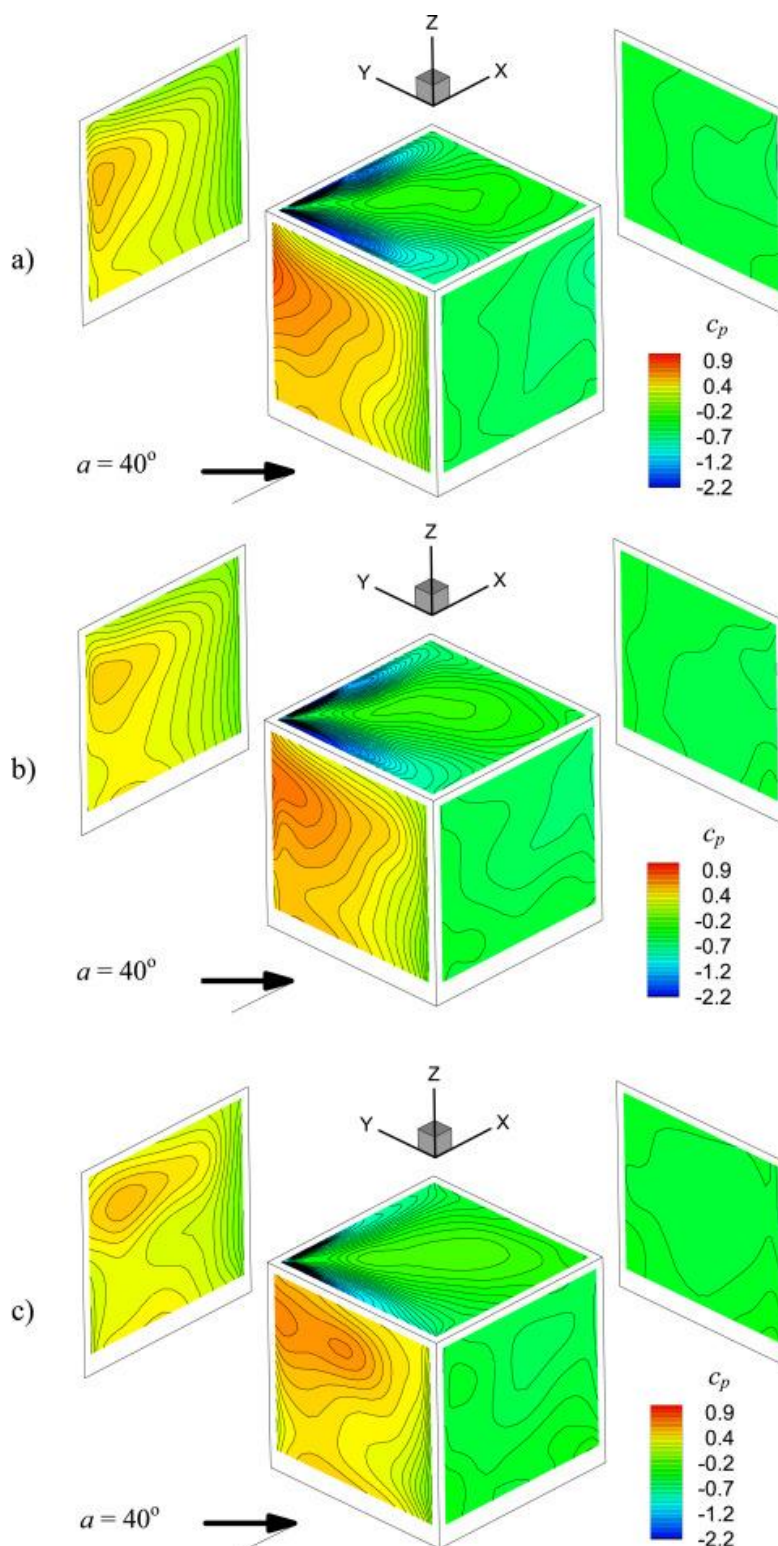


Figure 48: Contours of steady pressure distribution over surfaces of a cube for a corner wind at an incidence angle of  $40^\circ$  [74].

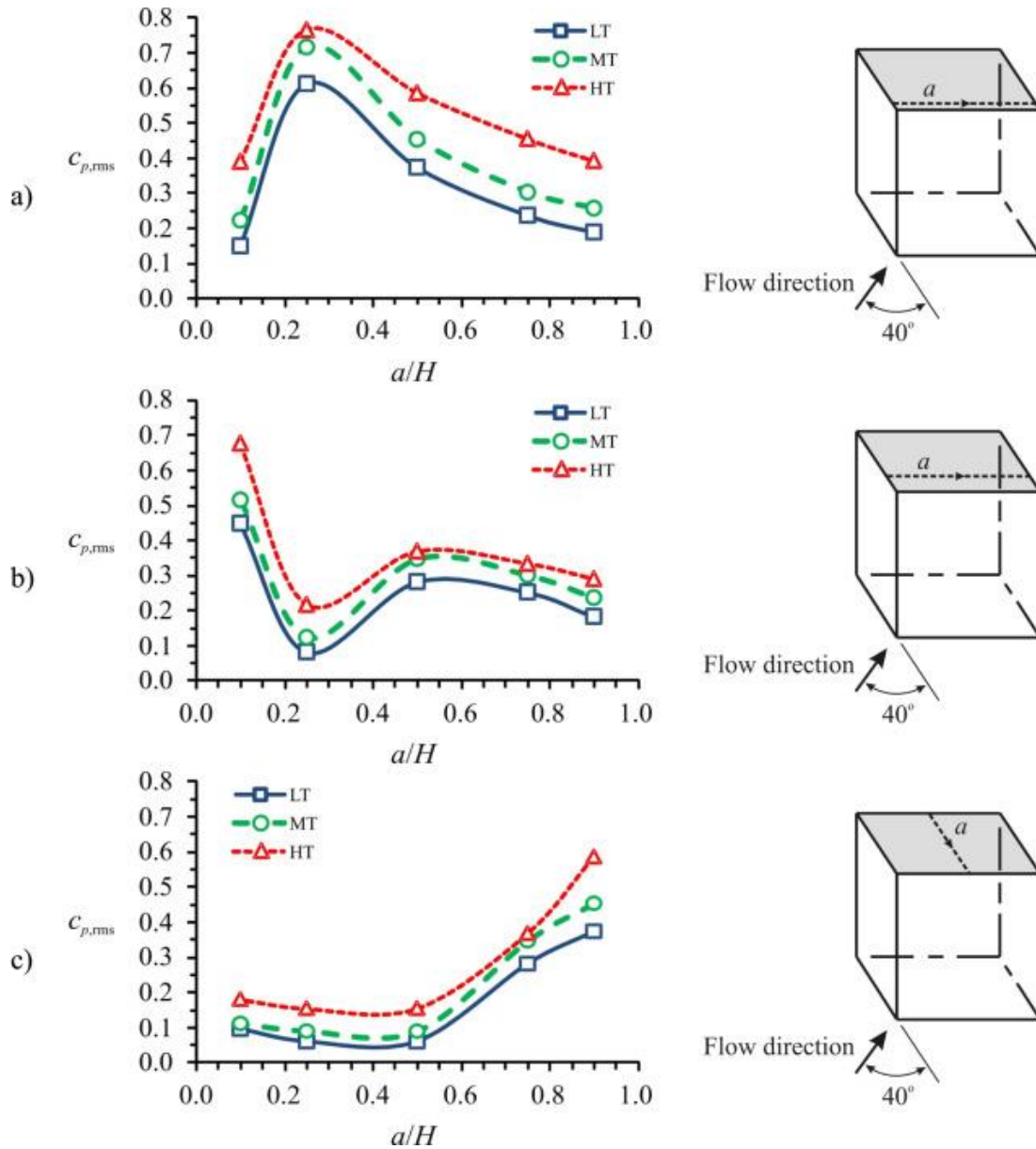


Figure 49: Profiles of  $C_{p,rms}$  coefficients on the top surface of the wall-mounted cube at (a)  $x/H = 0.1$ , (b)  $x/H = 0.25$  and (c)  $y/H = 0.5$  [74].

Power spectral density plots of surface pressure fluctuations at various points along the three lines considered in Ref. [74] (see Figure 50 to Figure 52). Close to the corner at point #1 in Figure 50, vortex shedding due to the body edges occurs at a characteristic (for a cube [57]) Strouhal frequency of 0.1. As the vortex develops from point #1 to #5, the primary frequency progressively shifts to  $\sim 0.03$  and a secondary peak at  $St = 1.0$  emerges and becomes dominant. This change in periodicity has been attributed to reattachment of the flow on the top surface downstream from the corner [74]. The shift in primary periodicity is also noted in PSD plots for the  $x/H = 0.25$  profile in Figure 51. The secondary high-frequency peak is not as pronounced along this profile. The PSD plots in Figure 52 correspond to a profile where points #3 and #8 are outside the conical vortex region and points #11 and #12 are underneath the conical vortex. Outside the conical vortex region, the dominant periodicity occurs at  $St = 0.03$  with a less

pronounced secondary peak at  $St = 2.0$ . As we move along the profile into the region underneath the vortex, the secondary peak at  $St = 2.0$  becomes more dominant.

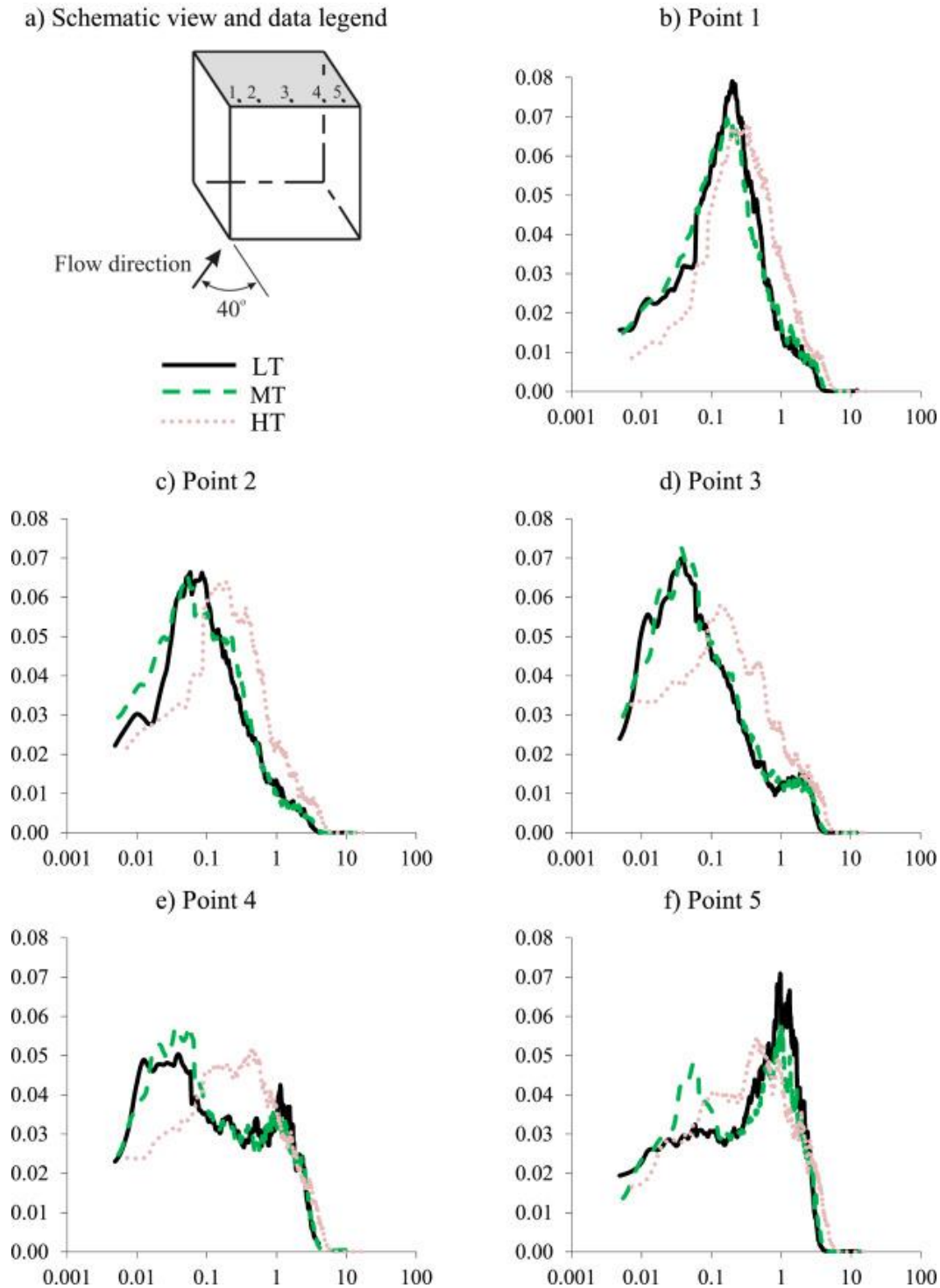


Figure 50: PSD of surface pressure fluctuations on top surface of cube along  $x/H = 0.1$ ;  $x$ -axis is Strouhal frequency  $f \cdot H/U$  and  $y$ -axis is the dimensionless power spectrum [74].

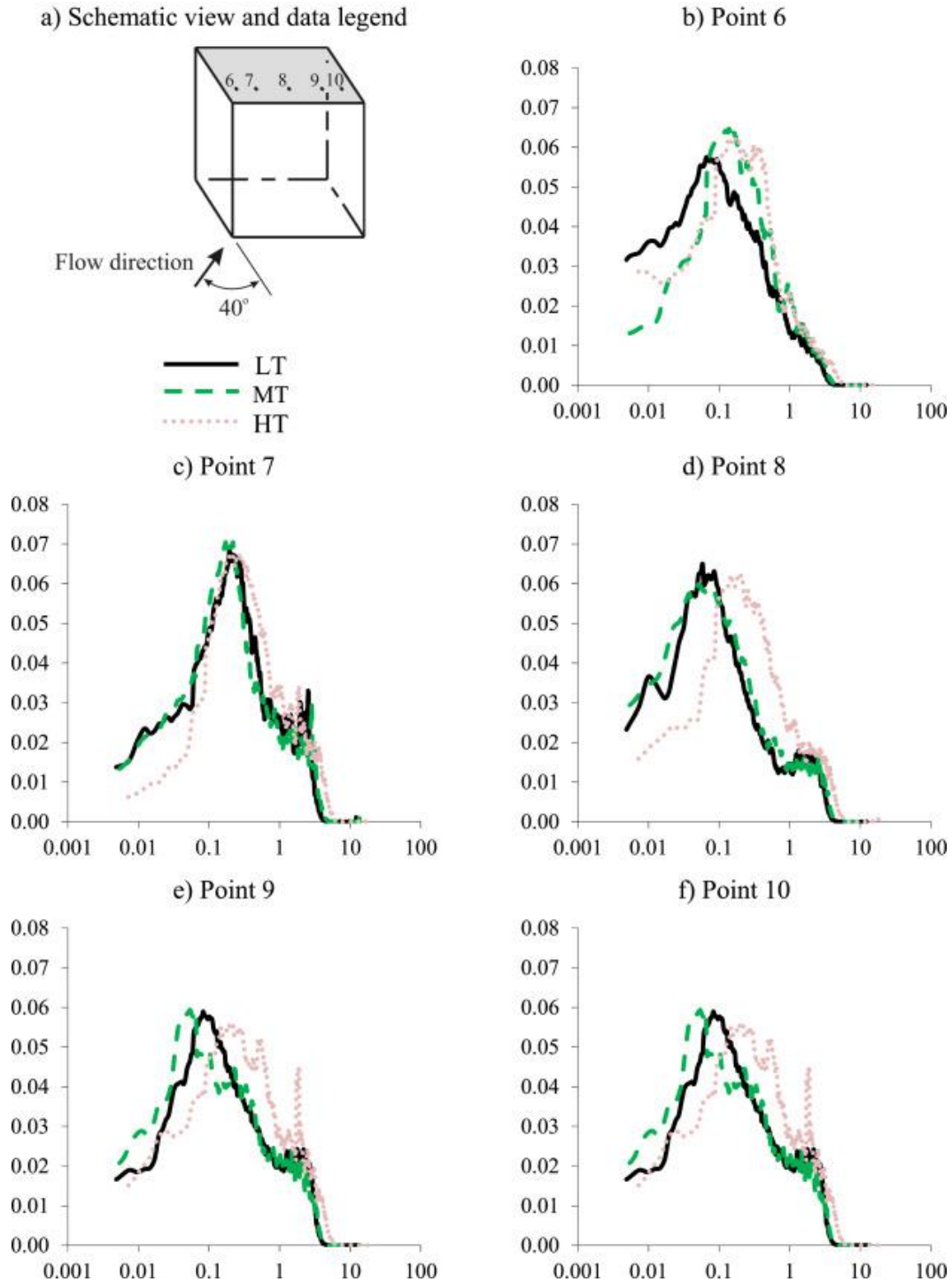


Figure 51: PSD of surface pressure fluctuations on top surface of cube along  $x/H = 0.25$ ;  $x$ -axis is Strouhal frequency  $f \cdot H/U$  and  $y$ -axis is the dimensionless power spectrum [74].

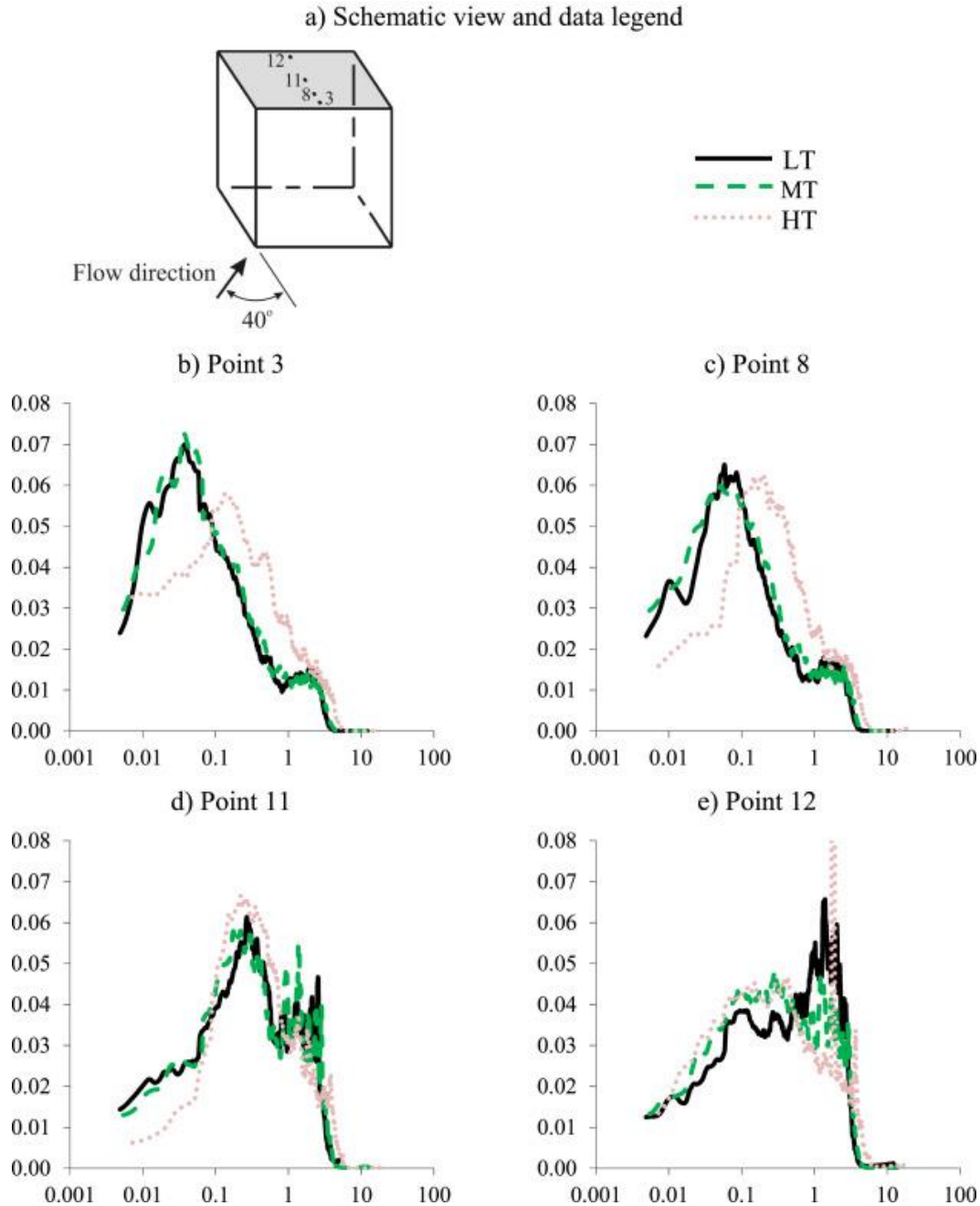


Figure 52: PSD of surface pressure fluctuations on top surface of cube along  $y/H = 0.5$ ;  $x$ -axis is Strouhal frequency  $f \cdot H/U$  and  $y$ -axis is the dimensionless power spectrum [74].

From the preceding discussion, corner flow produces a complex distribution of pressure with varied periodicity over a surface. It is important to note that the wind engineering community tends to focus on surface level measurements. For the ship-helicopter DI scenario, we also need to characterize corner flow properties above and beyond the surface since those are the regions where aircraft operate.

#### Influence of Ship Motion on Ship Airwake

The ship-helicopter DI is further complicated by wave induced motion of the ship [80], which can alter the vortex shedding strength and frequency of the airwake, and also induce a

phenomena known as vortex “lock-in” [81], where the natural airwake shedding frequencies are reinforced by oscillatory motion of the ship. These effects can in turn, influence pilot workload during shipboard operations. Most investigations in literature tend to focus on airwake over static ship configurations, with ship motion receiving attention only recently [81-84].

The influence of static and dynamic pitch attitude of a 1/100 scale model of the SFS2 ship geometry (see Figure 41) on ship airwake was investigated by Sydney et al. in Ref. [81] using PIV measurements of longitudinal and vertical velocities along the symmetry plane of the ship model. Time-averaged velocity magnitude contours for the baseline condition with the ship at an even keel in a headwind are shown in Figure 53 with relevant flow features highlighted.

When the ship is statically pitched, the wake boundary – the layer between the ship airwake and freestream - translates higher, as evident in Figure 54 [81]. Note that the pitch angle is applied relative to waterline point of the bow. Wake boundary translation is due to airwake from the bow and front step of the superstructure, both of which are located at greater heights when the ship is pitched compared to the baseline case. Corresponding TKE contours are shown in Figure 55. As pitch attitude increases, the point at which turbulent airwake contributions from the front step, funnel or stack, and rear step interact moves closer to the hangar, and the recirculation region immediately aft of the hangar shrinks. The magnitude of the turbulence above the ship also increases relative to the baseline case.

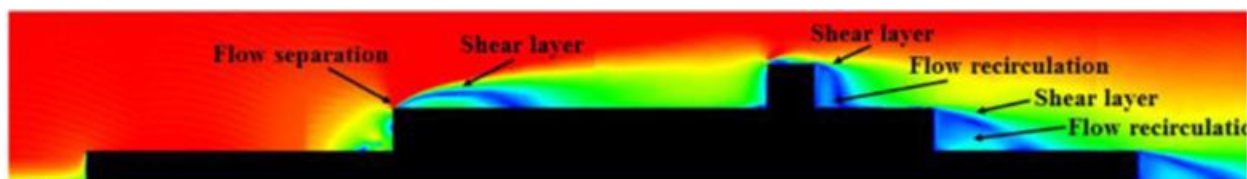


Figure 53: Time-averaged velocity magnitude over SFS2 model ship at even keel in a headwind; red indicates high velocity and blue indicates low velocity [81]

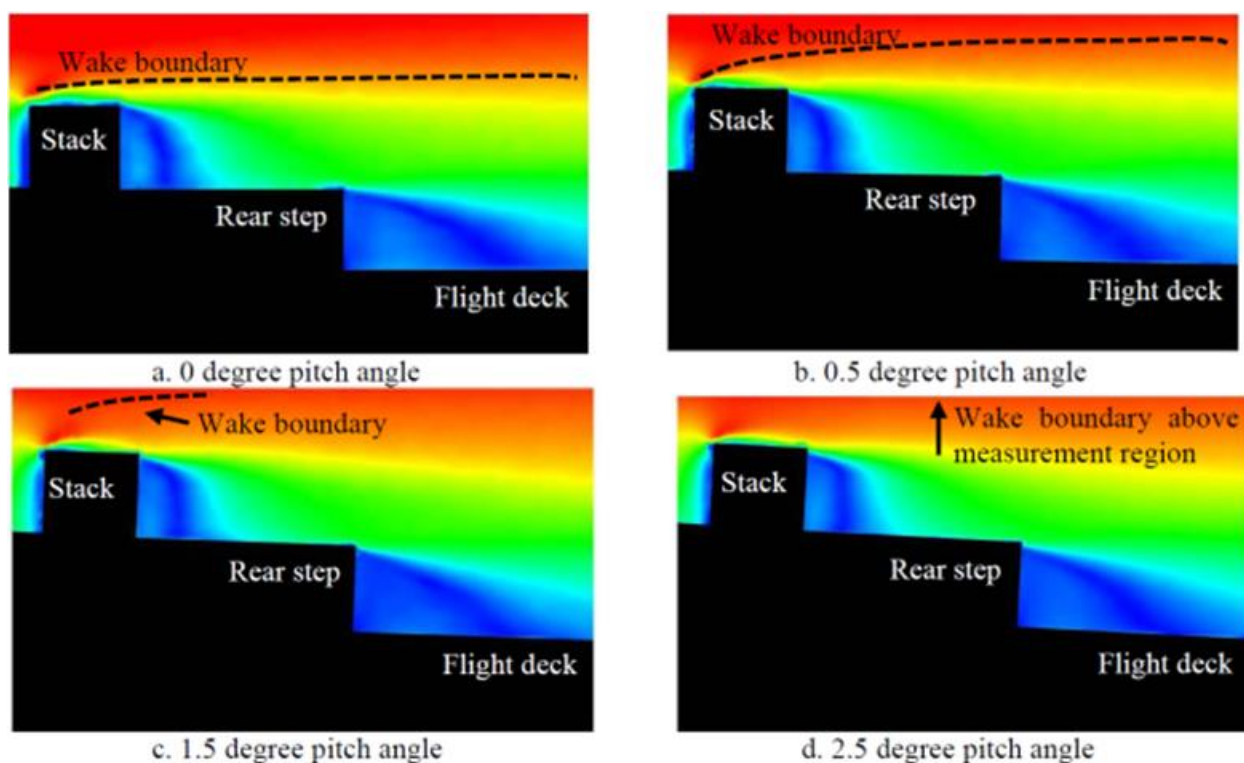


Figure 54: Effect of static pitch angle on mean velocity magnitude; red indicates high velocity and blue indicates low velocity [81]

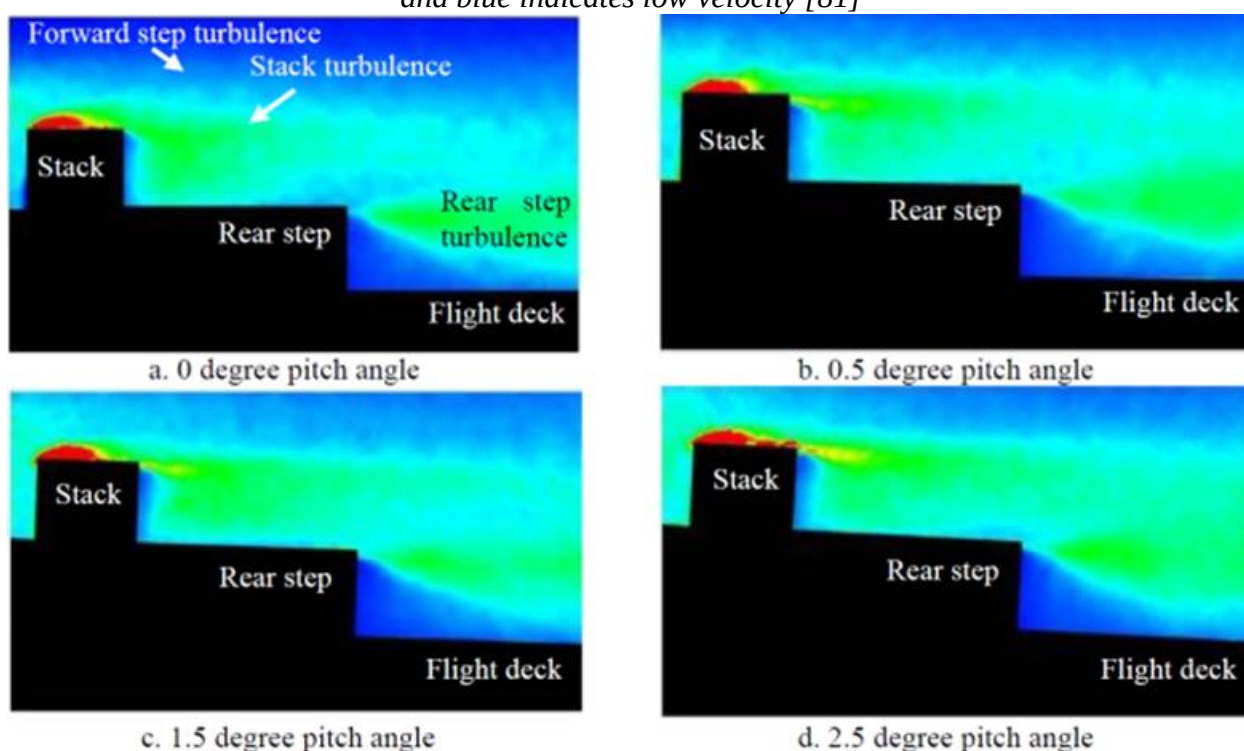


Figure 55: Effect of static pitch angle on TKE; red indicates high TKE and blue indicates low TKE [81]

Dynamic pitching of the ship at 1 and 2 Hz with an amplitude of  $2.5^\circ$  was also considered [81], and instantaneous velocity magnitude contours over one cycle are shown in Figure 56. The influence of deck motion is apparent when cases (b) and (d), corresponding to pitch attitudes of  $0^\circ$  with the bow moving up and down respectively, are examined. In case (d), flow perturbations are observed in the region where freestream was dominant in case (b), as indicated on the figure. These perturbations correspond to wake from the bow and front step that have convected downstream from the instance when these components were pitched upwards. Therefore, there is a lag between ship motion and airwake response over the deck. The frequency spectrum of velocity magnitude at a point located horizontally at the hangar and vertically at 1.4 hangar heights is shown in Figure 57, and not surprisingly indicates that there is an additional primary frequency in the wake associated with the ship motion frequency [81].

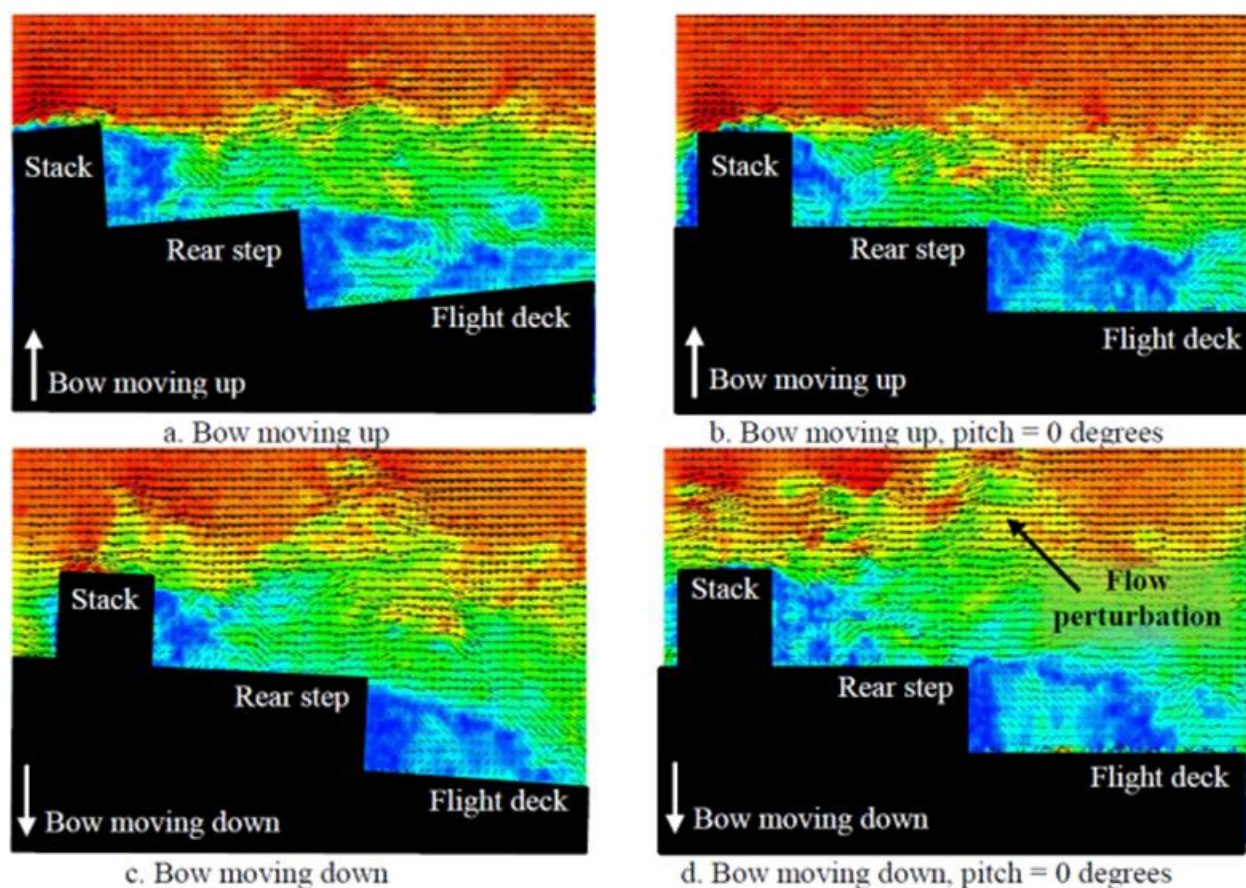


Figure 56: Instantaneous contours of velocity magnitude over one period of pitch motion [81]

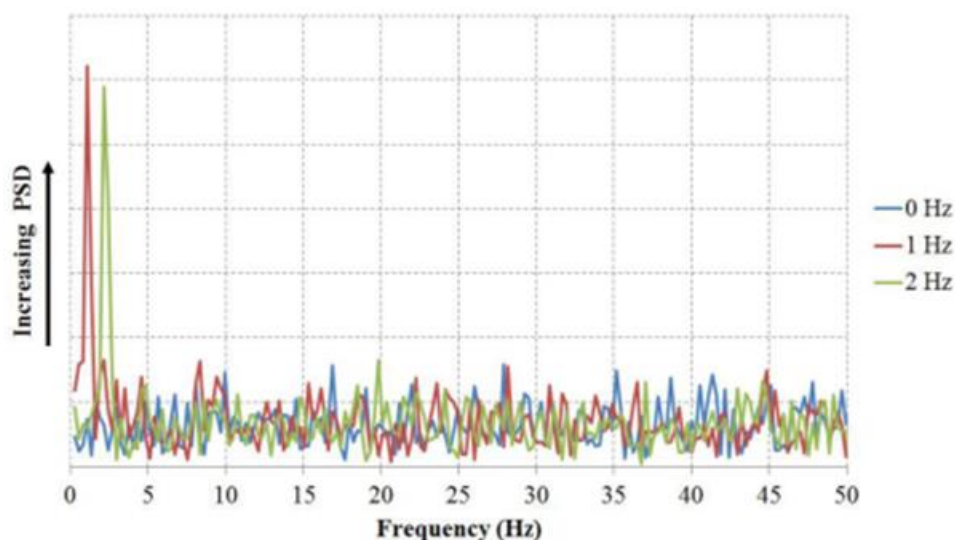


*Figure 57: Spectrum of velocity magnitude at a point located horizontally at hangar and vertically at 1.4 times hangar height with ship oscillating in pitch attitude [81]*

Similar observations were made by Yuan et al. [82] in experimental investigations of flow over a 1/50 scale model of a Canadian Patrol Frigate (CPF) - Figure 58 - where four probes were used to measure airwake velocities on the deck. Isolated single DOF heave, roll and pitch motions at 0.08 Hz with amplitudes of 1.5 m, 3.4° and 5°, respectively, were considered for a headwind condition. Figure 59 shows the frequency spectra of velocity magnitude, and, not surprisingly, exhibits peaks at the ship motion frequency of 0.08 Hz heave and pitch excitation cases. The influence of roll motion on velocity magnitude is minimal, although this may not be the case for oblique WOD conditions, where deck roll attitude can cause the horizontal edges of the deck to effectively act as backward facing steps that can produce large regions of separation [85].



*(a) wind tunnel setup*



*(b) probes used to measure airwake velocities*

*Figure 58: Model of CPF used by Yuan et al. during experimental investigations of influence of ship motion on airwake [82]*

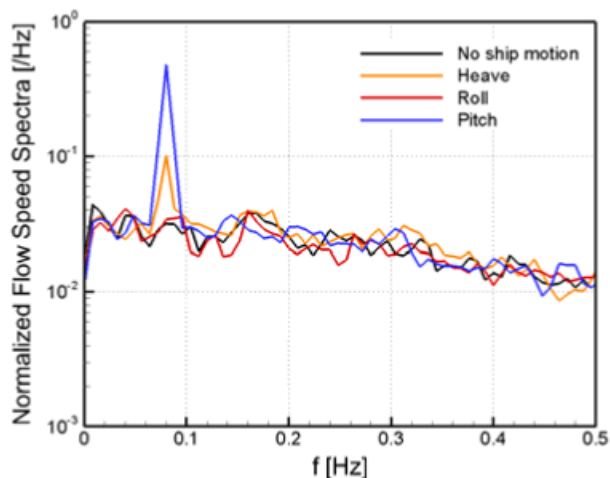


Figure 59: Spectrum of velocity magnitude measured by probe #3 from Figure 58(b) for isolated single DOF deck motion [82]

CFD investigations of the influence of ship motion using an ONR Tumblehome<sup>7</sup> (ONRT) ship geometry (i.e. Figure 60) were undertaken by Dooley et al. [83]. Dooley et al. noted that in addition to inducing deck motion, waves can influence the airwake by “pumping” air onto the ship, and thus performed CFD simulations that accounted for wave motion, wave-induced ship motion, ship forward speed and wind-at-sea [83]. Sea states #3 (SS3) and #6 (SS6) were simulated using wind speeds of 15 kts and 35 kts, respectively, and four simulations are performed for each sea state:

- a) no wave motion and no ship motion included (baseline),
- b) wave motion included but no ship motion (designated as “NM/WW” in figures),
- c) no wave motion but ship motion included (designated as “WM/NW” in figures), and
- d) both wave and ship motions included (designated as “WM/WW” in figures).

For conditions (b) and (d), the wave and ship motions determined from case (d) are superimposed individually in the simulation. The REX CFD code - a dynamic overset, unsteady Reynolds Averaged Navier-Stokes (URANS) or DES/DDES solver with unique capabilities for ship hydro/aerodynamics - was used. Combined heave and pitch motions are considered, and ship motion for one wave cycle is shown in Figure 61, where a wave phase of  $0^\circ$  corresponds to the time instance when the wave crest or peak is at the bow and the wave trough is at  $\lambda = 0.6 L$ , where  $\lambda$  is the wave length and  $L = 154$  m is the ship length. From Figure 61, the magnitudes of pitch and heave motion for SS6 are approximately four times greater than those for SS3. The influence of SS3 motion was found to be minimal [83] and thus, this report will discuss findings from the SS6 simulations.

<sup>7</sup> The ONRT ship represents the research version of the U.S. Navy Zumwalt class destroyer DDG-100, which was designed to have a small radar cross section.

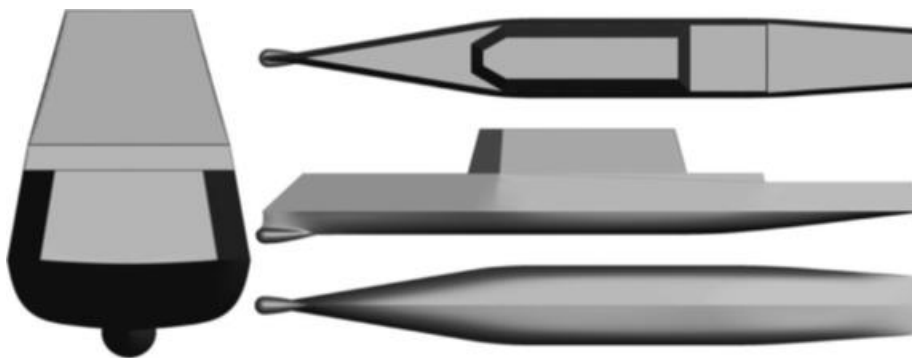


Figure 60: ONR Tumblehome ship geometry [83]

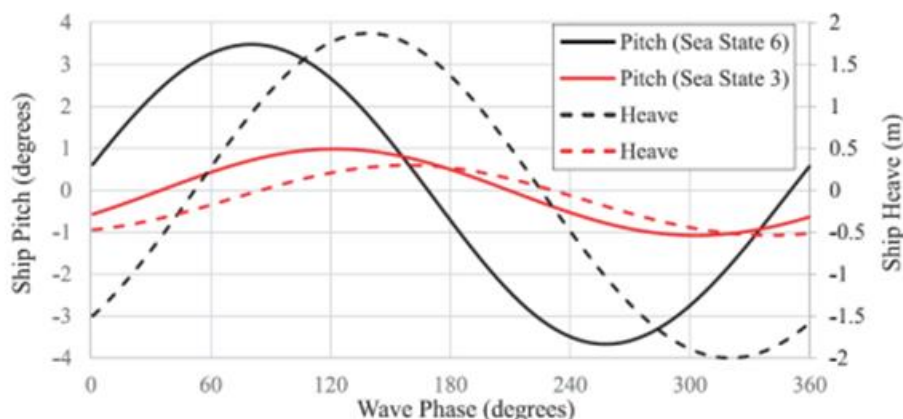


Figure 61: Ship pitch and heave motions through one wave encounter period for SS3 and SS6 [38];  $0^\circ$  corresponds to the instance when the wave crest or peak is at the bow and the wave trough is at  $\lambda = 0.6 L$ , where  $\lambda$  is the wave length and  $L = 154$  m is the ship model length [83]

The influence of ship motion on flow features is depicted in Figure 62, where instantaneous isosurfaces of vortical structures over a period of motion are plotted for SS6 [83]. From the figure, the “V” deck-edge vortices from the bow occur periodically during the cycle, progressively disappearing as the bow reaches motion peaks. The necklace vortex between the hull and superstructure persists during the wave cycle, with its strength decreasing as the bow pitches up and increasing as the bow pitches down. The detached structures on the top and behind the superstructure also persist through the cycle. The recirculation region aft of the superstructure expands when the bow pitches up and contracts when the bow pitches down, indicating a translation of the deck impingement location of the shear layer shed from the superstructure top-aft-edge. Note that this shear layer translation occurs due to ship motion and represents a separate phenomenon from the shear layer flapping [44] described previously.

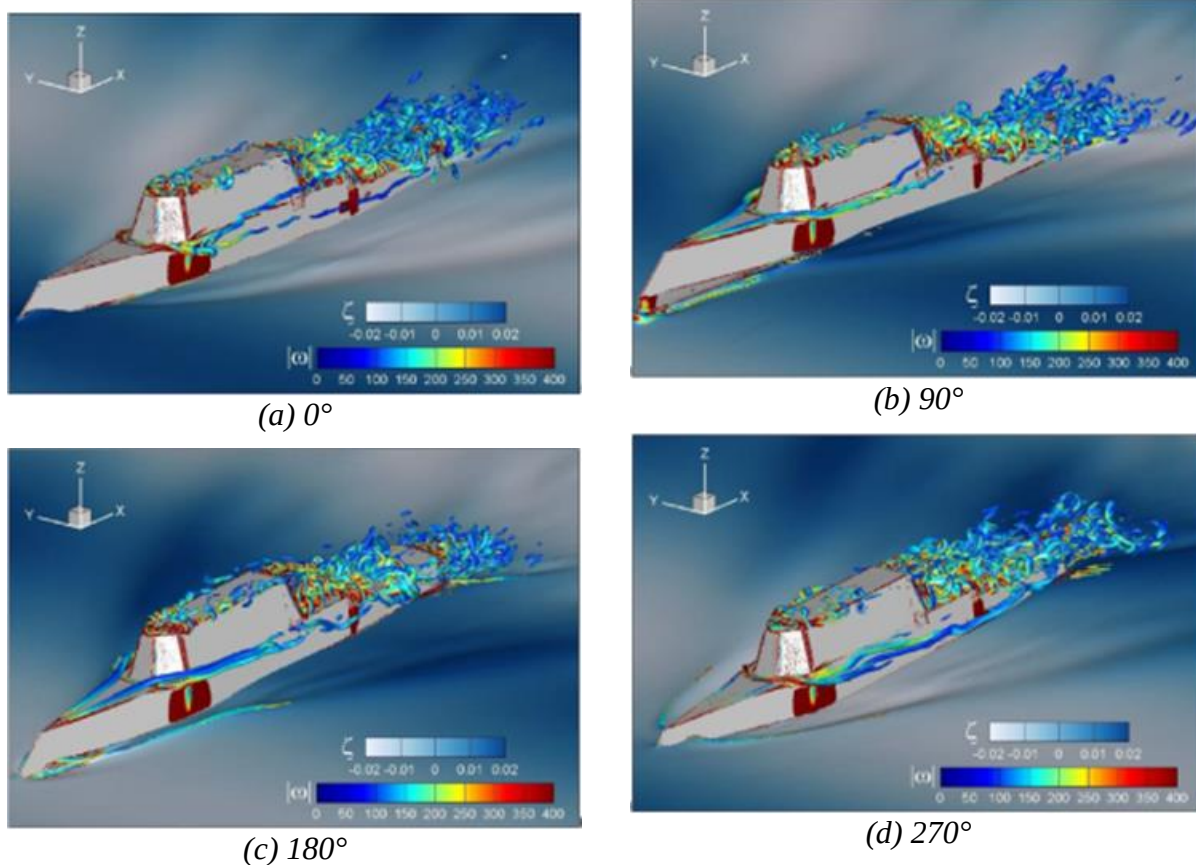


Figure 62: Isosurfaces of vortical structures for ONRT advancing at 15 kts in a 35 kts headwind (SS6) with wave and ship motion;  $0^\circ$  corresponds to the instance when the wave crest or peak is at the bow and the wave trough is at  $\lambda = 0.6 L$ , where  $\lambda$  is the wavelength and  $L = 154$  m is the ship model length [83]

Root-mean-square contours of longitudinal and vertical velocity fluctuations for SS6 are shown in Figure 63, where velocities are nondimensionalized using a speed of  $U_0 = 25$  kts [83]. Ship motion increases the RMS of both velocity components aft of the superstructure and stern when compared to the baseline, most notably for the longitudinal component. The influence of ship motion is further displayed on Figure 64, where phase-averaged deviations in longitudinal  $\Delta u_T$  and vertical  $\Delta w_T$  velocities from baseline mean velocities are plotted over one wave period. Phase-averaged velocities are obtained by time-averaging velocities recorded at specific wave phase angles during the simulation. Large changes of approximately  $\pm 0.25U_0$  are noted immediately aft of the superstructure in the longitudinal velocity component. Closer to center of the flight deck, where piloted operations take place, the deviations in longitudinal and vertical velocities are approximately  $\pm 0.1U_0$  and  $\pm 0.2U_0$ , respectively. Note that the Dirichlet boundary condition used to model the ship geometry results in an upward velocity above the flight deck when the deck moves upward and conversely, generates a downward velocity when it moves downward [83].

Phase-averaged longitudinal and vertical velocities at a point located 30 ft above the center of the flight deck are shown in Figure 65, where the wave encounter period is 8.5 s [83]. The vertical displacement of mid-flight deck location is also shown on the figure. There is a  $\sim 90^\circ$  lag

evident between ship motion and the longitudinal velocity response in the WM/WW and WM/NW cases in Figure 65. The existence of this lag reinforces findings from Sydney et al. [81], discussed above. Peak-to-peak amplitude of the time-averaged longitudinal velocity reaches  $0.25U_0$  relative to the baseline, while that of the vertical velocity reaches  $0.1U_0$  in Figure 66. The vertical velocity response is also noted to be in phase with ship motion.

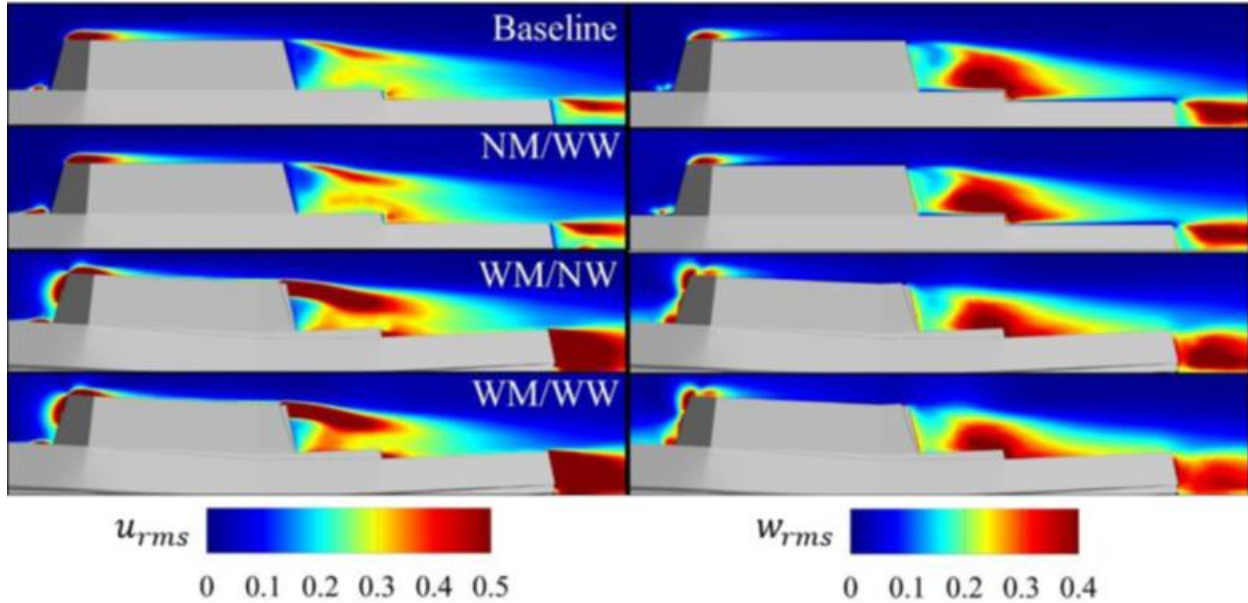


Figure 63: RMS of longitudinal (left) and vertical (right) velocity fluctuations for ONRT advancing at 15 kts in 35 kts headwind; “NM/WW” stands for “no ship motion/with wave motion”, “WM/NW” for “with ship motion/with wave motion”, “WM/WW” for “with ship motion/with wave motion”; velocities are nondimensionalized using an arbitrary speed of  $U_0 = 25$  kts [83]

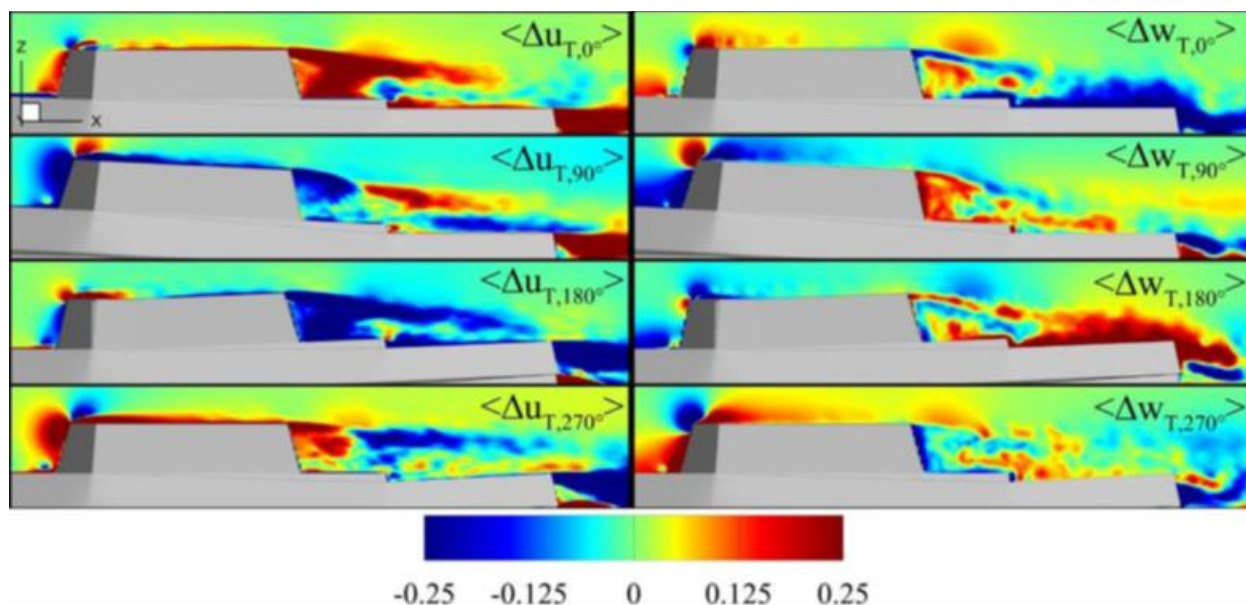


Figure 64: Phase-averaged deviations of longitudinal  $\Delta u_T$  (left) and vertical  $\Delta w_T$  (right) velocities from baseline over one wave cycle period;  $0^\circ$  corresponds to the instance when the wave crest or peak is at the bow and the wave trough is at  $\lambda = 0.6 L$ , where  $\lambda$  is the wave length and  $L = 154$  m is the ship model length; velocities are nondimensionalized using an arbitrary speed of  $U_0 = 25$  kts [83]

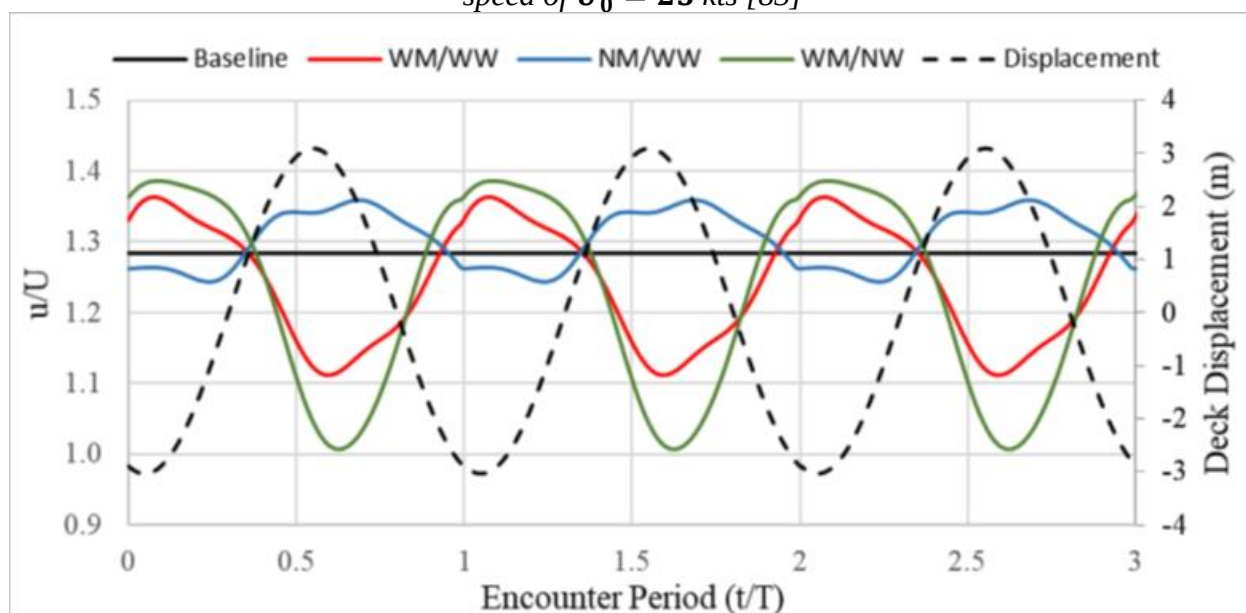


Figure 65: Phase-averaged longitudinal velocity at a point 30 ft above the center of the flight deck of ONRT advancing at 15 kts with a 35 kts headwind (SS6); wave encounter period is 8.5 s [83]

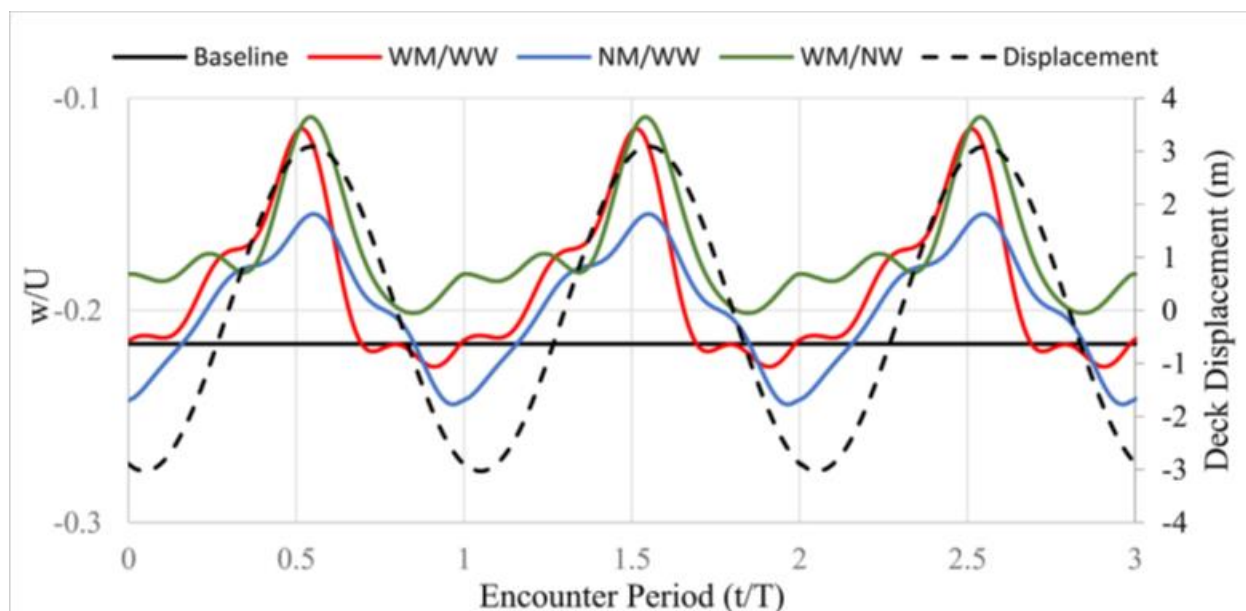


Figure 66: Phase-averaged vertical velocity at a point 30 ft above the center of the flight deck of the ONRT advancing at 15 kts with a 35 kts headwind (SS6); wave encounter period is 8.5 s [83]

Dooley et al. also investigate the influence of ship motion on helicopter dynamics using one-way and two-way coupled simulations [83]. One-way coupled simulations are performed using 3 scaled versions of a simplified SH-60 Seahawk rotor: a full-scale rotor with  $D = 16.35$  m, a 50% scaled down rotor representative of a Robinson R22 rotor, and a 4.5% scaled down rotor representative of a drone. Simulations are performed with the rotor hovering 30 ft above the center of the flight deck. Time histories of rotor thrust are shown in Figure 67. All three rotors exhibit a degree of periodicity. The 0.045D rotor is noted to be most sensitive to the airwake, producing fluctuations with the largest magnitudes in Figure 67. In the context of the present ONR effort, this finding indicates the size of the rotor is an important factor to take into consideration when identifying airwake features that influence vehicle dynamics. The corresponding thrust frequency spectra for the three rotors are shown in Figure 68, where  $f_e$  is the wave encounter frequency. Thrust fluctuations for all three rotors are periodic with the wave encounter frequency  $f_e$ . Peaks are also evident at  $2f_e$ .

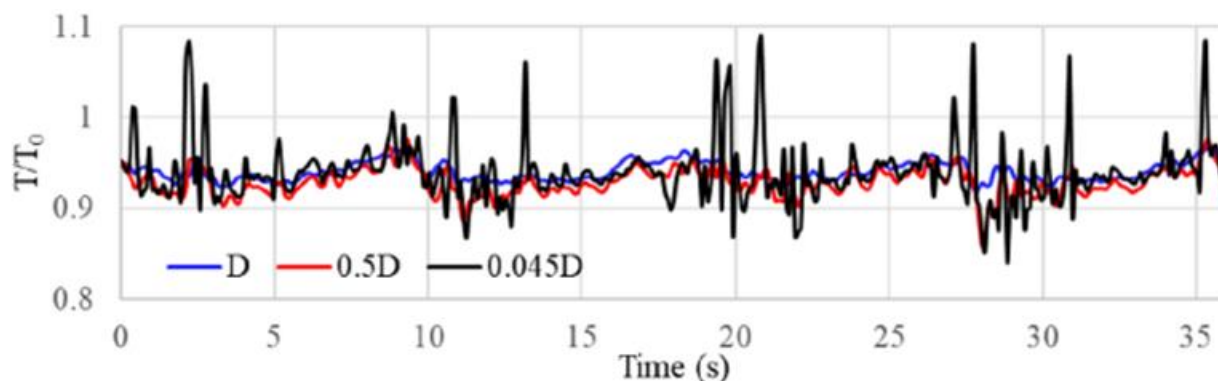


Figure 67: Instantaneous thrust produced by three scaled versions of a simplified SH-60 helicopter rotor model located 30 ft above the center of the flight deck of the ONRT ship advancing at 15 kts with a 35 kts headwind (SS6) during one-way coupled simulations [83]

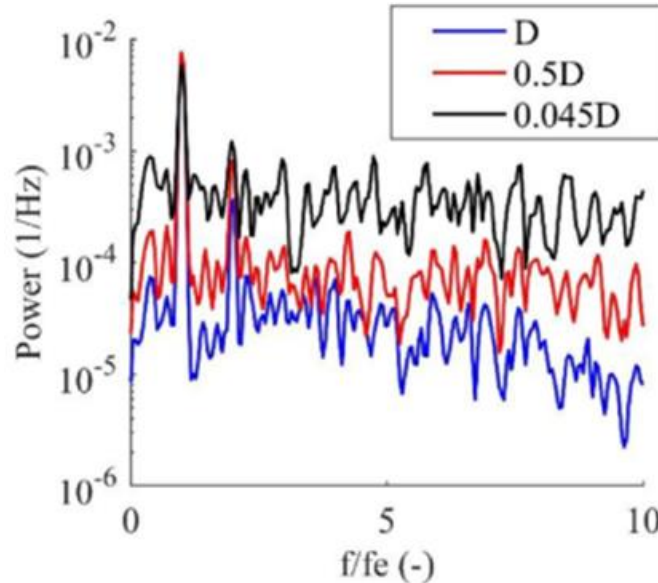


Figure 68: Frequency response function corresponding to thrust time-histories in Figure 67;  $f_e$  is the wave encounter frequency [83]

Two-way coupled simulations were performed using a generic helicopter loosely modeled after the SH-60 Seahawk helicopter [83]. The helicopter model had a forward speed of 15 kts to hold hover 30 ft above the center of moving ship deck (SS6). Trim was not performed, and the aircraft was held stationary above the deck using constant main and tail rotor collective inputs only. Time histories of the vertical force generated by the helicopter are shown in Figure 69 for three cases: isolated helicopter, helicopter hovering above advancing ship with no motions, and helicopter hovering above advancing ship deck with motion. The black line indicates instantaneous values while the red line represents a moving-average over one main rotor rotation. Ship motion causes the vertical force to change in a sinusoidal manner with peak changes of approximately 5 kN, relative to the 85 kN mean baseline. Note that these changes are a combined result of periodic ship airwake and dynamic ground effect from the moving deck, where part of the helicopter rotor downwash is redirected back towards to the rotor. Power spectral density plots of the vertical force shown in Figure 70 reveal periodicity with the wave encounter frequency  $f_e$ . Note that the CFD results presented by Dooley et al. [83] were not validated against experiments or full-scale test data.

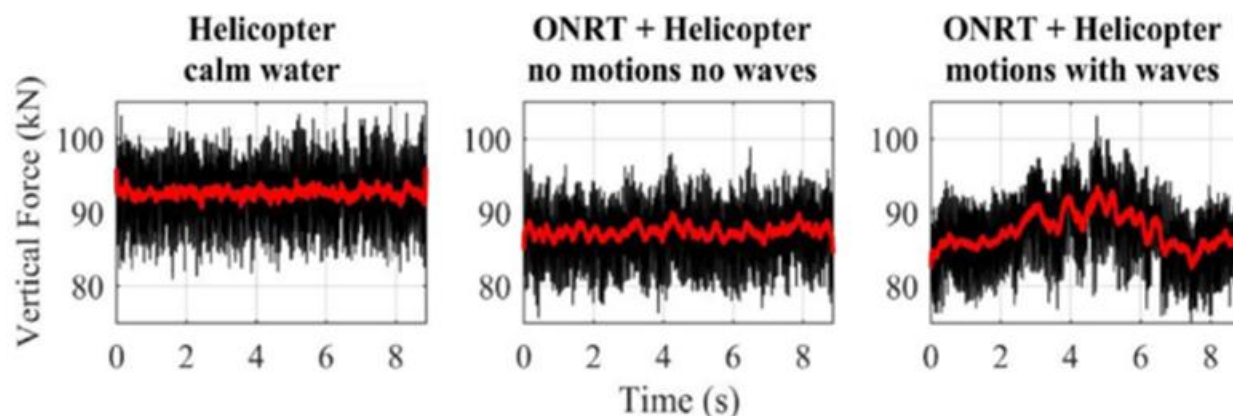


Figure 69: Time-histories of vertical force generated by a simplified SH-60 helicopter model located 30 ft above the center of the flight deck of the ONRT ship advancing at 15 kts with a 35 kts headwind (SS6) during two-way coupled simulations; black line indicates instantaneous values and red line represents a moving-average over one main rotor rotation [83]

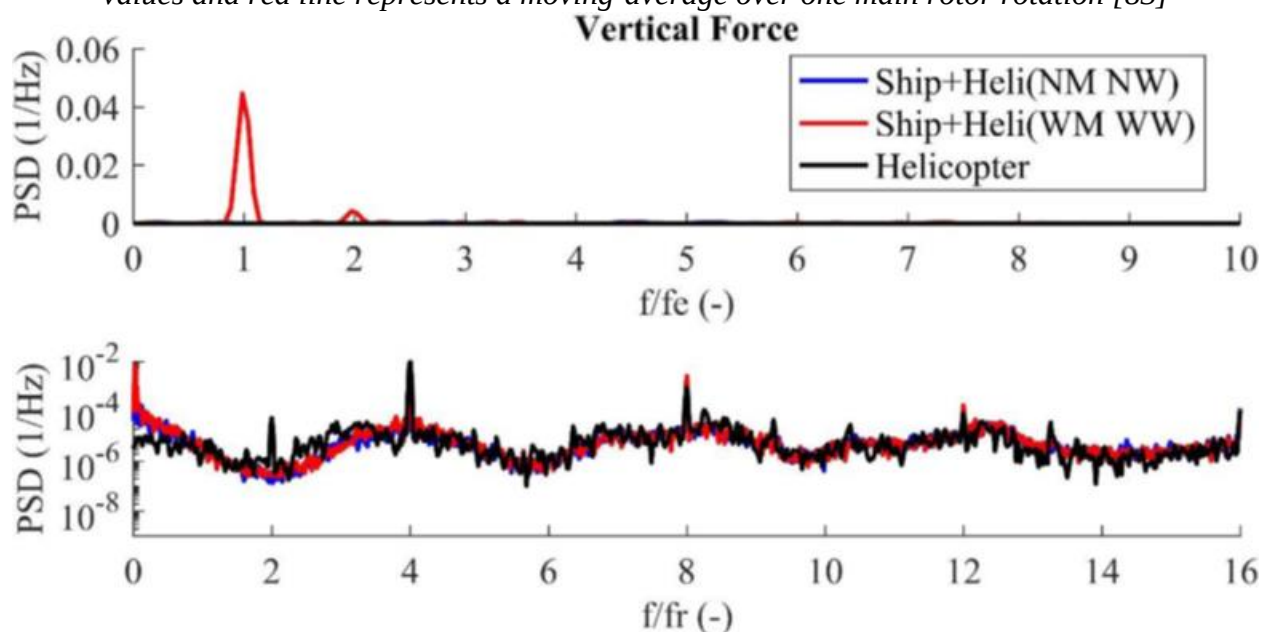


Figure 70: Frequency response functions corresponding to vertical force time-histories in Figure 69;  $f_e$  is the wave encounter frequency, and  $f_r$  is the main rotor rotational frequency [83]

Experimental investigations of the influence of ship motion on ship airwake were performed by Krebill [84] using a 1/77 scale model of the ONRT ship geometry in a water channel as shown in Figure 71. Heave and pitch motions were considered. These motions were applied at the CG of the ship model. PIV measurements were made for both static and dynamic ship configurations along the planes shown in Figure 72. Laser Doppler velocimetry (LDV) measurements at the point indicated in Figure 72 within PIV domain was used to verify the PIV measurements. Sea states #3 to #6 were investigated using different combinations of motion amplitude, frequency, and periods as indicated in Table 1. From the mean longitudinal and vertical velocity contours for the four sea states shown in Figure 73 and Figure 74, respectively, flow features remain similar between the sea states [84]. In this report, results from SS5 investigations will be considered.

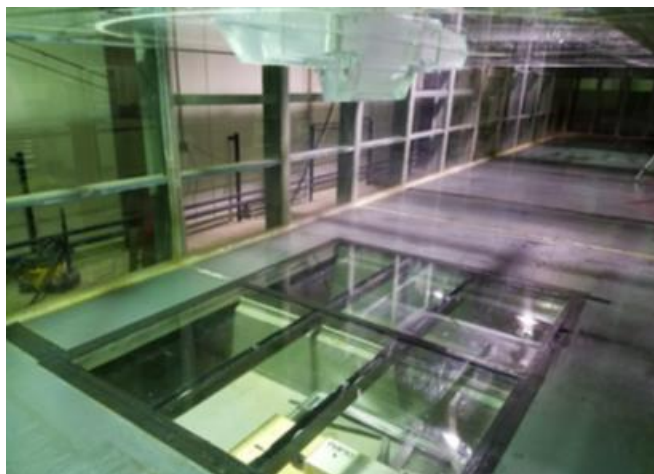


Figure 71: ONRT model in flume channel [84]

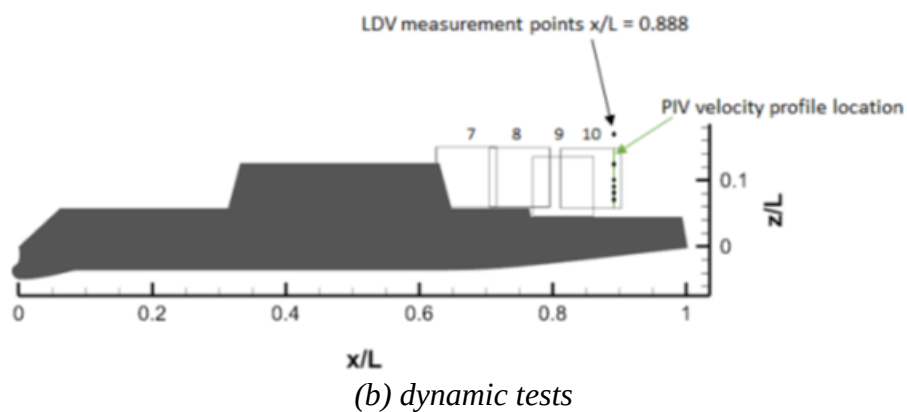
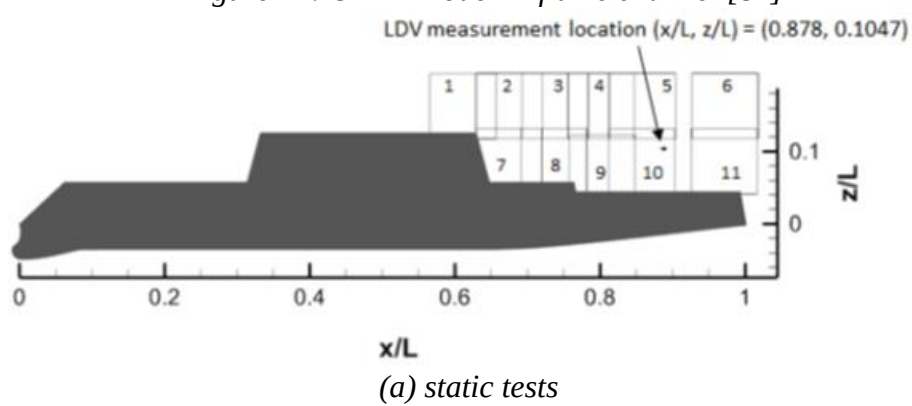


Figure 72: PIV and LDV measurement locations [84]

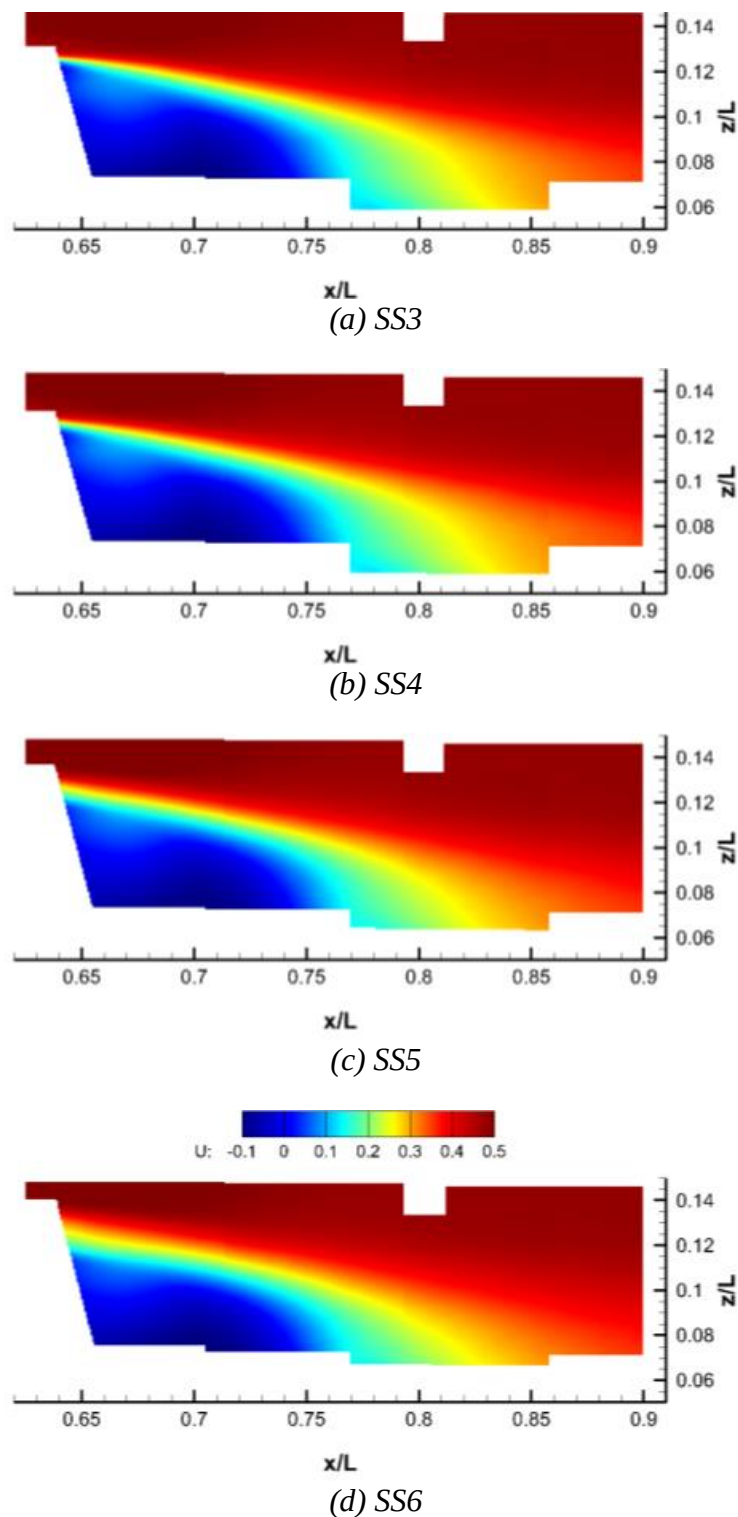


Figure 73: Mean longitudinal velocity contours recorded during water channel experiments for SS3 to SS6; velocity values are given in meters-per-second [84]

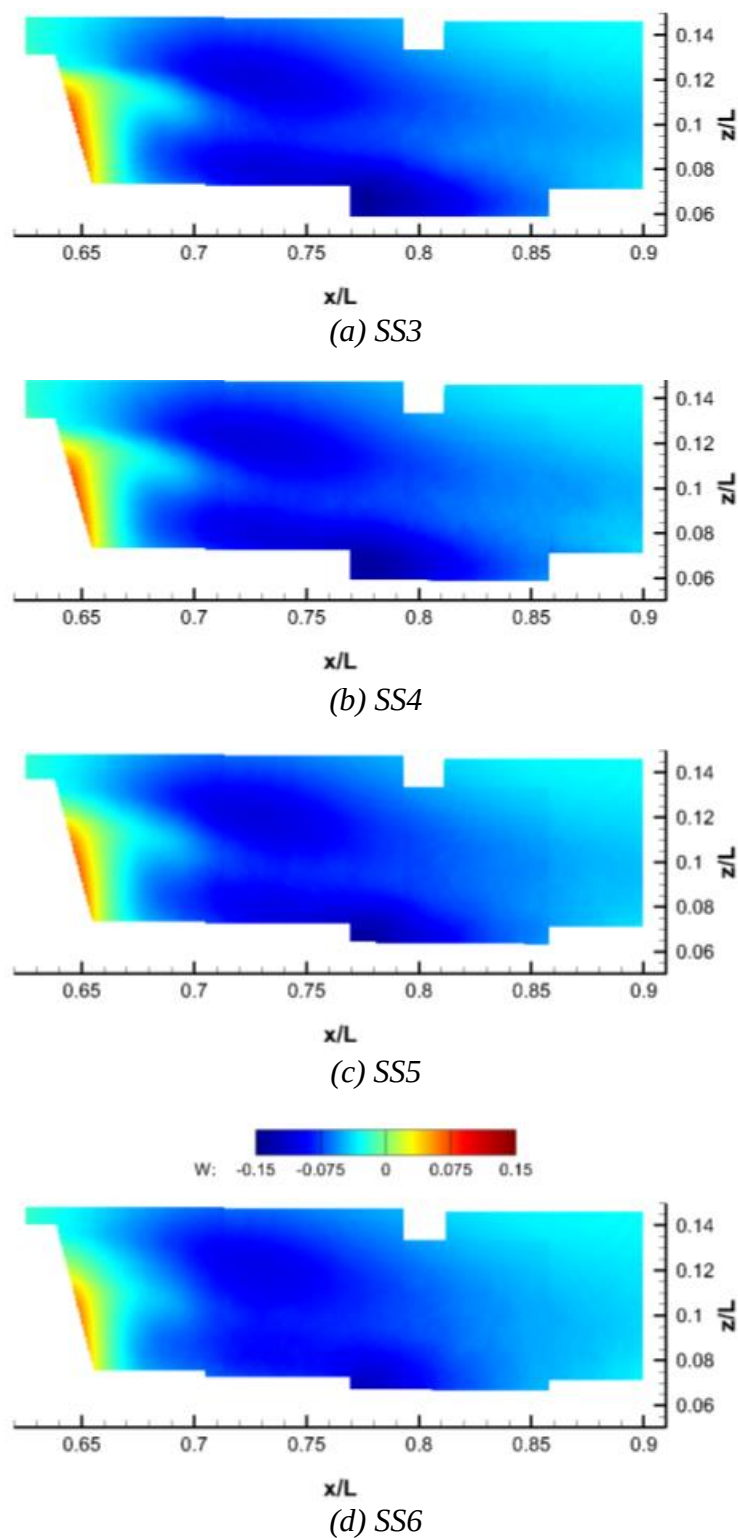


Figure 74: Mean vertical velocity contours recorded during water channel experiments for SS3 to SS6; velocity values are given in meters-per-second [84]

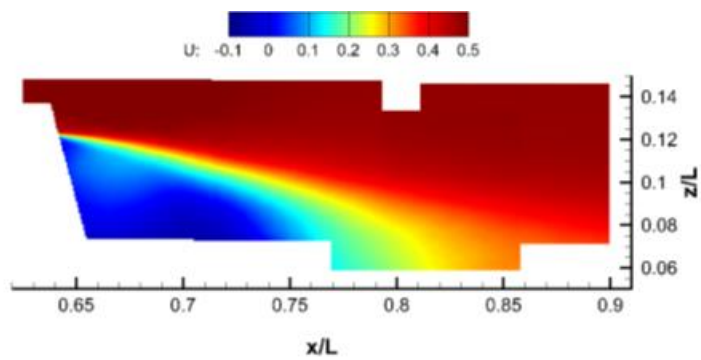
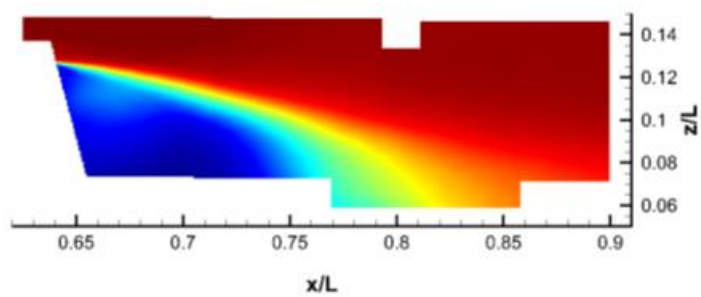
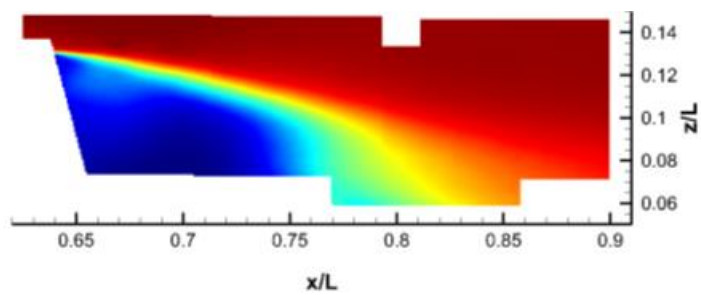
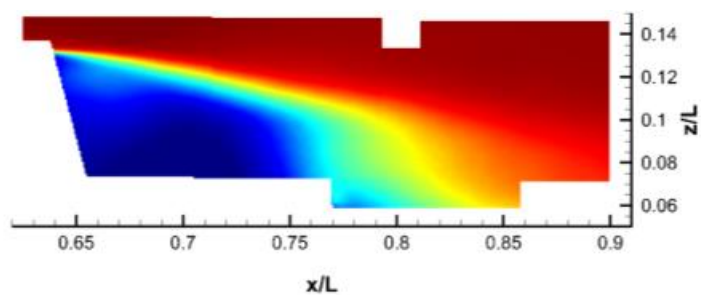
Table 1: Test matrix used to emulate sea states #3 - #6 in Ref. [84]

| <b>Sea-State</b>             |                               | <b>SS3</b> | <b>SS4</b> | <b>SS5</b> | <b>SS6</b> |
|------------------------------|-------------------------------|------------|------------|------------|------------|
| <b>Motion Frequency (Hz)</b> |                               | 0.33       | 0.28       | 0.26       | 0.17       |
| <b>Motion Period (s)</b>     |                               | 3.00       | 3.53       | 3.78       | 5.93       |
| <b>Heave</b>                 | <b>Amplitude (m)</b>          | 0.00527    | 0.0082     | 0.0153     | 0.025      |
|                              | <b>Offset from mean (m)</b>   | -0.00141   | -0.00141   | -0.00139   | -0.00147   |
| <b>Pitch</b>                 | <b>Amplitude (m)</b>          | 1.07       | 1.50       | 2.49       | 3.58       |
|                              | <b>Offset from mean (deg)</b> | -0.0734    | -0.0783    | -0.0947    | -0.141     |
|                              | <b>Lead (deg)</b>             | 37.2       | 43.3       | 53.3       | 59.0       |

Mean longitudinal velocity contours at different phase angles<sup>8</sup> of ship motion are shown in Figure 75 [84], and illustrate that the shear layer impingement location translates up and down in the  $0.77 \leq x/L \leq 0.85$  region of the deck during the motion. Mean vertical velocity contours at different phase angles of ship motion are shown in Figure 76 [84]. From the figure, the shear layer shed from the superstructure top-aft-edge juts into the recirculation region for phase angles  $45^\circ$  and  $90^\circ$ . The shear layer then starts receding to its original location from  $135^\circ$  onwards. Also evident from the figure is existence two distinct downwash regions separated by a smaller magnitude downwash region in the  $90^\circ$  and  $135^\circ$  phase contour plots, which are a consequence of the ship model changing directions.

Similar to the previous studies discussed in this section, Krebill observed airwake velocities at a position  $(x/L, z/L) = (0.888, 0.110)$  (see Figure 72) to be periodic with wave encounter frequency  $f_e$  for all sea states considered as shown in Figure 77 [84]. For SS5 and SS6, a secondary peak is evident at  $2f_e$ , which the author attributes the secondary peak to the measurement location going in and out of the two vertical velocity downwash regions shown in Figure 76 for phase angles of  $90^\circ$  and  $135^\circ$ . Krebill also examines mean velocities measured for static ship orientations corresponding to different phases of ship motion and notes larger velocity fluctuations in the static measurements, implying static tests provide conservative estimates of the influence of ship motion [84].

<sup>8</sup> Note that a definition of ship motion in terms of phase angle is not provided by the author. However, based on the translation of the shear layer impingement location in Figure 75, it is probable that motion is defined such that the ship bow pitches downward and the deck upwards between  $0^\circ$  and  $180^\circ$ , and in the opposite directions between  $180^\circ$  and  $360^\circ$ , with the switch in directions occurring at  $90^\circ$  and  $270^\circ$ .

(a)  $0^\circ$ (b)  $45^\circ$ (c)  $90^\circ$ (d)  $135^\circ$

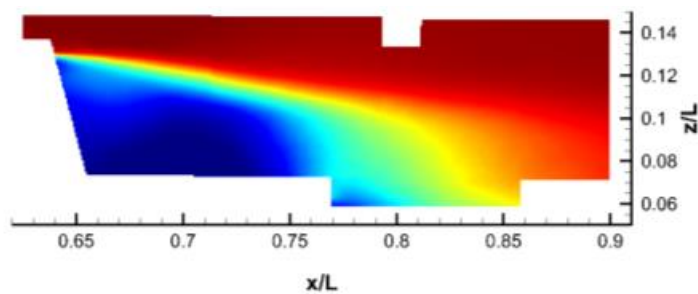
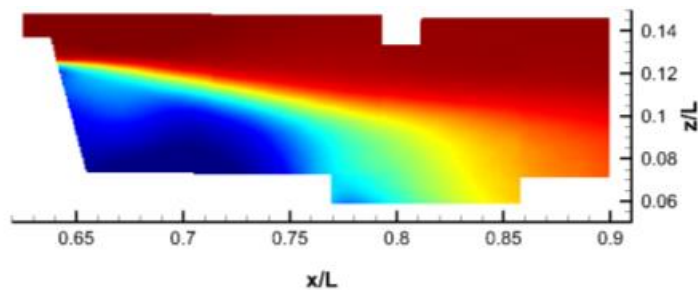
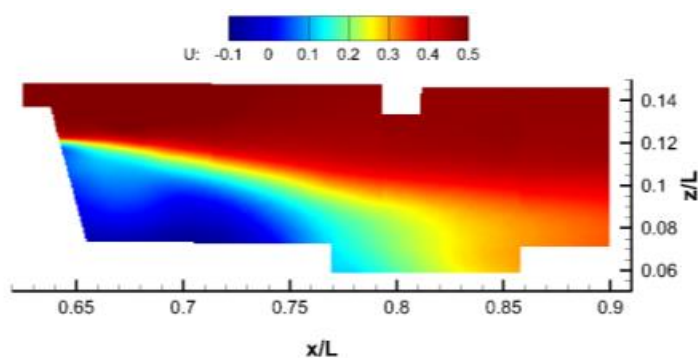
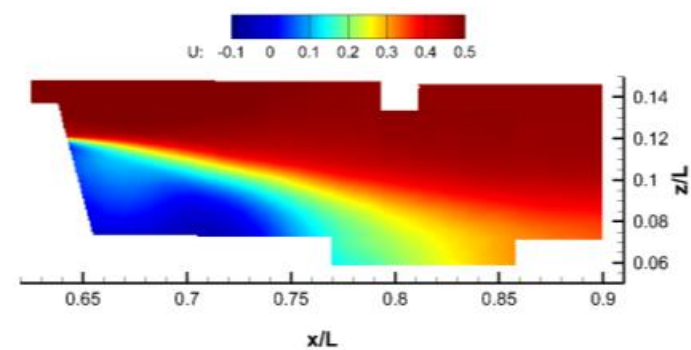
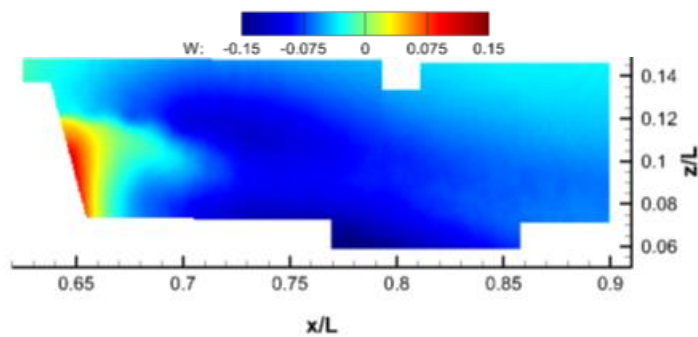
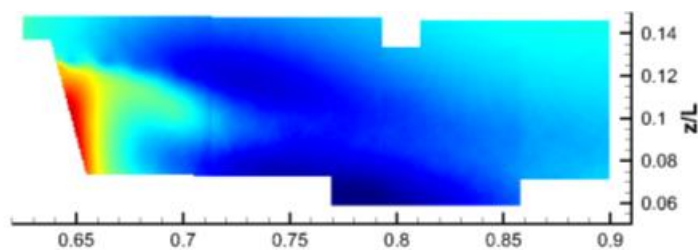
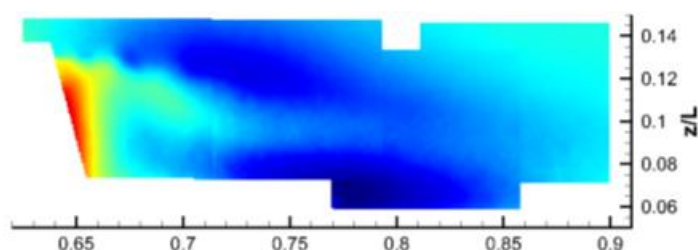
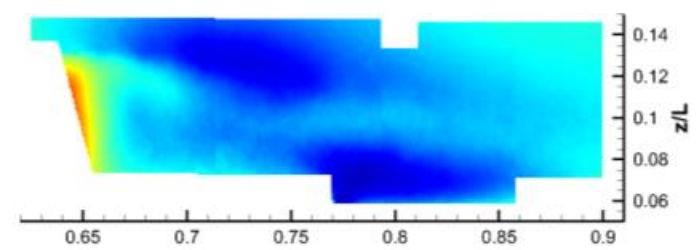
(e)  $180^\circ$ (f)  $225^\circ$ (g)  $270^\circ$ (h)  $315^\circ$ 

Figure 75: Mean longitudinal velocity contours for different phase angles of ship motion; velocity values are given in meters-per-second [84]

(a)  $0^\circ$ (b)  $45^\circ$ (c)  $90^\circ$ (d)  $135^\circ$

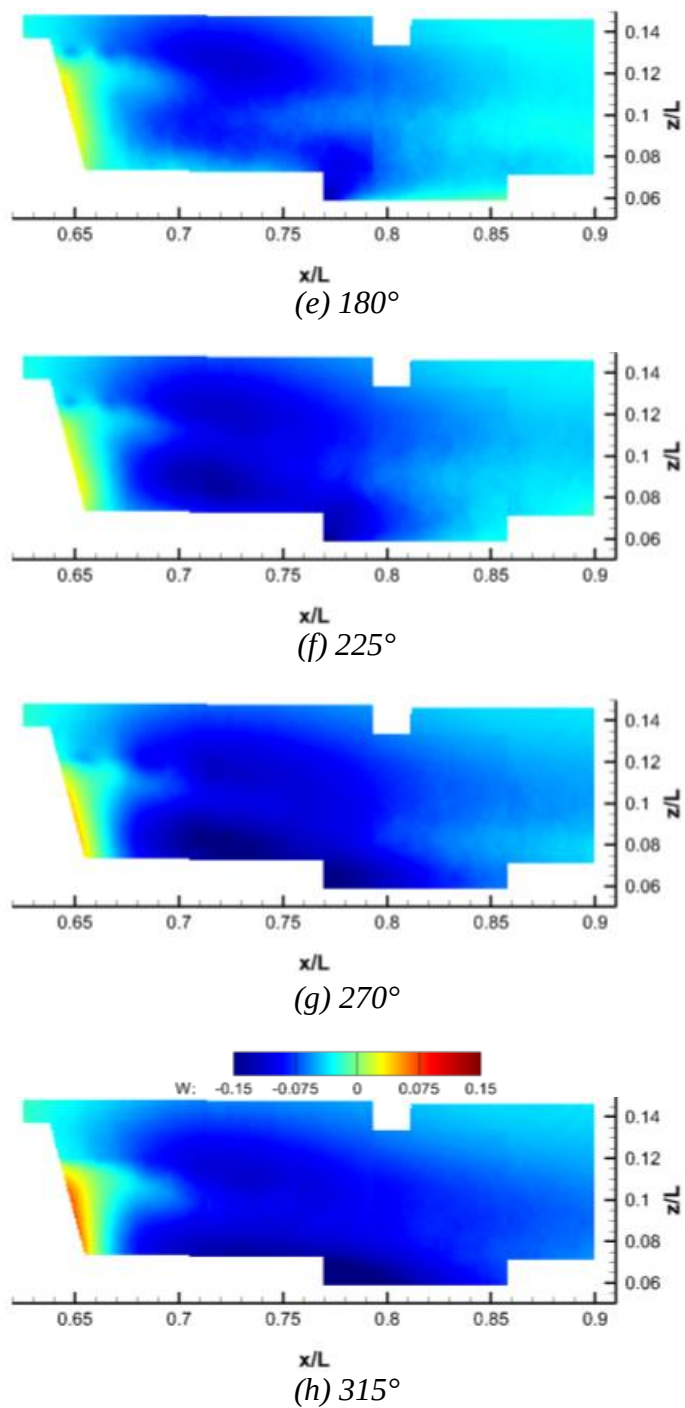
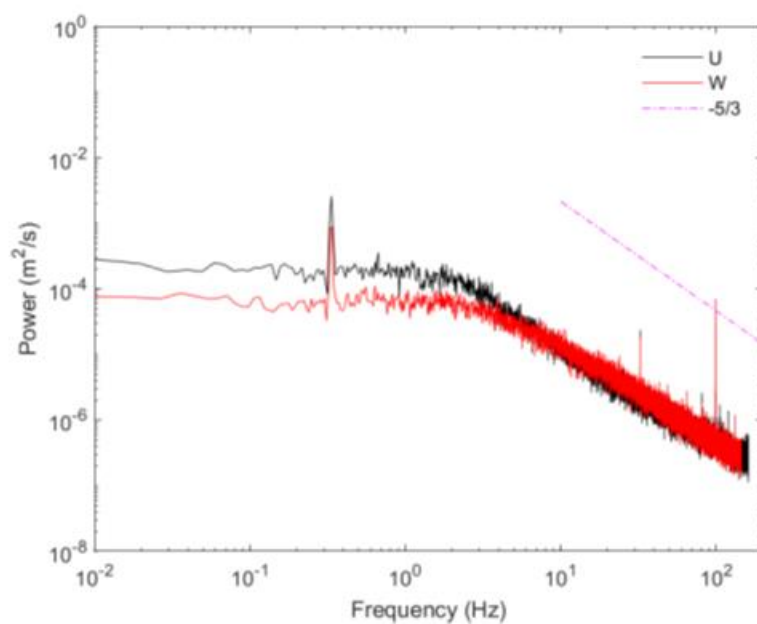
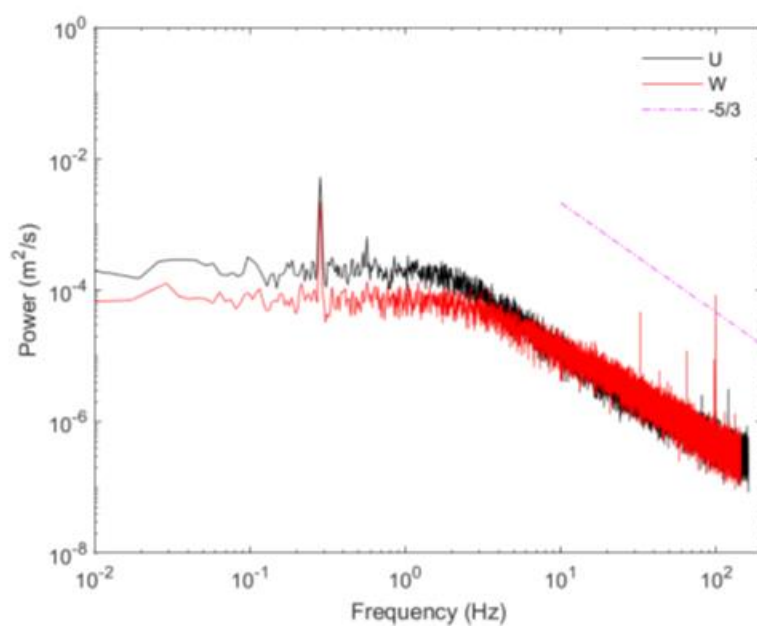


Figure 76: Mean vertical velocity contours for different phase angles of ship motion; velocity values are given in meters-per-second [84]



(a) SS3



(b) SS4

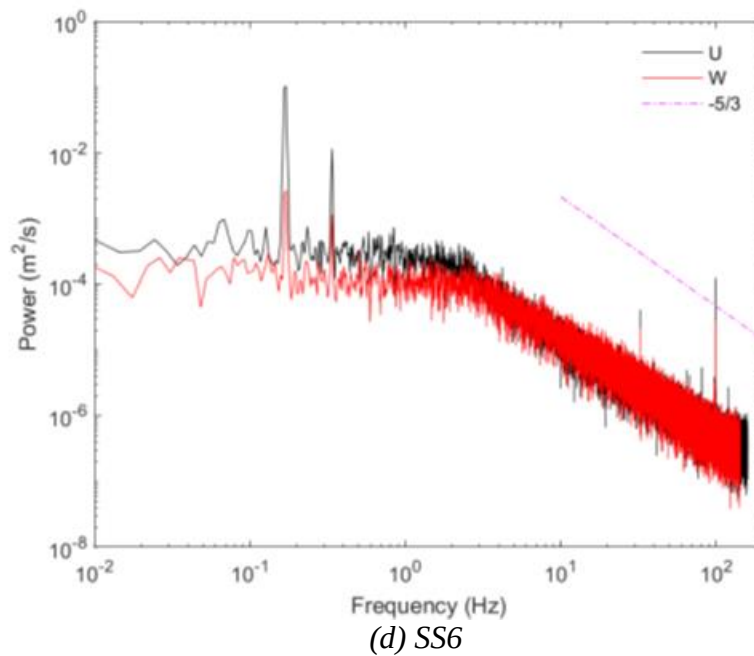
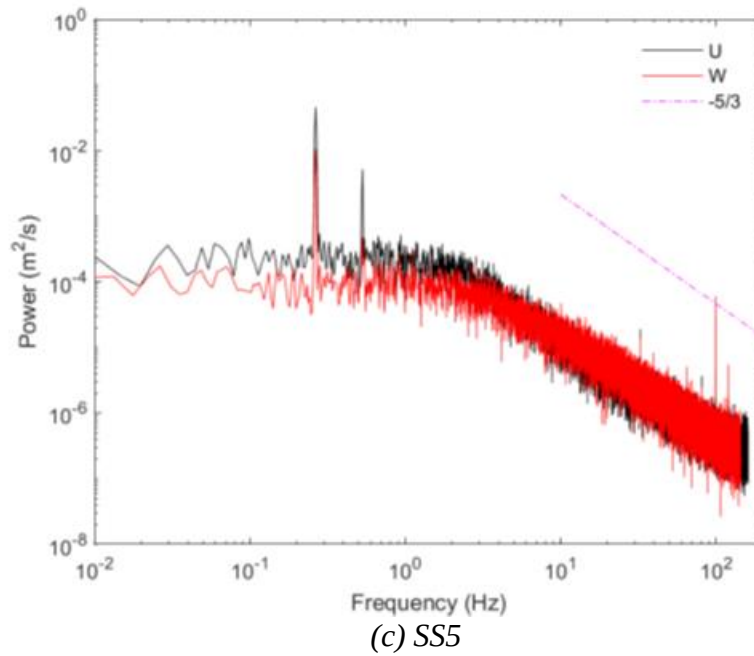


Figure 77: Frequency spectra of longitudinal and vertical velocities recorded at  $(x/L, z/L) = (0.888, 0.110)$  (see Figure 72) during water channel experiments [84]

#### Wake Parametrization

Work has been previously performed at CDI to parametrize fluctuating velocity components in the airwake of a landing helicopter assault (LHA) class ship to allow for predictions of flow unsteadiness to circumvent computational and data storage limitations associated with CFD

Distribution Statement A

Approved for public release: distribution unlimited

based solutions [86]. The approach used a stochastic representation of the unsteady velocity components and a data reconstruction algorithm that preserved the spatial correlations relationships that resulted due to vortex structures in the ship airwake. Realistic velocity fluctuation predictions were obtained from the resulting model when compared to those obtained via CFD. The model however, was dependent on previously obtained CFD data and thus dependent on the configuration under consideration. A configuration independent methodology was proposed in an early publication by Woofield et al. [87], where flow over the edges and faces of a ship structure were first characterized as one of three fundamental flow patterns depending on the incoming wind direction: corner flow (further categorized as separated or vortex corner flow), lee rotor flow and profile drag flow, illustrated in Figure 78. Each flow pattern was associated with a dataset of velocity increments which could be accessed for a given edge/surface and wind-over-deck angle. A simplified standard deviation-based turbulence model was also introduced but its influence during piloted simulation could not be assessed due to unforeseen simulation issues. The generic airwake model showed good agreement with wind tunnel velocity data in the transverse plane but discrepancies were noted in the streamwise plane. Despite the lack of turbulence, the authors note that favorable responses were obtained from pilots during helicopter-shipboard simulations with the generic model and emphasize the need for further testing of the methodology with more representative turbulence models.

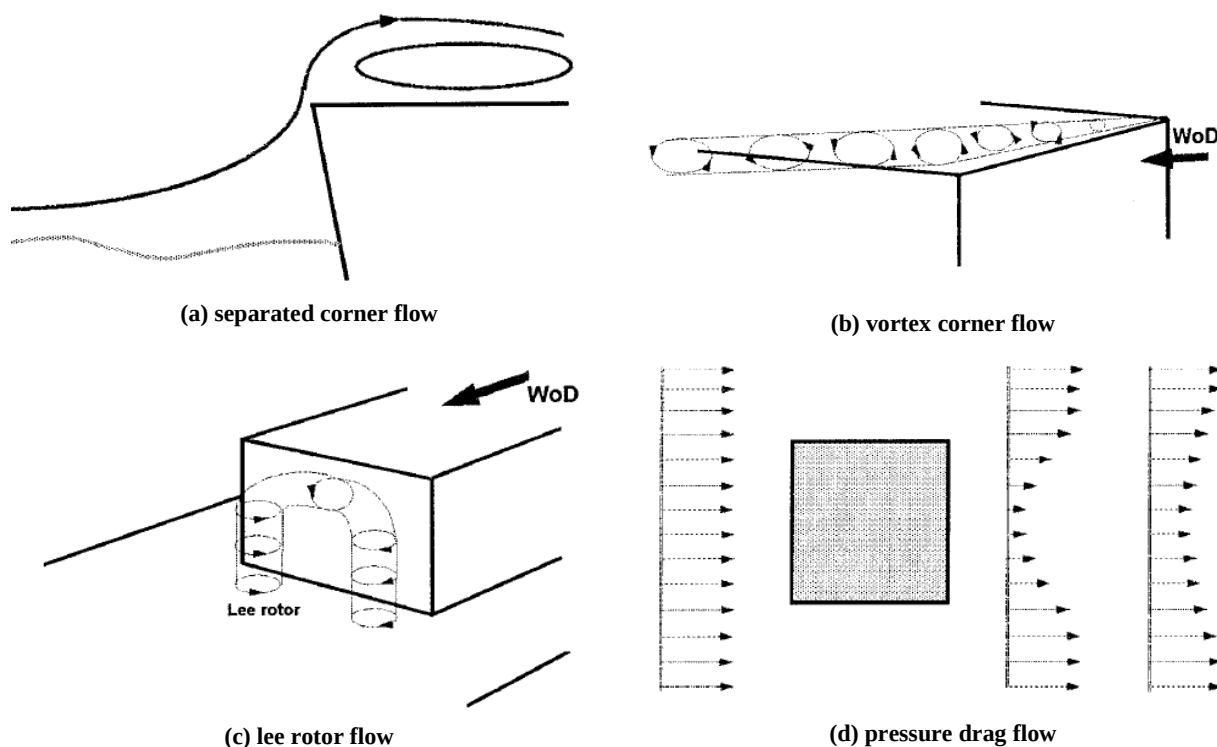


Figure 78: Flow patterns used to parametrize airwake in Ref. [87].

Generic airwake models have also been derived for flow over urban settings (e.g. Ref. [88]) and further review of relevant literature is needed at this point to identify parametrization techniques suitable for the project.

## *Simplified Wake Representation*

### CFD Simulation and Validation

CFD simulations were undertaken of the flow around a cube in order to develop a dataset against which to compare analytical models. Flow over cube was chosen as it represents a minimum complexity canonical obstacle, and simulation results were compared to those from literature for validation. CFD predictions were performed using the CDI CGE/VorTran-M flow solver [12, 89, 90]. CDI's adaptive cut-cell octree Cartesian Grid solver (CGE) solves the compressible URANS equations using either a Spalart-Allmaras (SA and SA DES) or a shear stress transport (SST) turbulence models. Modern turbulence specific adaptive wall functions are available in CGE in order to efficiently represent the flow in boundary layers. Resolving the evolution of vortical structures, such as those shed from bluff bodies, can be particularly expensive in primitive variable CFD solvers; therefore, CGE is coupled to the VorTran-M vorticity-velocity solver in order to reduce computational costs. The CGE/VorTran-M coupling arrangement consists of implementing the CGE solution on the grid as an “inner” solver that extends sufficiently far to contain regions with significant compressibility or flow turbulence and VorTran-M as an “outer” solver that is applied in the essentially incompressible flow region and efficiently transports and evolves the vorticity with minimal dissipation. Such an arrangement allows for computational savings in mesh resolution of several orders of magnitude when compared to pure CGE simulations.

Primary flow features associated with a headwind flow over a cube are illustrated in Figure 9, and include

- A horseshoe vortex that forms upstream of the obstacle where the flow stagnates and wraps around the cube (feature “A” on the figure);
- On the top and side surfaces, local separation and reattachment regions (“B” and “C” on the figure);
- In the lee of the obstacle, an arch vortex (feature “D” on the figure) forms due to the separation of the shear layers from the top and side surfaces. The streamlines aft of the obstacle indicate a recirculation region, where the direction of the flow reverses.

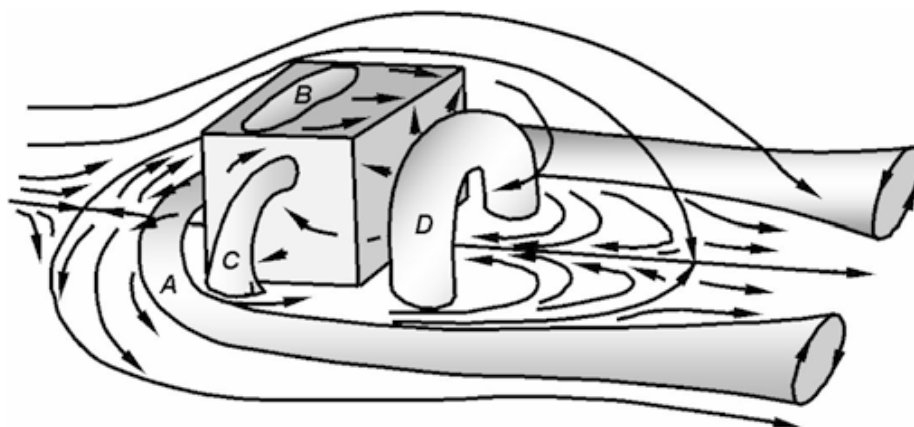


*Figure 79: Schematic of principal flow features around surface-mounted obstacle [55].*

CGE/VorTran-M simulations were performed for a 30 m/s headwind simulation over a 10 m cube. Although this speed is larger than would be typically encountered in practice, it is necessary to use a larger freestream velocity to maintain good numerical conditioning of the

compressible flow solver. An alternative approach would have been to take advantage of the preconditioners implemented in CGE for low-speed flows, however, we deemed using the higher velocity appropriate given that the flow field is dominated by separation from the sharp edges of the cube, which has a weak dependence on viscosity. This further allowed us to perform simulations for inviscid flow only as in [12] since neglecting viscosity effects allows for scaling of time-accurate flow predictions using freestream velocity. Note that no explicit atmospheric boundary layer (ABL) model was included in the simulation, though this option is available. Simulations were performed using a fixed time step of 0.005 seconds, and, after washing out the initial conditions, a transient simulation was of 25 seconds was undertaken. Results were post-processed based on the last 22 seconds of the simulations. The first three seconds of the prediction were removed in case there were any residual non-physical transient effects.

Vortex shedding frequency is characterized by the Strouhal number, which is defined as

$$St = \frac{fb}{U}, \quad (8)$$

where  $f$  is the vortex shedding frequency and  $b$  is the characteristic length. Results from literature indicate dominant periodicity for flow aft of a cube at a distance of  $6H$  [57] to have a characteristic frequency of  $St = 0.104$  [57, 91] (compared to  $St = 0.2$  for a flat plate). Using the 10 m cube length as characteristic length and the freestream velocity of 30 m/s as characteristic velocity, this Strouhal number corresponds to a vortex shedding frequency  $f = 0.3$  Hz, which, in turn corresponds to a period of 3.2 seconds. The trimmed 22 second simulation duration thus allows for approximately 7 cycles of vortex shedding to be captured, with approximately 640 time steps per cycle.

The qualitative flow features described previously are captured by the CGE/VorTran-M simulations as shown in Figure 80(a) to (d). The origin of the global coordinate system shown in the figures is located at the center of the bottom face of the cube. In Figure 80(a), the separation region over the top of the cube surface and recirculation zone aft of the obstacle are evident. The velocities in the recirculation zone reach magnitudes close to 15 m/s, which is half of the freestream velocity. Figure 80(b) displays the horseshoe vortex which starts upstream of the obstacle. Longitudinal velocity contour plots along a transverse plane that passes through the center of the obstacle at  $t = 4.7s$  and  $t = 6.7s$  are shown in Figure 80(c) and 2(d), respectively. Flow separation on the sides of the obstacles is evident in these figures. An alternating vortex shedding pattern representative of the classic von Karman streak is noted as well.

Spectral analysis of longitudinal velocity and velocity magnitude were performed for three points aft of the cube to obtain a quantitative validation of the simulation. These points correspond to  $z = \{H, 2H, 3H\}^T$  and  $y = 0.5H$ , where  $H = 10$  m is the cube height. Velocity magnitude was considered to account for 3D flow effects in characterizing vortex shedding frequency. Time histories and Fourier Transform plots for longitudinal velocity and velocity magnitude are shown in Figure 81 and Figure 82, respectively. The dominant frequency in all spectral plots is 0.5 Hz. Secondary peaks are noted at 0.4 Hz in the  $z = 2H$  and  $z = 3H$  plots. The actual frequency is thus expected to be between 0.4 and 0.5 Hz. Using the cube length of 10 m as characteristic length, and the freestream velocity of 30 m/s as characteristic velocity, this corresponds to characteristic frequency of approximately 0.15. There is thus a minor discrepancy between the Strouhal frequency obtained from the CGE/VorTran-M simulation and that from the literature, but is deemed acceptable for the current investigation which is focused on phenomenology and developing simplified representations.

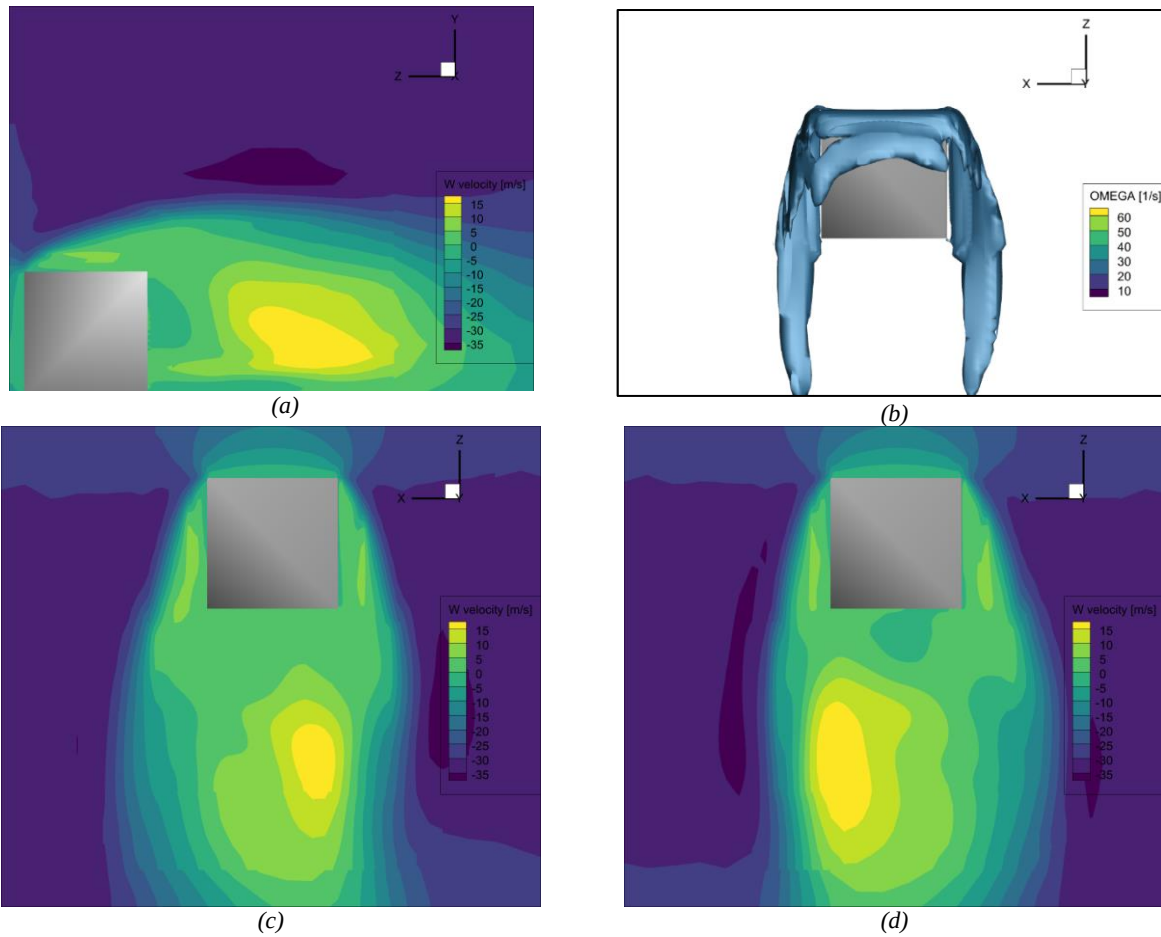


Figure 80: Flow features associated with a 30 m/s headwind flow simulation over a 10m cube using CGE/VorTran-M: (a) contour of longitudinal velocity through midplane (side view), (b) iso-surface of vorticity magnitude (top view), (c) contour of longitudinal velocity on a midplane at  $t = 4.7$  s (top view) and (d) contour of longitudinal velocity on a midplane at  $t = 6.7$  s (top view).

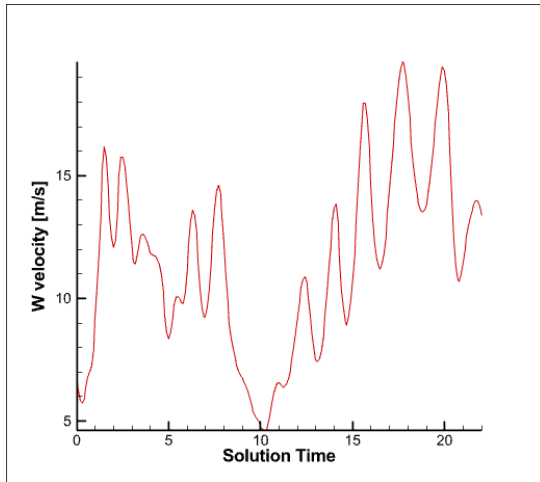
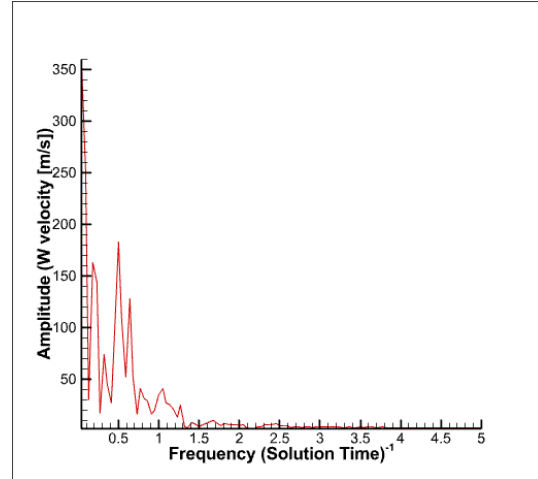
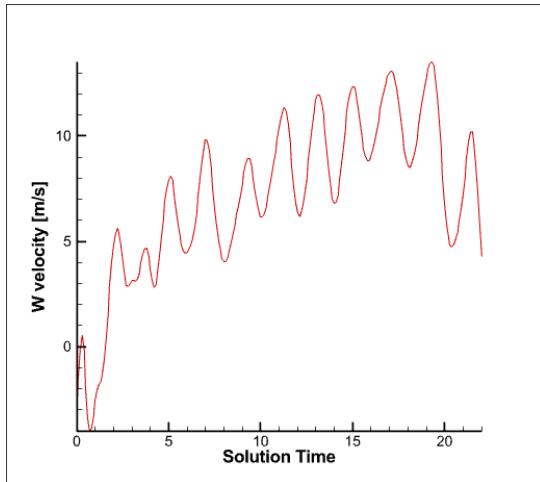
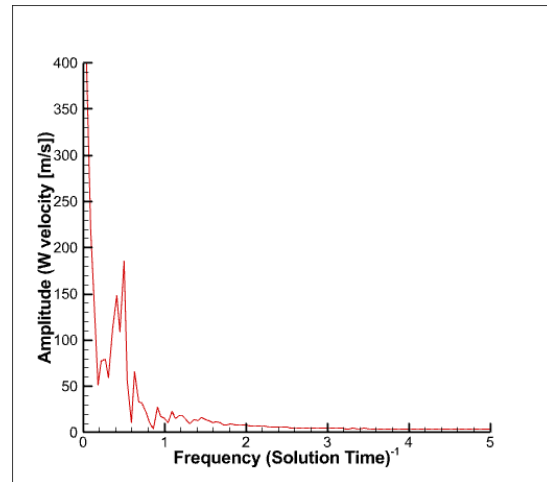
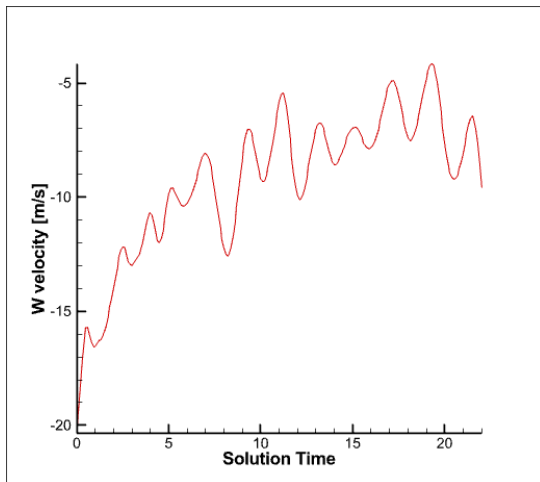
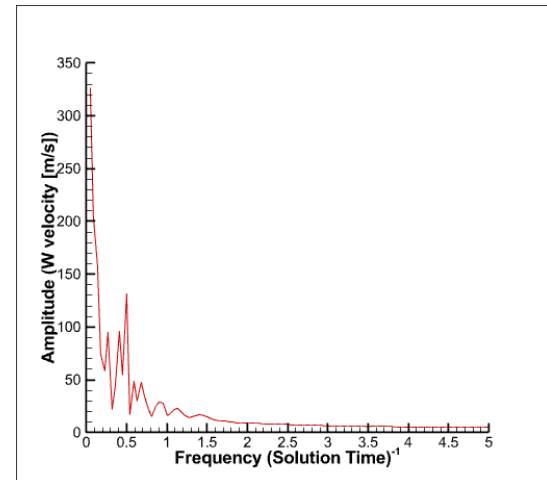
(a)  $z = H$ , time history(b)  $z = H$ , Fourier transform(c)  $z = 2H$ , time history(b)  $z = 2H$ , Fourier transform(e)  $z = 3H$ , time history(b)  $z = 3H$ , Fourier transform

Figure 81: Time histories and corresponding Fourier transform plots of longitudinal velocity at points aft of the cube at  $y = 0.5H$ .

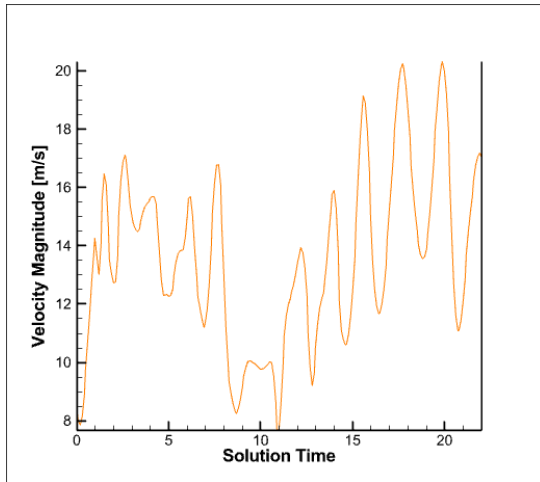
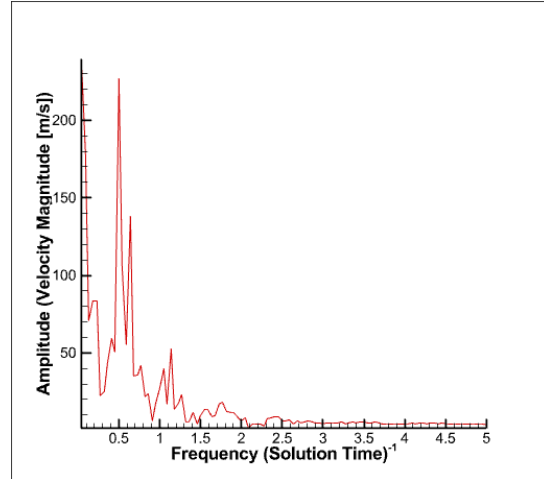
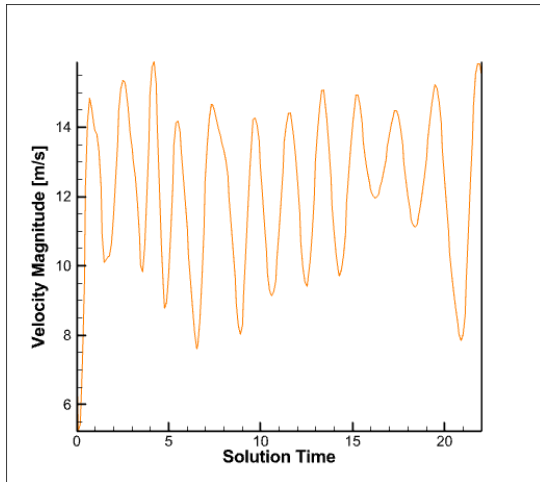
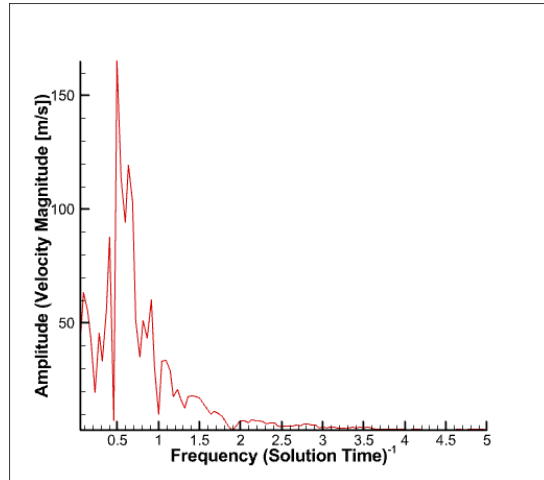
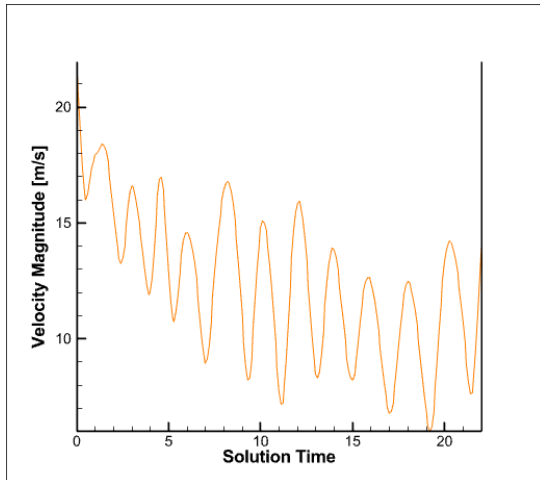
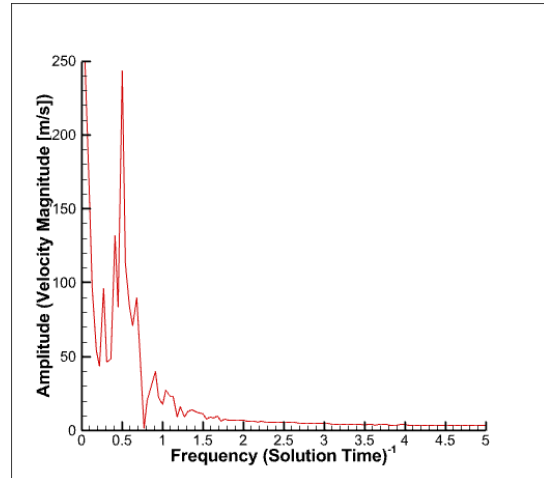
(a)  $z = H$ , time history(b)  $z = H$ , Fourier transform(c)  $z = 2H$ , time history(b)  $z = 2H$ , Fourier transform(e)  $z = 3H$ , time history(b)  $z = 3H$ , Fourier transform

Figure 82: Time histories and corresponding Fourier transform plots of velocity magnitude at points aft of the cube at  $y = 0.5H$ .

### Vortex Model Fitting

Six primary flow features were identified from the CFD simulation simulations and are shown in Figure 83. Given these vortex-based features, we investigate the development of an airwake model that would reconstruct the airwake velocities through a linear superposition of analytical vortex models that capture the velocity profile distributions associated with each of these flow features. Two-dimensional vortices are chosen as starting point for simplicity. When superimposing these vortices to generate 3D airwake profiles, proper coordinate transformations and weightings to control the influence of the individual vortices in the 3D spatial and temporal domain will have to be implemented. The use of analytical vortex models paves the way for parametrization and scaling such that the airwake around different ship geometries may be obtained by modifying inputs to the vortex models, without having to carry out explicit CFD calculations. Model fitting has been performed for the top surface flow separation (Figure 83(a)), separation over the side surfaces Figure 83(b)), backward-facing step separation (Figure 83(d)) and edge vortices (Figure 83(f)). The von Karman streak illustrated in Figure 83(e) is only noted in time-accurate simulations. In time-averaged contours of velocity components, a symmetrical distribution of velocity is noted. Model fitting has been performed for this symmetrical representation as well. These are further described below.

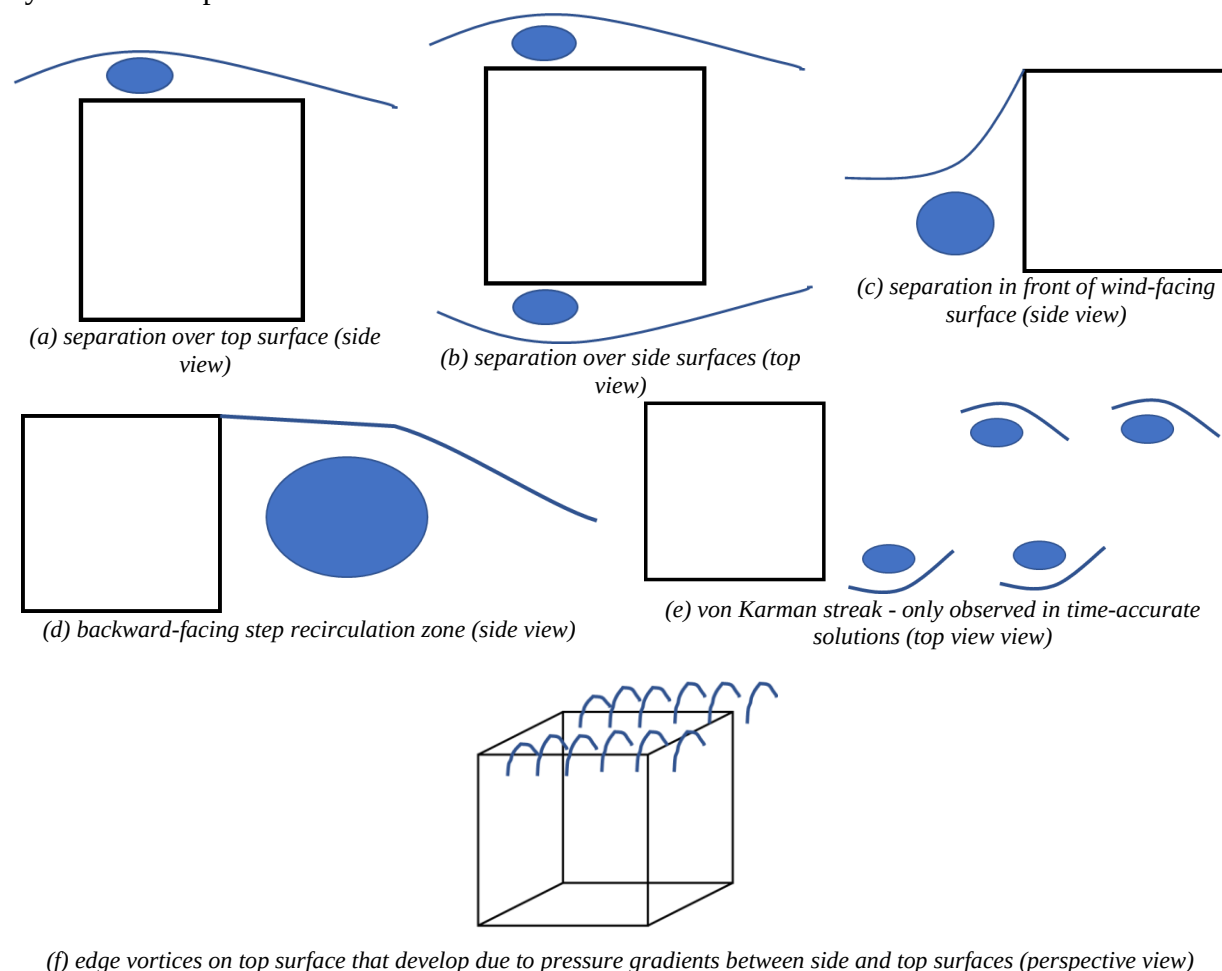


*Figure 83: Primary flow features identified from CGE/VorTran-M simulation of 30 m/s headwind over 10 m cube; freestream direction from left to right.*

### *Top-surface flow separation modeling*

Flow detaches at the top edge of the wind-facing surface and reattaches in the lee of the obstacle (see. Figure 9). Due to the interaction of the incoming freestream above the obstacle with this separation, a recirculation zone is created on the top surface. The flow can also reattach on the top surface, depending on the incoming free stream speed and obstacle dimensions [58]. Recirculation zones are typically associated with a swirling velocity field, making the use of vortex models suitable. For this region, a vortex model based on the one proposed Banks and Meroney [92, 93] is used to capture the longitudinal velocity profile along the midplane of the obstacle.

This validated 2D model was developed to capture the influence of conical vortices on surface pressure. Conical vortices are generated from sharp corners in oblique wind conditions (see Figure 84). These vortices have been studied extensively by the wind engineering community given that they generate high suction regions on roofs of low-rise buildings, which, in turn, can lead to severe structural damage [74, 75, 77]. A depiction of the vortex flow model is shown in Figure 85 [92], where a vortex of radius  $h$  with center at point  $O$  is placed above the surface such that it touches the surface at point  $S$ . To allow for a smooth transition from constant vorticity within the vortex to regions of zero vorticity where potential flow theory applies, a transition region is introduced, with point  $M$  on Figure 85 indicating the start of the potential flow region. Point  $C$  represents the location of maximum velocity in the transition region and is typically set to the midpoint location in the transition region. The vortex is envisioned to be driven like a wheel, spinning due to freestream velocity acting at point  $M$  [92]. Banks and Meroney provide piece-wise continuous expressions for velocity across five distinct regions of the domain, shown in Figure 86. The nondimensional coordinate  $a = \zeta/h$  will be used when referencing distances relative to the vortex center. The core is assumed to have a diameter of  $0.2h$  and represents a region dominated by viscosity, where flow follows a solid body rotation motion. In the region between the core and transition ( $0.2 < a < 1$ ), the velocity profile is based on a power curve fit to the velocity profile obtained from a Navier-Stokes based simulation for a  $65^\circ$  swept delta-wing at a  $10^\circ$  angle of attack at 70% chord and Mach number  $Ma = 0.85$ . This relation is used because of the similarity between conical vortices on the roof of a building and delta-wing vortices observed on delta-wing aircraft. Between the core and roof surface ( $-1 < a < 0.2$ ), pressure changes occur due to centrifugal accelerations of the vortex, such that the resulting governing equation is [92]

$$\frac{dP}{dn} = \frac{\rho U^2}{R_c}, \quad (9)$$

where  $P$  is the pressure,  $n$  is the unit normal to the curvature,  $\rho$  is the fluid density,  $U$  is the fluid speed in the direction of the vortex, and  $R_c$  is the radius of curvature. In the potential flow region, flow merges with freestream and both Bernoulli's equation and Eq. (9) are obeyed. The equations representing the velocity profile from this model are summarized in Table 2. There are three primary inputs to the model: (1) the vortex radius ( $h$ ), (2) the location of point  $M$  ( $R_M$ ), and (3) the velocity at point  $M$  ( $U_M$ ).

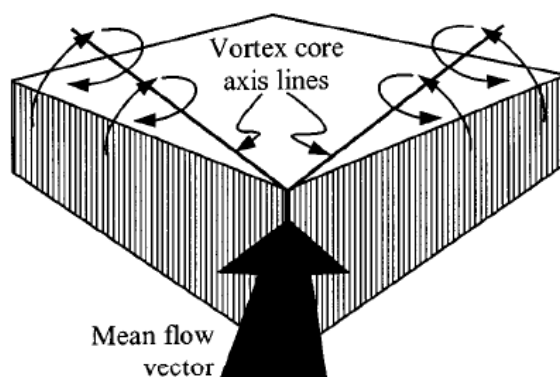


Figure 84: Illustration of conical vortices that form on roof in oblique wind conditions [92].

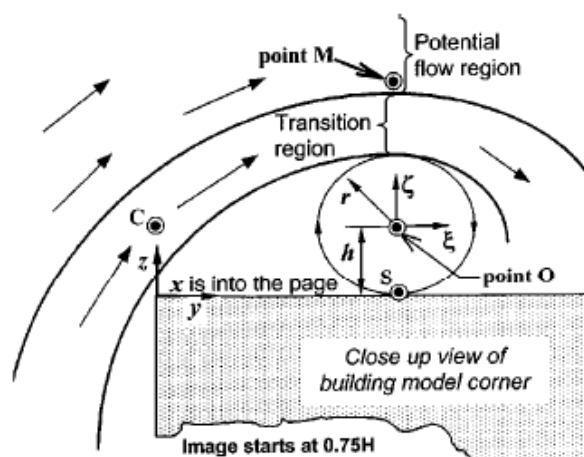


Figure 85: Illustration of the vortex model proposed by Banks and Meroney [93].

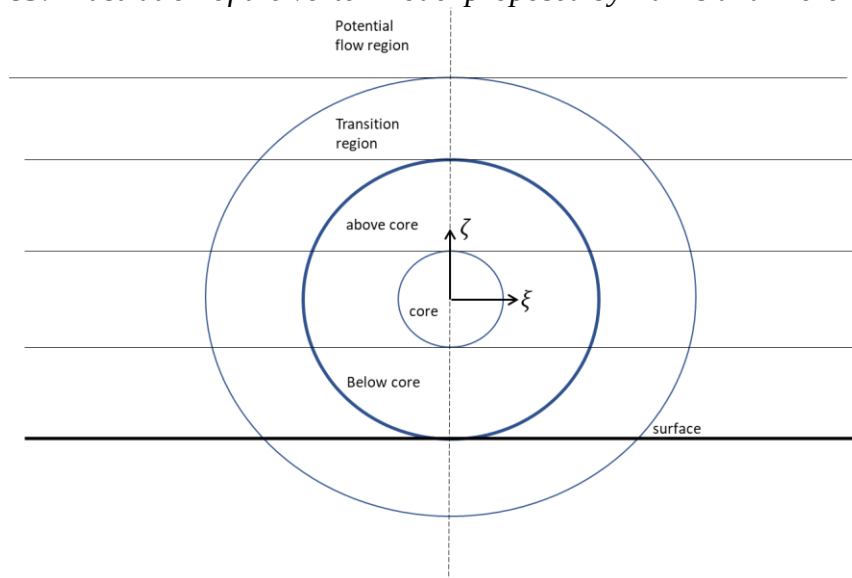


Figure 86: Illustration of the vortex model proposed by Banks and Meroney highlighting the five regions considered.

Table 2: Velocity profile equations for Banks and Meroney vortex model [92];  $a_1 = 0.2$  is used to delimit vortex core size,  $a_2 = 1$  is used to delimit vortex size,  $a_3$  is used to demark start of potential flow region,  $a_{max}$  is used to denoted the point in the transition region where maximum velocity is obtained,  $U_{max}$  is taken to be the velocity at point M,  $U_{a_1} = U_{vortex}(a_1)$ ,  $U_{a_2} = U_{transition}(a_2)$ ,  $U_{a_3} = U_{transition}(a_3)$  and  $n' = n/h$ .

| $R_c/h$     | Range                  | Description                  | Velocity Equation                                                                             |
|-------------|------------------------|------------------------------|-----------------------------------------------------------------------------------------------|
| $-\infty$   | $a = -1$               | Roof surface                 | $U = 0$                                                                                       |
| $a/(1+a)$   | $-a_2 < a$<br>$< -a_1$ | Between roof and vortex core | $U(a) = -U_{vortex}(a)$<br>$= -U_{a_2} \cdot \sqrt{a/-a_2}$                                   |
| $a$         | $ a  < a_1$            | Viscous vortex core          | $U(a) = U_{core}(a) = U_{a_1} \cdot a/a_1$                                                    |
| $a + a^3/2$ | $a_1 < a < a_2$        | Vortex, above core           | $U(a) = U_{vortex}(a)$<br>$= U_{a_2} \cdot \sqrt{a/a_2}$                                      |
| $a + a^3/2$ | $a_2 < a < a_3$        | Transition region            | $U(a) = U_{transition}(a)$<br>$= \frac{U_{max} \cdot 2 \cdot (a/a_{max})}{1 + (a/a_{max})^2}$ |
| $a + a^3/2$ | $a_3 < a$              | Potential flow region        | $U(a) = U_{potflow}(a)$<br>$= U_{a_3} \sqrt{e^{\int_{a_3}^a \frac{-2}{R_c/h} dn'}}$           |

Tecplot's vortex core extraction feature was used to identify the location of the time-averaged vortex for the top separation, highlighted in Figure 87. This location corresponds to  $z = 2.5$  m relative to the global coordinate system. Banks and Meroney's vortex model was used to determine the longitudinal velocity profile along a vertical line that passes through the identified vortex core center. The vortex radius  $h$  was determined by manual tuning to best capture the velocity profile predicted by CGE/VorTran-M. Following the approach laid out in Ref. [93], point M can be selected to be at a location where the correlation coefficient between the velocity at the given location and the surface pressure coefficient at point S is close to 1. Figure 88 shows a plot of the correlation coefficient as a function of height from the top surface, and suggests that any point above  $y = 3.2$  m can be chosen. The location of point M was determined by manual tuning to best capture the velocity profile. The final model parameters and corresponding velocity profile are shown in Table 3 and Figure 89(a), respectively. Velocity predictions from the model closely follow those from the CGE/VorTran-M simulation, with maximum offsets of approximately 4 m/s, noted in the regions above the core center. Below the core center, model predictions deviate from CFD, which, interestingly, seem to follow a viscous boundary layer (BL) profile despite being inviscid. This appears to be due to the suction pressures associated with the separated flow over the top surface, and the deviation from CGE predictions below the vortex center is expected given the symmetry in the vortex model. The velocity profile below the vortex center can be adequately captured using a quadratic fit of the form

$$U(y) = c_1(-y^2 - y), \quad (10)$$

as shown in Figure 89(b). The constant  $c_1 = 10$  was used to generate the figure.

Table 3: Vortex model parameter values; distances specified relative to top surface.

| Parameter   | Value |
|-------------|-------|
| $h$ (m)     | 1.5   |
| $R_M$ (m)   | 4.9   |
| $U_M$ (m/s) | -32.6 |

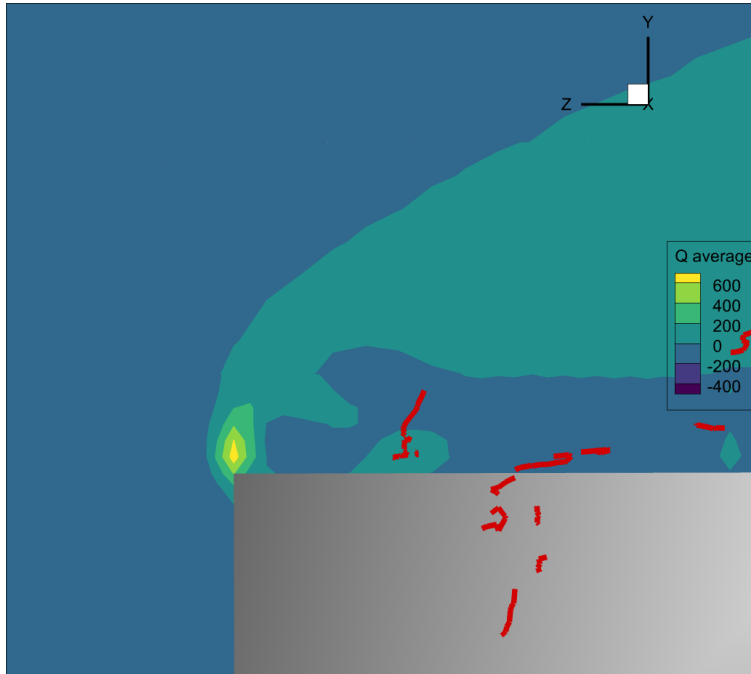


Figure 87: Contour of the time-averaged  $Q$  criterion across a midplane through the center of the obstacle; red lines indicated vortex core centers determined by Tecplot.

Steps were taken to expand the vortex model predictions from the one-dimensional vertical line across the vortex center to the full 2D region over the top surface of the cube. The equations in Table 2 are expressed in radial coordinates. However, the velocity profiles obtained using the model for radial lines at nonzero angular displacements from the vertical axis did not adequately capture results from CFD, as illustrated in Figure 90 where velocity profiles for radial lines at  $\theta = 30^\circ$  and  $\theta = 60^\circ$  (with  $\theta$  measured counterclockwise from the  $\zeta$  axis in Figure 86) are plotted.

An alternate method to obtain the 2D velocity profile was thus devised. Vortex model fitting was performed at five different span locations along the top surface of the cube;  $z = \{5, 2.5, 0, -2.5, 5\}$  m, resulting in five corresponding values for the model inputs  $h$ ,  $R_M$  and  $U_M$ , summarized in Table 4. The velocity profile prediction for  $z = -2.5$  m is shown in Figure 89 whereas the other four locations are shown in Figure 91. Except for the region within a height of 1 m from the top surface at  $z = 5$  m (Figure 91(a)), the manually tuned vortex models at the various span locations show reasonable agreement with CFD predictions. The  $z = 5$  m location corresponds to the leading-edge corner of the top vortex. The discrepancy within the 1 m region from the surface is acceptable given that in practice, the aircraft will be very close to landing at that position, and it's unlikely that velocities at this height will have any influence on flying qualities. Furthermore, if this height is nondimensionalized by the cube height  $H = 10$  m, we can generalize that model predictions are likely not valid in the  $0.1H$  region above the obstacle.

near the leading edge. Maximum offset at the  $z = 5\text{ m}$  span location is noted in the potential flow region, where deviations of approximately 20% from the  $-30\text{ m/s}$  predicted by CGE/VorTran-M are evident. These deviations are also considered to be acceptable in the current study.

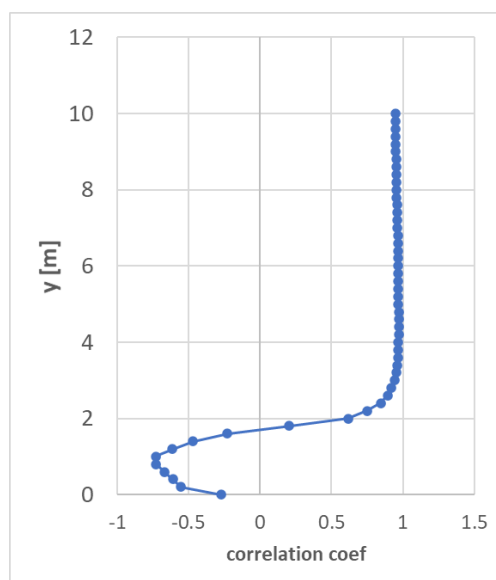


Figure 88: Correlation coefficient between longitudinal velocity at various heights above the top surface and surface pressure.

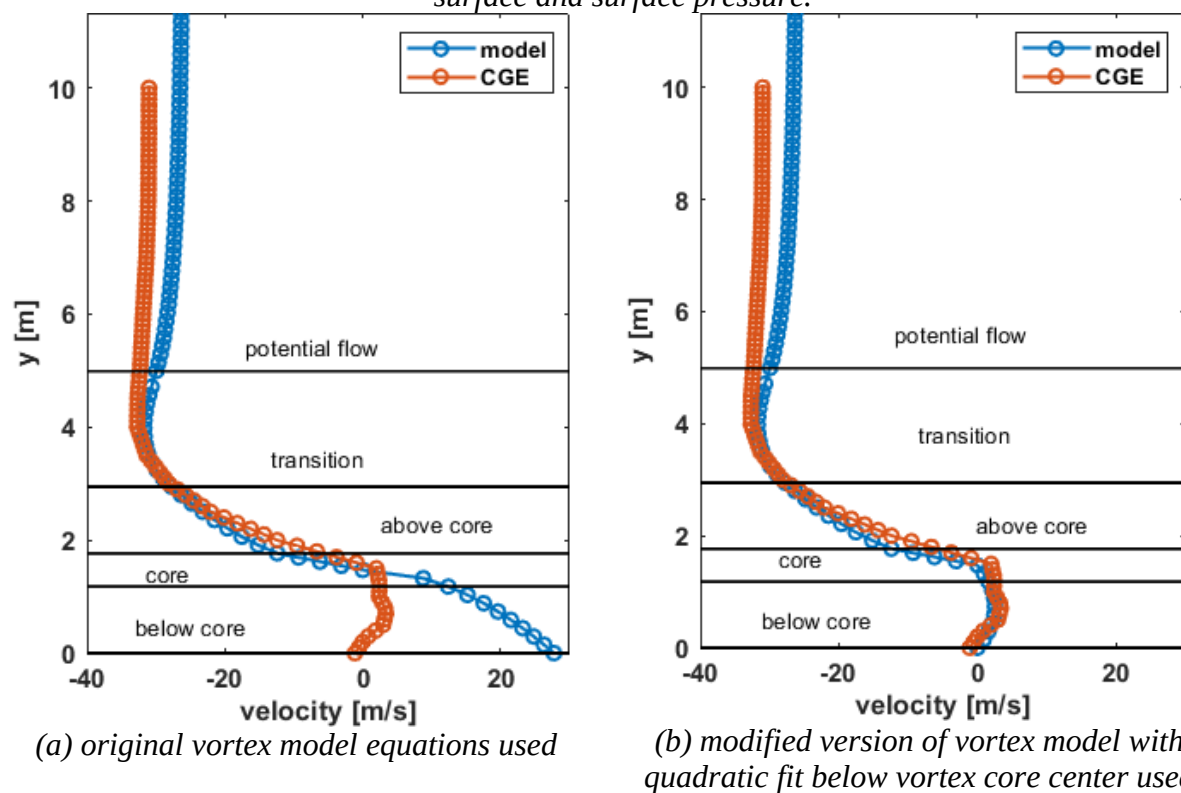


Figure 89: Velocity profile along vertical line at  $z = 2.5\text{ m}$ .

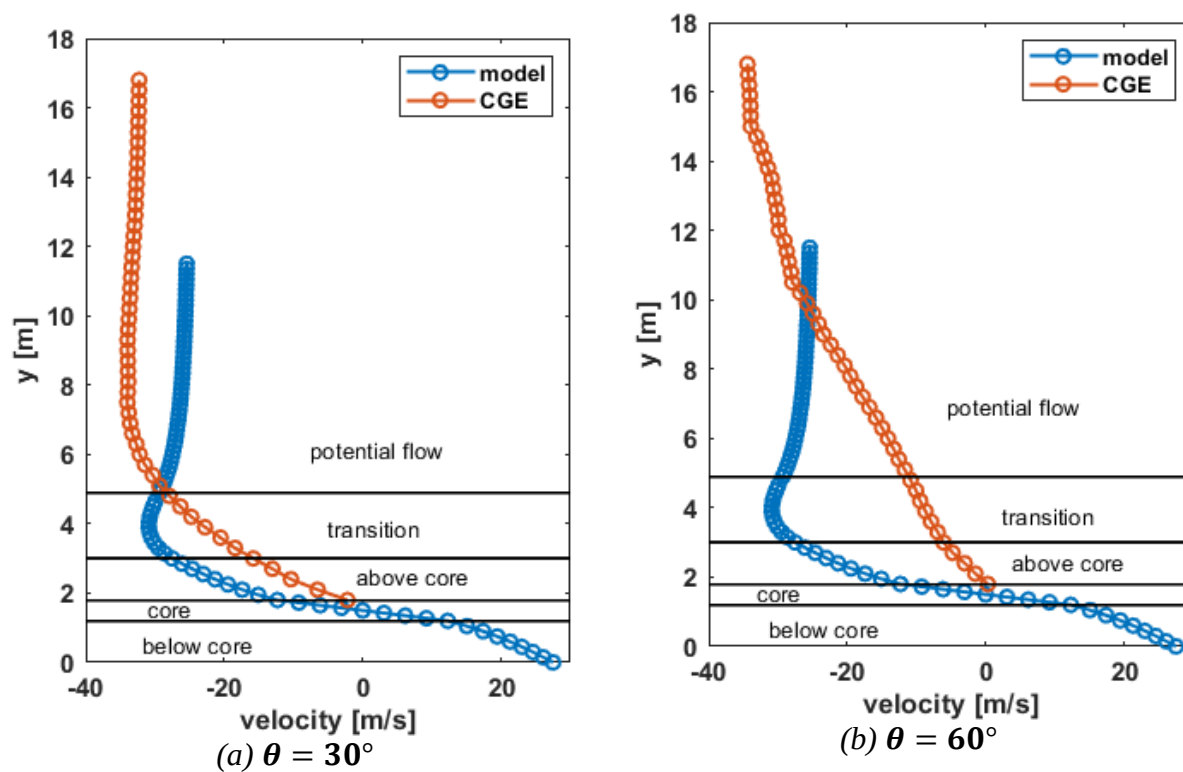


Figure 90: Velocity profile along non-vertical radial line with vortex center at  $z = 2.5$  m.

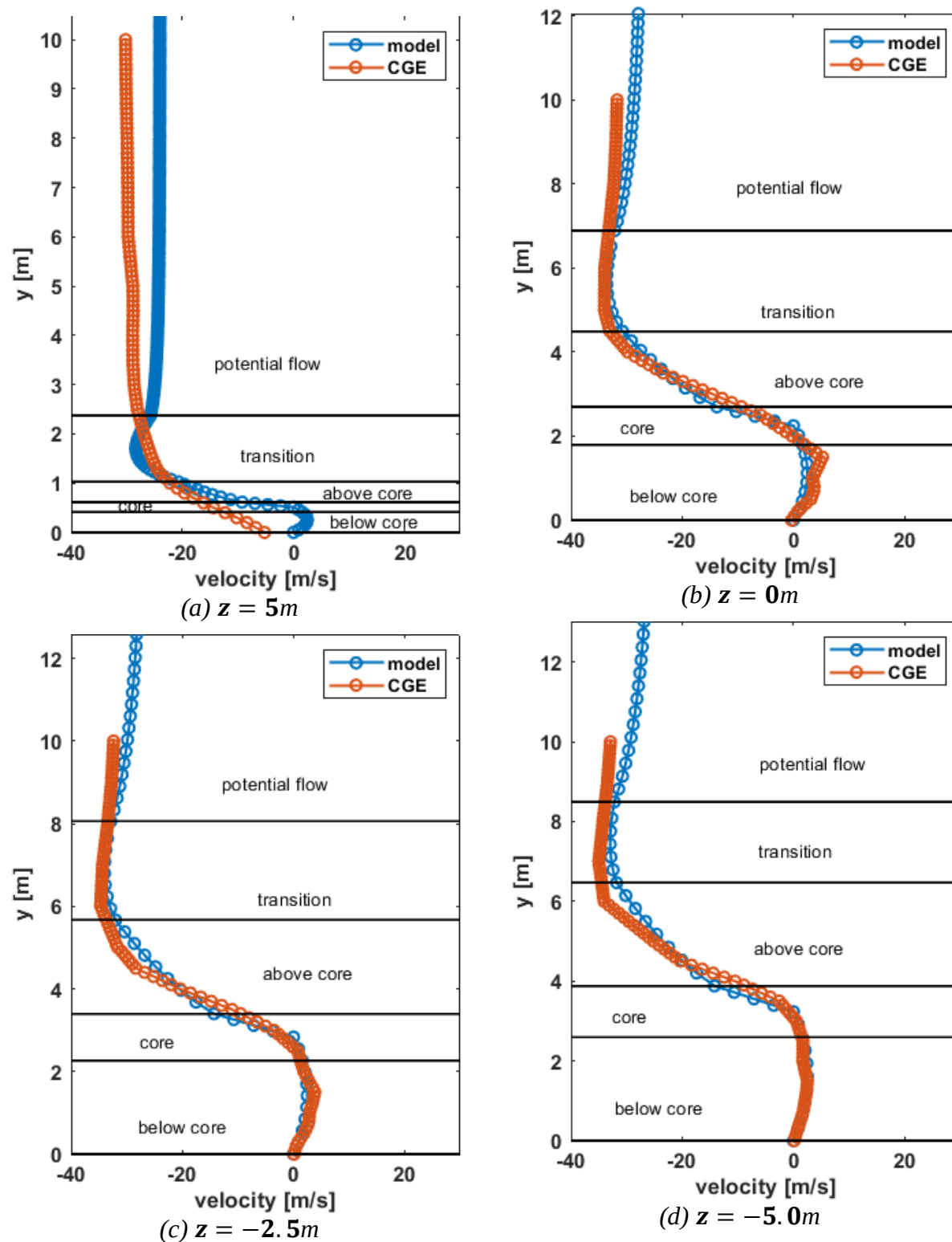


Figure 91: Velocity profile at various longitudinal span locations for a 30 m/s headwind.

Table 4: Vortex model input parameters for various longitudinal span locations.

| $z$ [m] | $h$ [m] | $U_M$ [m/s] | $R_M$ [m] |
|---------|---------|-------------|-----------|
| 5       | 0.5     | -28.0       | 2.4       |
| 2.5     | 1.5     | -32.6       | 4.9       |
| 0       | 2.25    | -33.2       | 7.0       |
| -2.5    | 2.8     | -33.6       | 8.0       |
| -5      | 3.25    | -33.2       | 8.5       |

Quadratic polynomials were fitted onto the values in Table 4 to obtain spanwise functions  $h(z)$ ,  $R_M(z)$  and  $U_M(z)$ . The resulting equations are valid over the top surface of the cube ( $-5 \leq z \leq 5$  m). Velocity profile predictions at  $z = \{3.5, -3.5\}$  m are plotted in Figure 92. These locations were not considered during the fitting but still show good agreement with predictions from CFD.

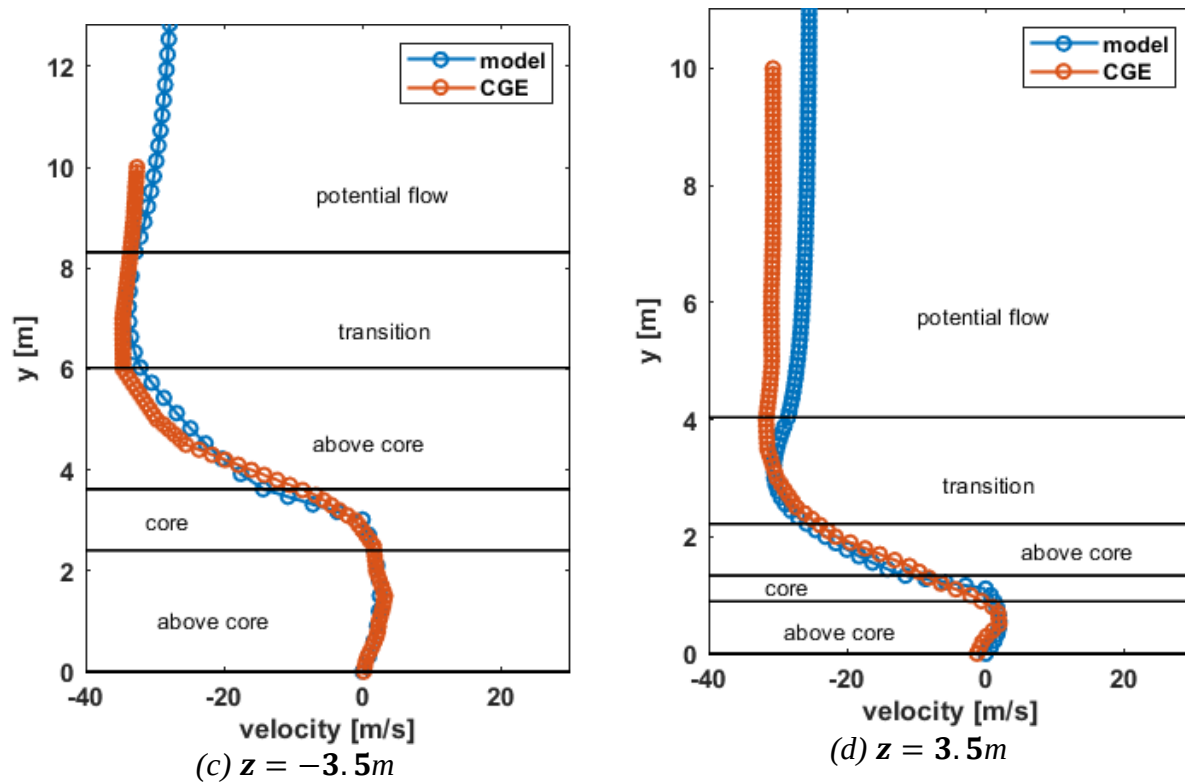


Figure 92: Velocity profile at two longitudinal span locations not used in determining  $h(z)$ ,  $R_M(z)$  and  $U_M(z)$ .

The 2D vortex model used to capture the velocity profile along the top surface of the cube is effectively an extension of the Banks and Merony model [92, 93] that assumes an infinite number of overlapping vortices of differing sizes exist along the top surface. This representation adequately captures the velocity profile for the current case study where flow reattaches in the lee of the obstacle. For flow regimes where reattachment occurs on the top surface directly, modifications to this model may be required.

A preliminary model scaling investigation was also carried out using CGE/VorTran-M simulations of a 40 m/s headwind over the cube. It was found that the velocity profile along the top surface of the cube along the midplane could be captured by the model by simply scaling  $U_M(z)$  for the 30 m/s case accordingly, see Figure 93. As noted previously, this scaling relationship is expected given the inviscid solver being used but also is a good approximation when viscous flow effects are included given the magnitude of the operating Reynolds number.

The application limits of the vortex model presented above still need to be investigated but we anticipate it to be applicable to rectangular prisms of varying dimensions. Model input parameters  $h$  and  $R_M$  can be potentially parametrized for scaling using either the height, length, or width of the obstacle, and the rotational velocity  $U_M$  using freestream velocity.

#### *Backward-facing step flow separation modeling*

Aft of the cube, shear layers are shed from the top and side edges of the backward-facing surface, see Figure 9. The shear layers shed from the top edge are considered here (Figure 83(d)), and vortex model fitting is currently being performed using time-averaged longitudinal velocity profiles aft of the obstacle. In practice, vortex shedding in this region is dynamic; the location where the shear layer impinges on the wall fluctuates, shifting the reattachment zone upstream and downstream in a low frequency motion [45, 46, 50, 51], with the recirculation bubble shrinking and expanding accordingly. We plan to account for these dynamics once vortex models that adequately capture time-averaged velocity profiles have been obtained.

Time-averaged streamlines across the midplane of the cube are shown in Figure 94, and a single leeside primary vortex structure is evident. The center of the vortex was found to be at  $(z, y) = (-18.8, 10.9)$  m using Tecplot's vortex core extraction feature.

Vortex model fitting was performed to capture the longitudinal velocity profile along a vertical line through the identified vortex center. Peak longitudinal velocity of 14.7 m/s was noted at  $y = 2.9$  m along that line. A Rankine vortex model was first used with a velocity profile given by

$$U(y) = \begin{cases} \Omega r & r \leq r_c \\ \frac{\Omega r_c^2}{r} & r > r_c \end{cases}, \quad (11)$$

where  $\Omega$  is the angular rotation speed of the vortex, determined using the peak longitudinal velocity and vortex core radius  $r_c$  (set here to 8 m, which is the distance from the vortex center to the location where peak longitudinal velocity is noted). As evident in Figure 95, the linear velocity variation associated with the Rankine vortex core failed to adequately represent the velocity profile predicted by CGE/VorTran-M.

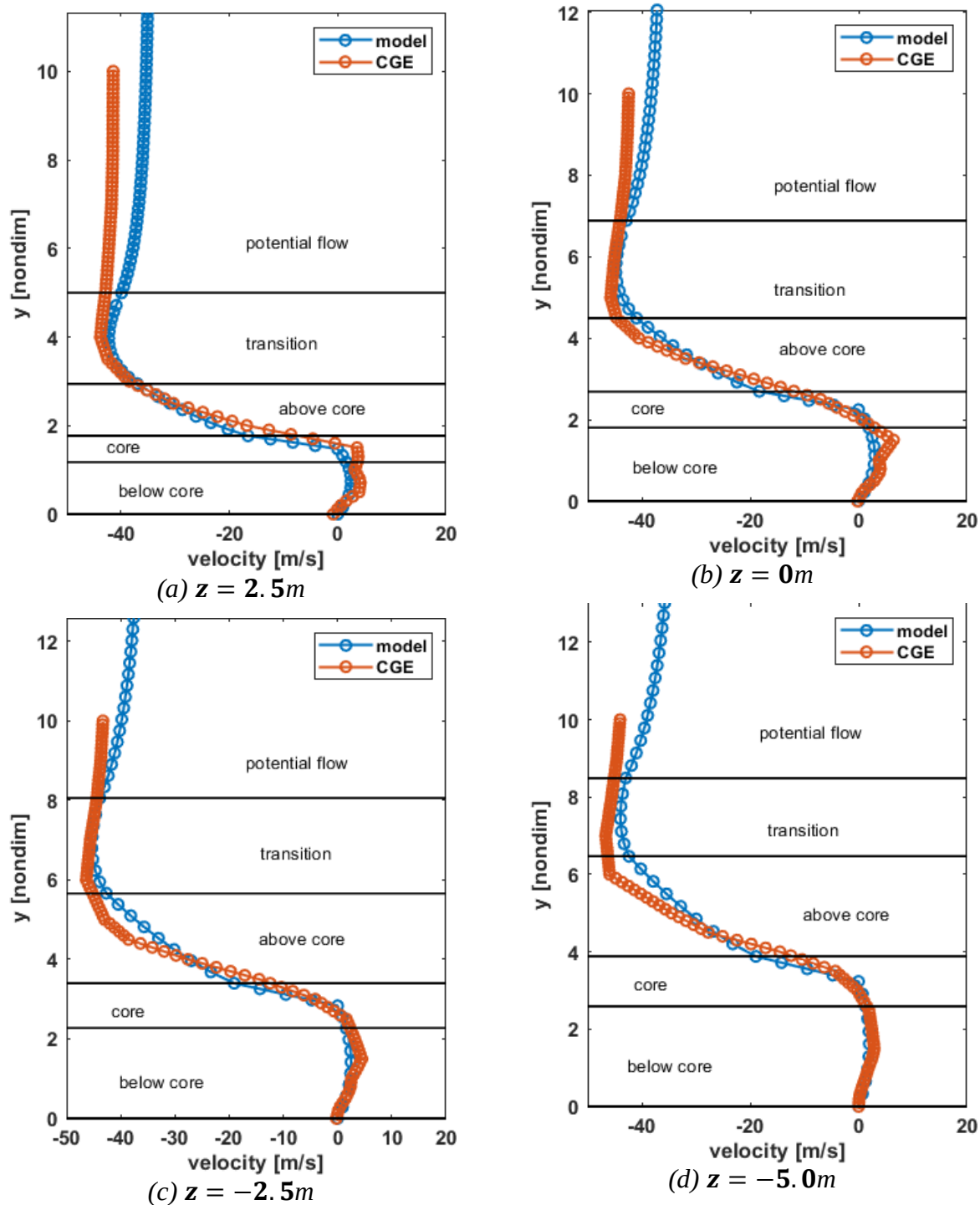


Figure 93: Velocity profile at various longitudinal span locations for a 40m/s headwind.

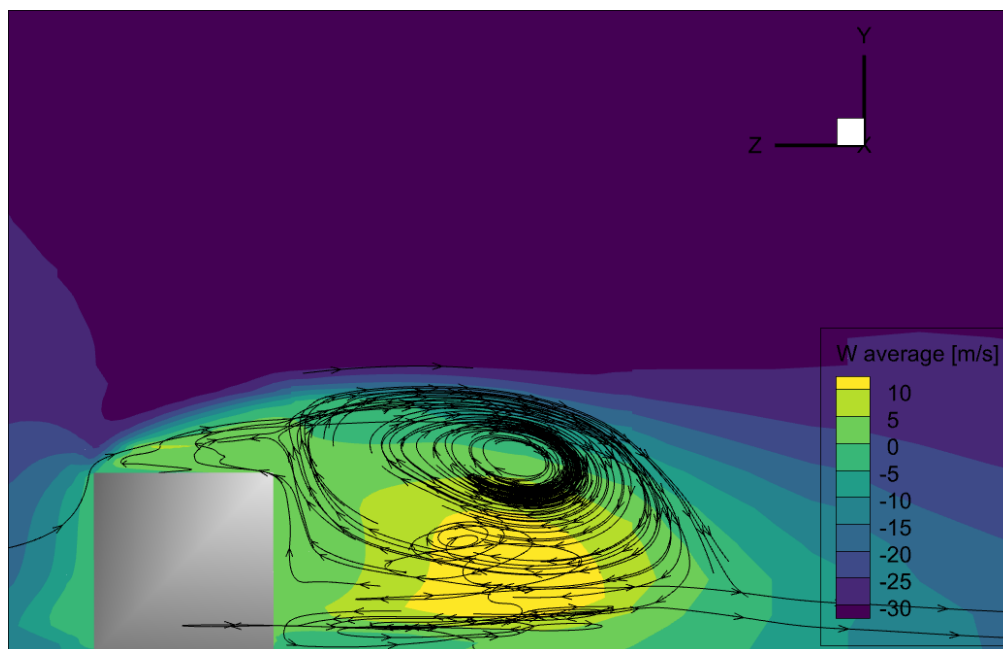


Figure 94: Contour of time-averaged longitudinal velocity profile along midplane with streamlines indicating presence of aft of the obstacle.

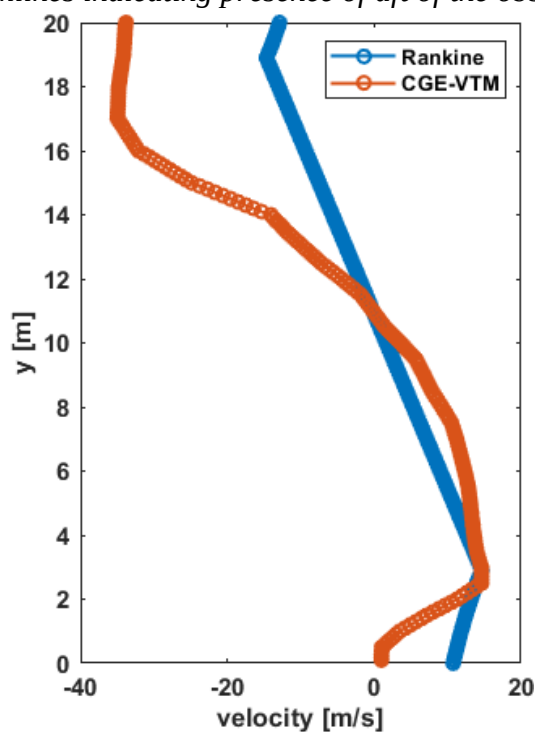


Figure 95: Rankine vortex-based velocity profile along vertical line through vortex center.

A modified version of the Banks and Meroney model [92, 93] introduced in the previous section was found to produce a better agreement. The equations in Table 2 were still applied but in this case, the vortex was shifted upwards such that a transition region existed below the vortex center as well, as depicted in Figure 96. With this shift, the vortex model can be thought of as

wheel being spun at two points: point  $M_{upper}$  at a velocity  $U_{M_{upper}}$  and point  $M_{lower}$  at a velocity  $U_{M_{lower}}$ . Inputs to the model thus consisted of

1. the vortex radius ( $h$ ),
2. the location of points  $M_{upper}$  and  $M_{lower}$  (assumed to be equidistant from the vortex center)
3. the velocity at point  $M_{upper}$  ( $U_{M_{upper}}$ ) and
4. the velocity at point  $M_{lower}$  ( $U_{M_{lower}}$ ).

To maintain the piece-wise continuity of the vortex model velocity profile predictions, the maximum velocity  $U_{max}$  (see Table 2) used in the core, below core, and lower transition regions was set equal to  $U_{M_{lower}}$ . In the upper transition and potential flow regions,  $U_{max}$  was set equal to  $U_{M_{upper}}$ . In the above core region, linear interpolation between  $U_{M_{lower}}$  and  $U_{M_{upper}}$  was used to determine  $U_{max}$ . Two additional modifications were made to the vortex model introduced in the previous section:

1. the location where the maximum velocity was obtained in the transition region ( $a_{max}$ ) was changed from the midpoint location to the end point of the transition region; and
2. the size of the vortex core was increased from  $0.2h$  to  $0.4h$ .

Velocity predictions in the region below the lower transition were obtained using linear interpolation between the velocity at point  $M_{lower}$  and 0 m/s at a point  $y_0$ , below which the velocity was assumed to be zero. Vortex parameters and resulting velocity profiles are shown in Table 5 and Figure 97, respectively, where the model shows good agreement with the velocity profile predicted by CGE/VorTran-M, with a maximum deviation of approximately 7 m/s noted at  $y = 14$  m.

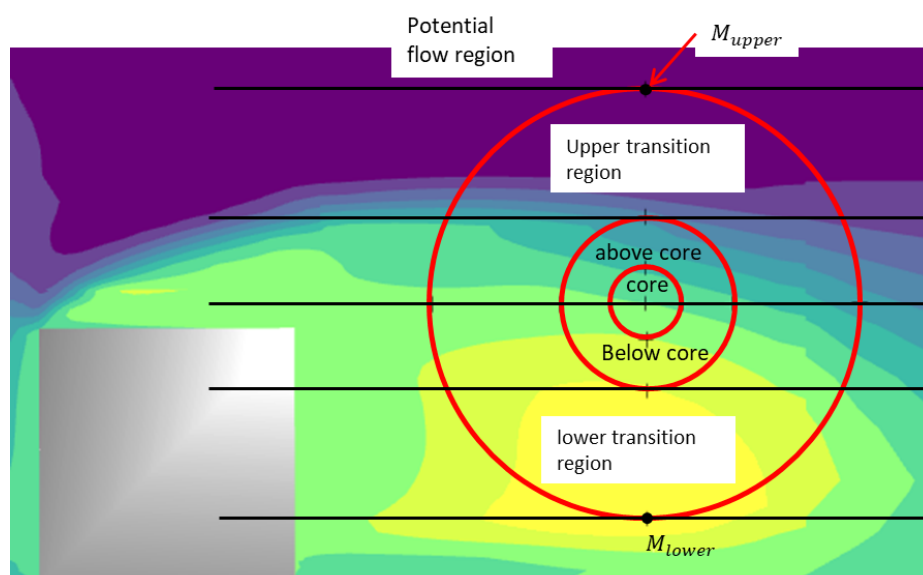


Figure 96: Illustration of modified Banks and Meroney vortex model shifted upwards.

Table 5: Modified vortex model parameter values; distances specified relative to global CGE coordinate system.

| Parameter             | Value              |
|-----------------------|--------------------|
| $h$ (m)               | 3.4                |
| $R_M$ (m)             | 8.4                |
| $U_{M_{lower}}$ (m/s) | -14.7 <sup>1</sup> |
| $U_{M_{upper}}$ (m/s) | -34.2              |
| $y_0$ (m)             | 0.5                |

<sup>1</sup> Value was negated to obtain correct profile.

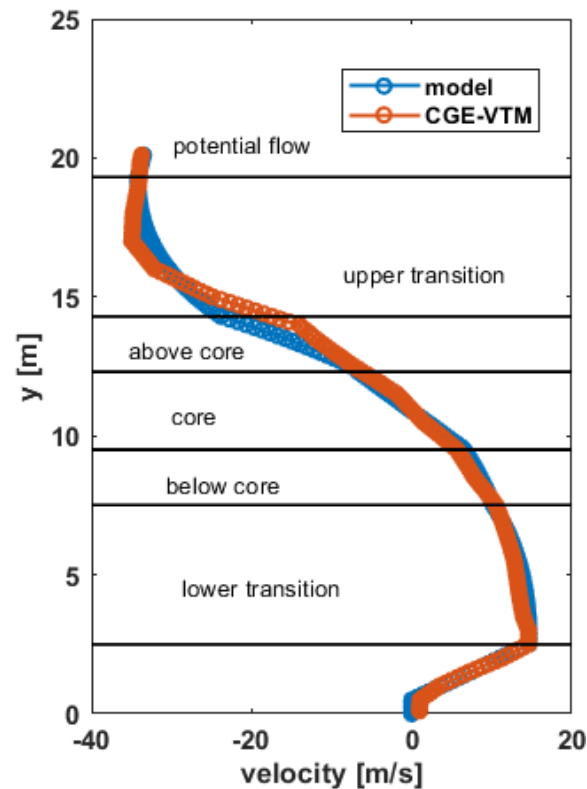


Figure 97: Longitudinal velocity profile along a vertical line through vortex center in the lee of the cube.

Next, velocity profiles along radial lines at  $\theta = \{30^\circ, 45^\circ, 135^\circ, 150^\circ\}^T$  were considered, where  $\theta$  is measured counterclockwise from an axis  $z'$ , which has its origin at the vortex center and is parallel to the global CGE  $z$  axis shown in the contour plots above. We found that a circular vortex model, as depicted in Figure 96, failed to fully capture the velocity predictions from CGE/VorTran-M (see Figure 98). Reasonable agreement is noted for  $\theta = 135^\circ$  and  $\theta = 150^\circ$  in Figure 98, but large discrepancies are noted for the  $\theta = 30^\circ$  and  $\theta = 45^\circ$  cases. These discrepancies are attributed to the influence of the top surface separation considered in the previous section, and the corresponding vortex model parameters are summarized in Table 6. Note that longitudinal velocity values were used as rotational velocities  $U_{M_{lower}}$  and  $U_{M_{upper}}$ . Using tangential velocities resulted in significant offsets in model predictions.

Table 6: Modified vortex model parameter inputs for the radial lines considered.

| Parameter             | $\theta = 30^\circ$ | $\theta = 45^\circ$ | $\theta = 135^\circ$ | $\theta = 150^\circ$ |
|-----------------------|---------------------|---------------------|----------------------|----------------------|
| $h$ (m)               | 3.4                 | 3.4                 | 3.4                  | 3.4                  |
| $R_M$ (m)             | 8.4                 | 8.4                 | 8.4                  | 8.4                  |
| $U_{M_{lower}}$ (m/s) | -2.5 <sup>1</sup>   | -8.0 <sup>1</sup>   | -8.9 <sup>1</sup>    | -7.0 <sup>1</sup>    |
| $U_{M_{upper}}$ (m/s) | -20.9               | -34.7               | -31.8                | -28.2                |
| $y_0$ (m)             | 0.5                 | 0.5                 | 0.5                  | 0.5                  |

<sup>1</sup> Values negated to obtain profile.

To better match CGE/VorTran-M velocity predictions, we then investigated the use of an ellipse-shaped vortex following the streamline pattern shown in Figure 94. The dimensions in Table 5 were used to define the semi-minor radii, and the semi-major radii were obtained by applying a scaling factor to the semi-minor radius values. A scaling factor value of 3.3 was used in this work based on manual tuning. The ellipse-based vortex was also assumed to have a built-in slant angle  $\gamma$ , measured counterclockwise relative to the  $z'$  axis. Using position coordinates of several points along the streamlines shown in Figure 94, the slant angle was determined to be approximately  $16^\circ$ . The resulting vortex shape is illustrated Figure 99.

Velocities obtained using the ellipse-based vortex model are shown in Figure 100. Results show good agreement with predictions from CGE/VorTran-M, with the largest discrepancies again noted for the  $\theta = 30^\circ$  and  $\theta = 45^\circ$  cases. These discrepancies, however, are significantly less pronounced than those noted in Figure 98.

It is important to clarify here that this “elliptical vortex” is a mathematical modeling construct that was found to better represent the CFD source data in the recirculation region. Physically, such a construct does not exist but can be viewed as an approximation of a vortical structure in which the vorticity is distributed over a spatial dimension (e.g., a vortex line or sheet). In this physical case, the actual velocity field associated with this vortical structure may not be precisely elliptical in shape, depending on the distribution of vorticity. This distinction is not deemed critical to the formulation of the canonical airwake models at the moment, but will be re-examined as necessary.

Note further that the velocities  $U_{M_{lower}}$  and  $U_{M_{upper}}$  were initially probed manually from Tecplot. To facilitate the vortex model fitting process, the value of  $U_{M_{lower}}$  at a given span location aft of the cube was set to the maximum longitudinal velocity along the vertical line at that location. Similarly,  $U_{M_{upper}}$  was set to the minimum longitudinal velocity. These definitions are based on the CGE problem setup, where freestream is in the negative  $z$  direction. A Tecplot macro script was used to query the velocity profiles at various span location aft of the cube, which were then post-processed to obtain quadratic functions  $U_{M_{lower}}(z)$  and  $U_{M_{upper}}(z)$ . These functions were used to generate the profiles in Figure 100.

Although this ellipse-based vortex fit showed good agreement for radial lines at nonzero  $\theta$  angles, it failed to capture the velocities along  $z$  span locations that did not traverse all three vortex regions shown in Figure 99, producing good predictions primarily in the  $-23 \leq z \leq -15$  m region. Further tuning of the model was performed, and better predictions were obtained using a semi-minor radius to semi-major radius of 10 and a slant angle of approximately  $8^\circ$ . The resulting vortex shape is illustrated Figure 101. Note that the vortex core region only extends to

$z \approx -32$  m downstream of the obstacle. Velocity predictions were obtained using a coordinate system that has its origin at the center of the ellipse, as illustrated in Figure 102. At any given span location, it was assumed that the radial coordinate from the center of the ellipse to any point below the intersection point between the  $\xi$  axis and the vertical line at the span location was negative. Similarly, any radial coordinate above the intersection point was positive. This sign convention was adopted for consistency with the Banks and Meroney model equations shown in Table 2.

Velocities obtained using the updated ellipse-based vortex model are shown in Figure 103 and Figure 104. Results show good agreement with prediction from CGE/VorTran-M up to  $z = -26$  m. Apart from velocity predictions at  $z = -18$  m, a kink in the predictions is noted in the core region at all span locations in Figure 103. This is due to the sudden switch in radial coordinate sign from negative to positive at those locations. The  $z = -18$  m location is close to the vortex center ( $z = -18.8$  m) and thus the magnitude of the radial coordinate at the intersection of the  $\xi$  axis and the vertical line at  $z = -18$  m is relatively small (see Figure 102). From Figure 103 and Figure 104, predictions from the model deteriorate downstream from  $z = -26$  m onwards. Sharp changes in velocities are also noted beyond  $z = -30$  m, where the vortex core region does not exist (see Figure 101).

To better match CGE/VorTran-M predictions beyond  $z = -26$  m, a secondary vortex model was adopted in this downstream region. This model follows a similar structure to that presented for the top surface separation above and assumes that an infinite number of overlapping vortices of differing sizes exist in the downstream region. Linear functions for the vortex center location  $y_0(z)$ , vortex radius  $h(z)$ , vortex core fraction  $c(z)$ , and location of  $M_{upper}(z)$  were derived by manually tuning vortex model parameters at various  $z$  locations aft of  $z = -26$  m:

$$y_0(\bar{z}) = -0.1514\bar{z} + 11.481, \quad (12)$$

$$h(\bar{z}) = 0.2045\bar{z} + 3.7321, \quad (13)$$

$$c(\bar{z}) = 0.0124\bar{z} + 0.4, \quad (14)$$

$$M_{upper}(\bar{z}) = 0.0661\bar{z} + 8.604, \quad (15)$$

where  $\bar{z} = z_1 - z$ , with  $z_1 = -26$  m. From Figure 104, the inviscid nature of the simulation is apparent from the nonzero velocities near the ground at span location downstream of  $z = -26$  m. In practice, velocity near the ground would be close to zero. To better match the *inviscid* predictions, a linear function for the velocity near the ground was derived:

$$V_{y_0}(z) = -0.7252\bar{z} - 1.3413. \quad (16)$$

Velocity predictions in the region below the lower transition (see Figure 96) at a given span location were then obtained using linear interpolation between the velocity  $U_{M_{lower}}(z)$  at point  $M_{lower}$  and  $V_{y_0}(z)$  from Eq. (16) at point  $y_0$ , below which the velocity was assumed to be  $V_{y_0}(z)$ .

The combined use of the ellipse-based vortex model up to  $z = -26$  m and the infinite vortex model beyond that location allowed for the velocity profile aft of the obstacle to be adequately captured as shown in Figure 105 and Figure 106. Note that the velocities in the core region in Figure 105 were obtained using an explicit linear interpolation between the velocity in the above core and below core regions to avoid the kinks seen in Figure 103.

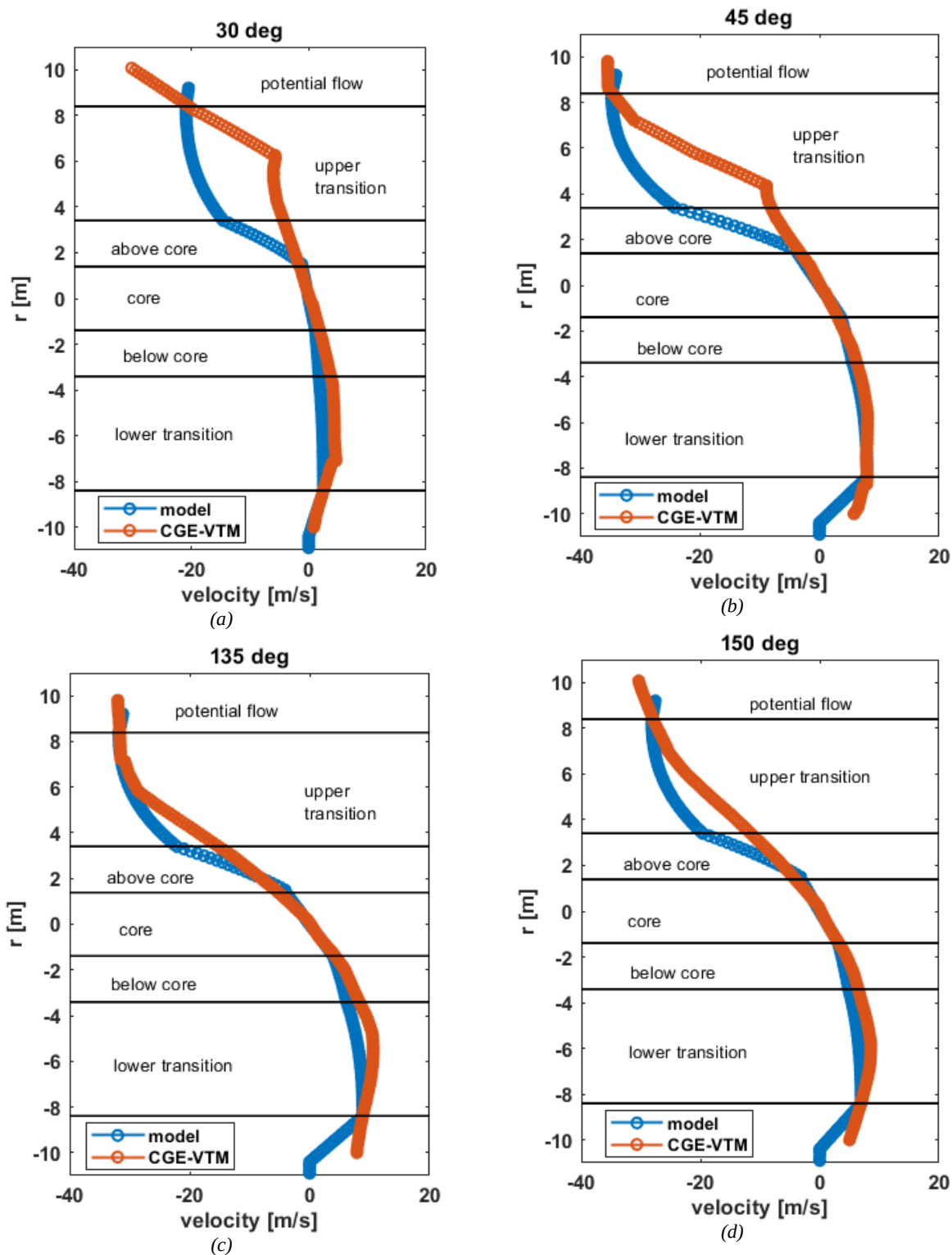


Figure 98: Velocity profiles along radial lines at nonzero angular displacements in the lee of the obstacle; vortex assumed to be circular.

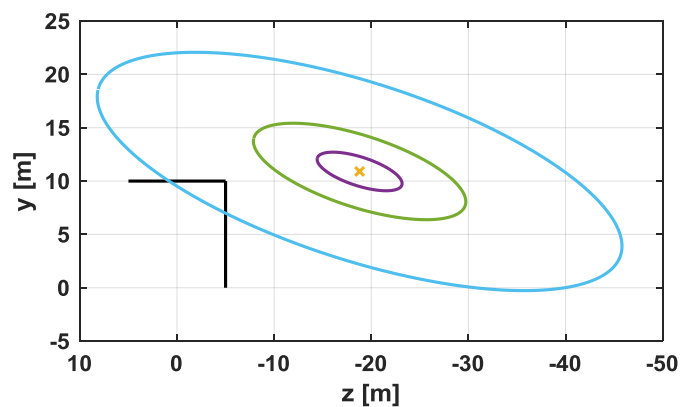


*Figure 99: Illustration of the ellipse-based vortex; black line outlines the obstacle; orange cross indicates vortex center; purple line delimits vortex core; green line delimits vortex size; and blue line indicates the end of the transition region.*

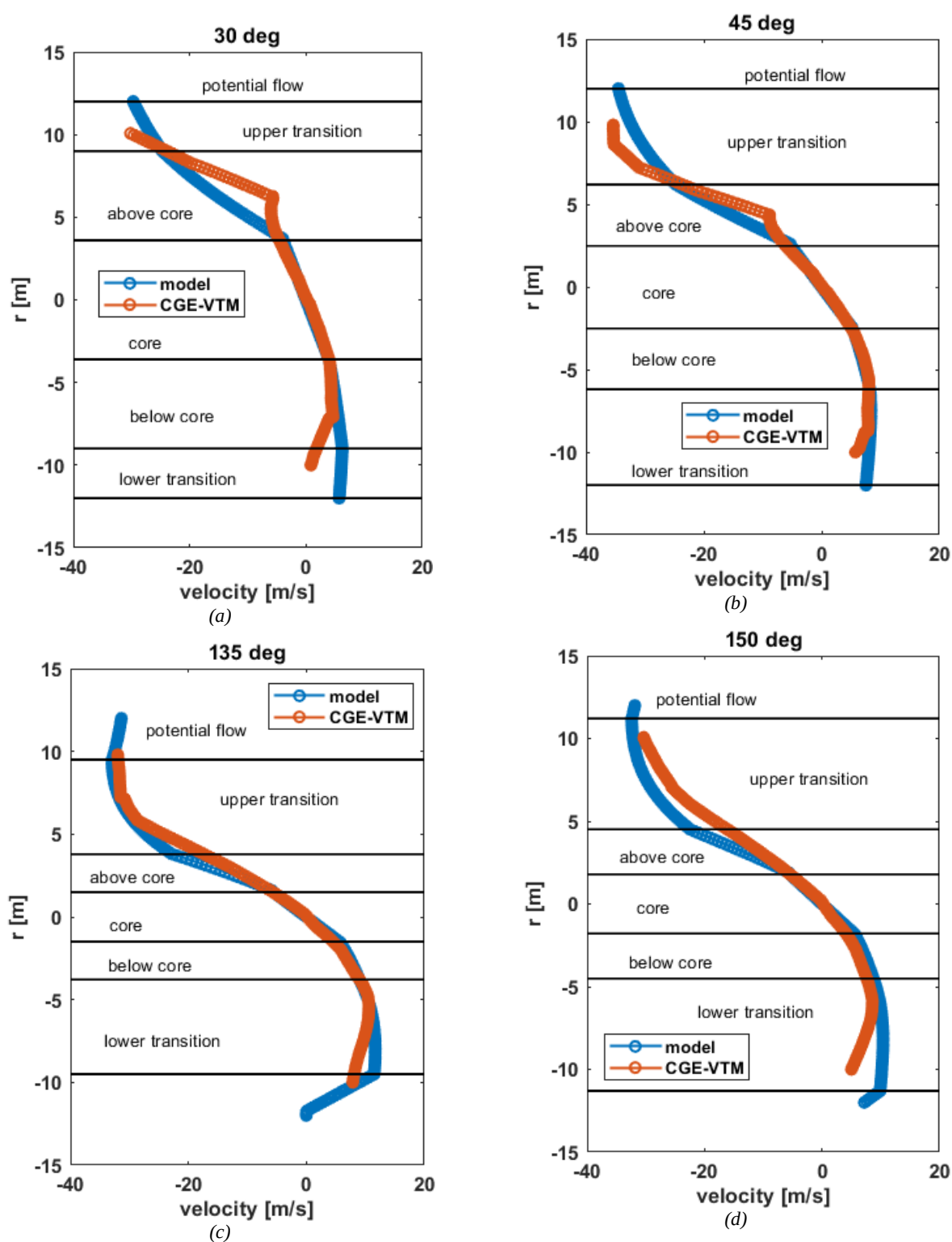


Figure 100: Velocity profiles along radial lines at nonzero angular displacements in the lee of the obstacle; vortex assumed to be a slanted ellipse.

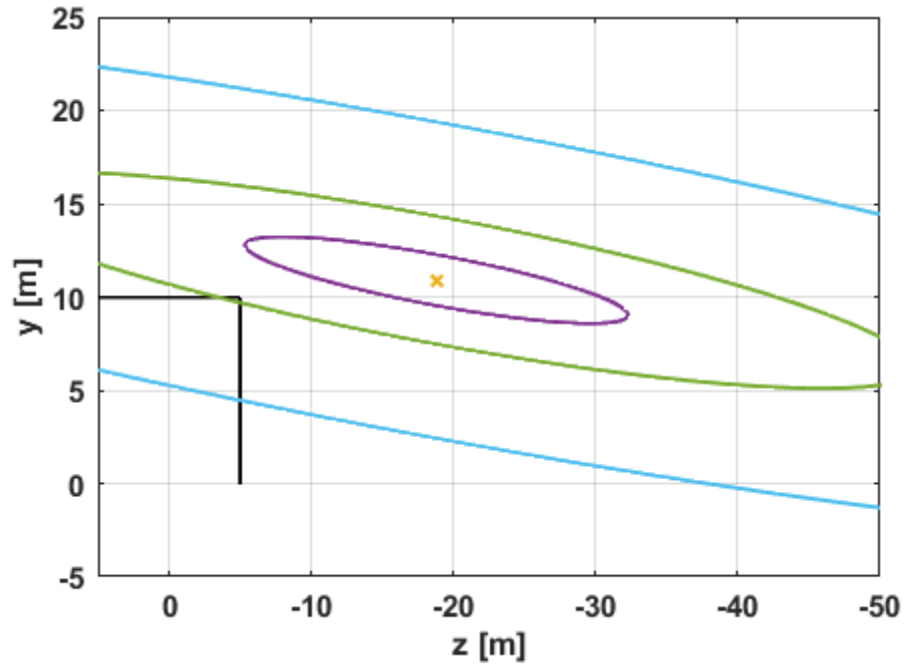


Figure 101: Illustration of modified ellipse-based vortex; black line outlines the obstacle; orange cross indicates vortex center; purple line delimits vortex core; green line delimits vortex size; and blue line indicates the end of the transition region.

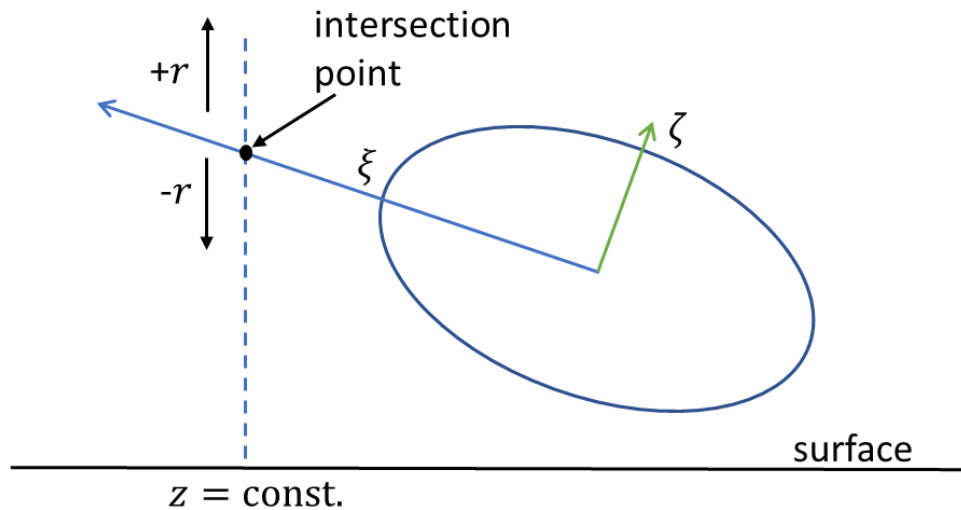


Figure 102: Illustration of slanted ellipse coordinate system and sign convention employed for radial coordinates from the center of the ellipse.

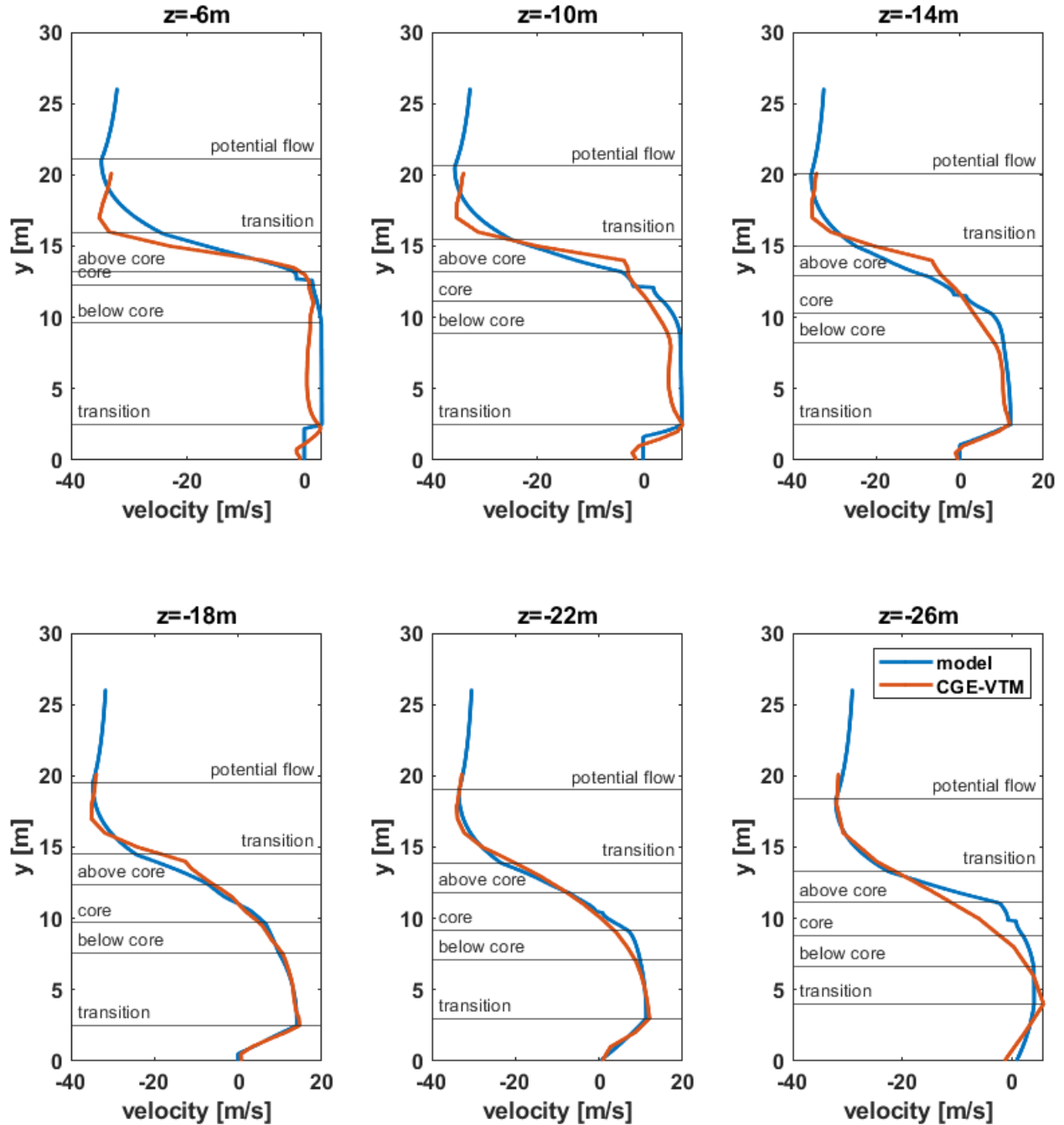


Figure 103: Longitudinal velocity profile at various span locations in the  $-6\text{ m} \geq z \geq -26\text{ m}$  range aft of the obstacle along the  $x = 0\text{ m}$  plane; only the slanted ellipse-based vortex model used; CGE/VorTran-M data queried in  $0.1\text{ m}$  height increments at every span location.

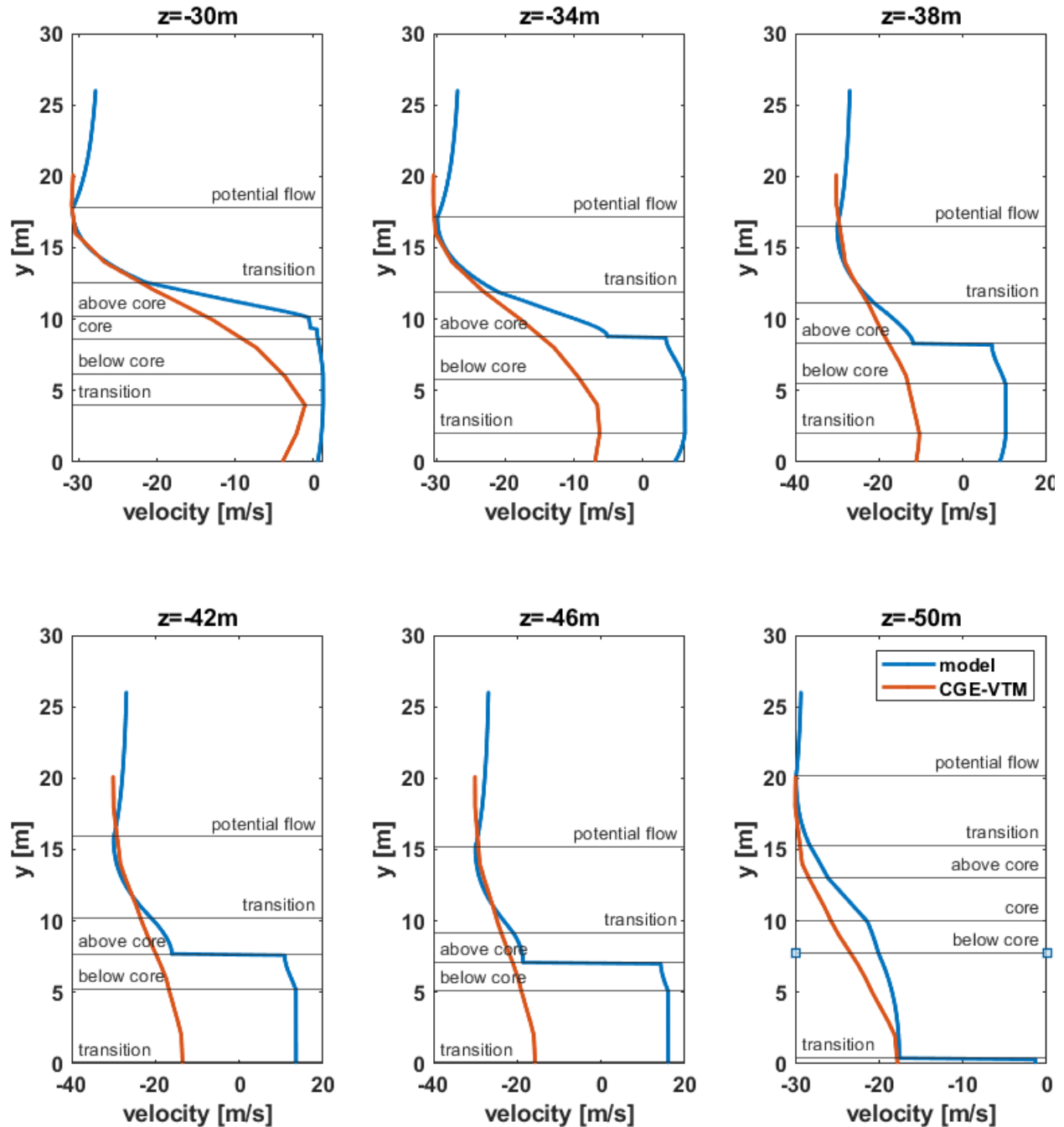


Figure 104: Longitudinal velocity profile at various span locations in the  $-30\text{ m} \geq z \geq -50\text{ m}$  range aft of the obstacle along the  $x = 0\text{ m}$ ; only the slanted ellipse-based vortex model used; CGE/VorTran-M data queried in  $0.1\text{ m}$  height increments at every span location.

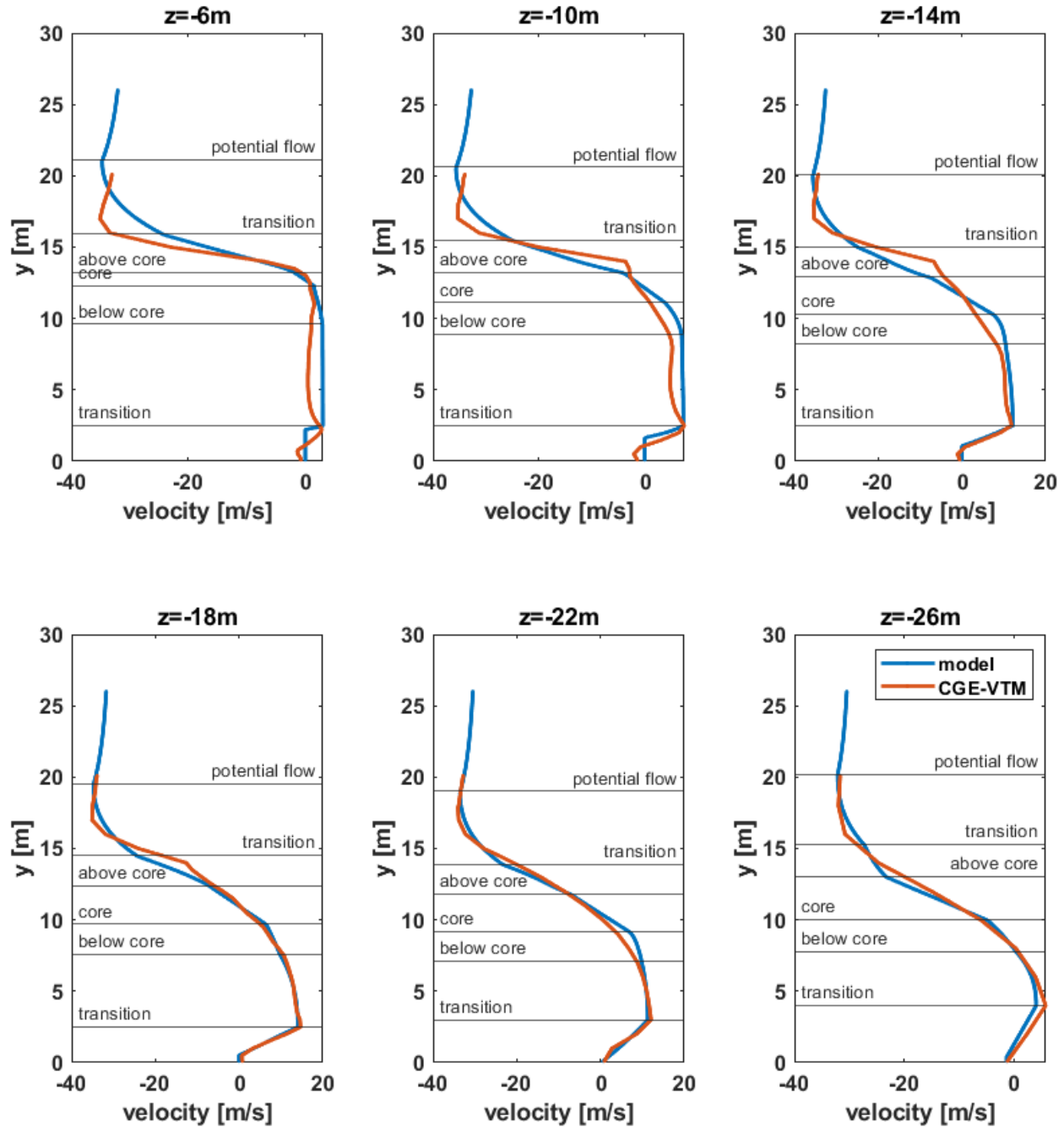


Figure 105: Longitudinal velocity profile at various span locations aft of the obstacle along the  $x = 0$  m plane using the slanted ellipse-based vortex model for  $-6 \text{ m} \geq z > -26 \text{ m}$  and the infinite vortex model for  $z = -26 \text{ m}$ ; CGE/VorTran-M data queried in  $0.1 \text{ m}$  height increments at every span location.

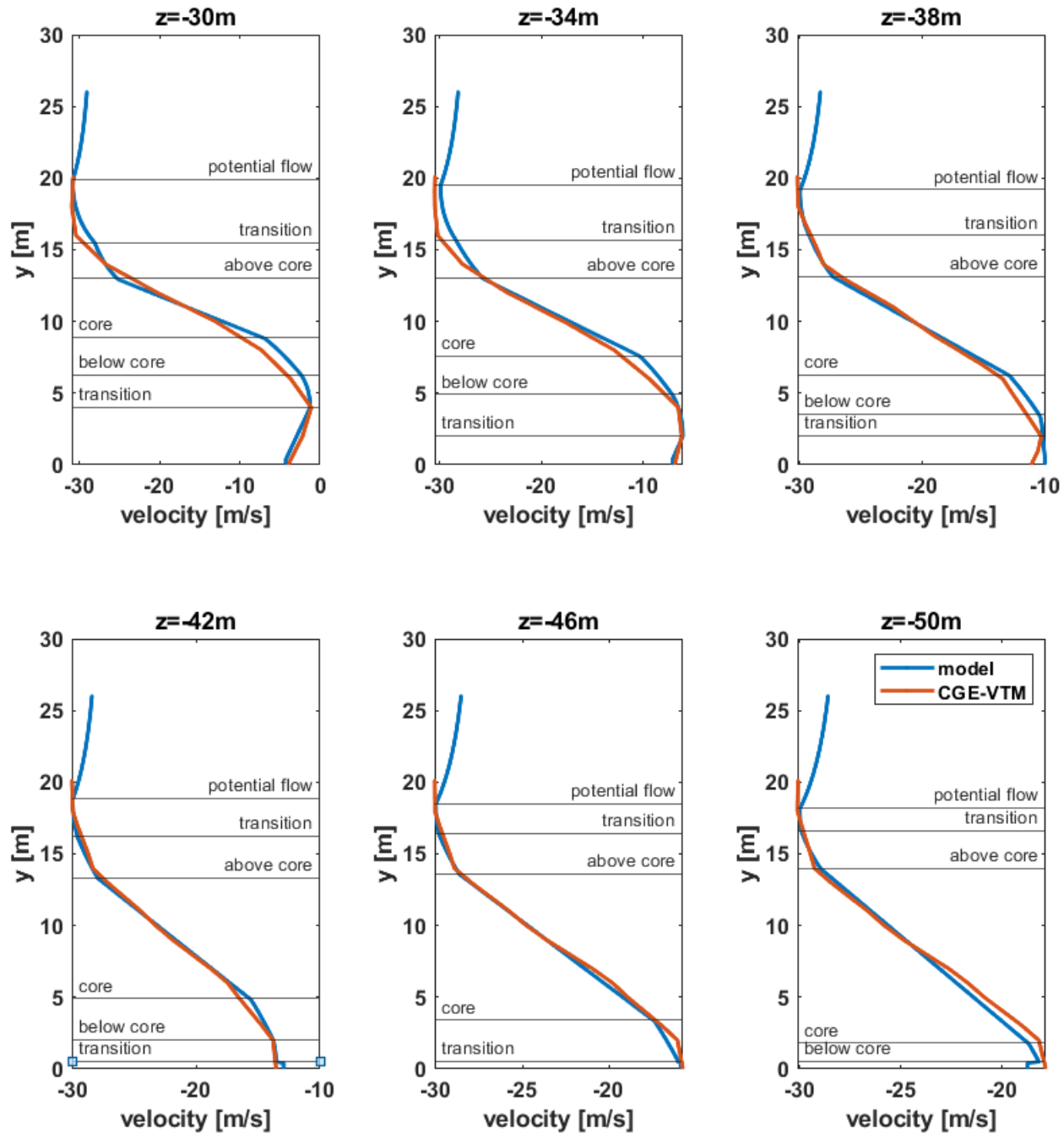


Figure 106: Longitudinal velocity profile at various span locations in the  $-30\text{ m} \geq z \geq -50\text{ m}$  range aft of the obstacle along the  $x = 0\text{ m}$  plane using the infinite vortex model; CGE/VorTran-M data queried in  $0.1\text{ m}$  height increments at every span location.

#### Side surface flow separation modeling

In addition to the top edge, flow also detaches at the side edges of the wind-facing surface and reattaches aft of the obstacle as depicted in Figure 9. A recirculation zone is created on the side surfaces due to interaction of the incoming freestream with this separation (see Figure 83(b) and Figure 107). Given the similarity of the side surface separation to the top surface separation considered previously, the infinite vortex model used to capture the velocity distribution across

the top surface was also employed here. Results are shown in Figure 108 across the  $y = 5$  m plane for separation over the left surface ( $x = 5$  m in Figure 107). Except for the region within a height of 1m from the side surface at  $z = 5$  m, reasonable agreement with CFD predictions is obtained without any modifications to the model. The discrepancy within the 1m region from the surface was also noted for the top surface and is deemed acceptable given that in practice, it is unlikely that the aircraft will be flying that close to the building during approach or take-off.

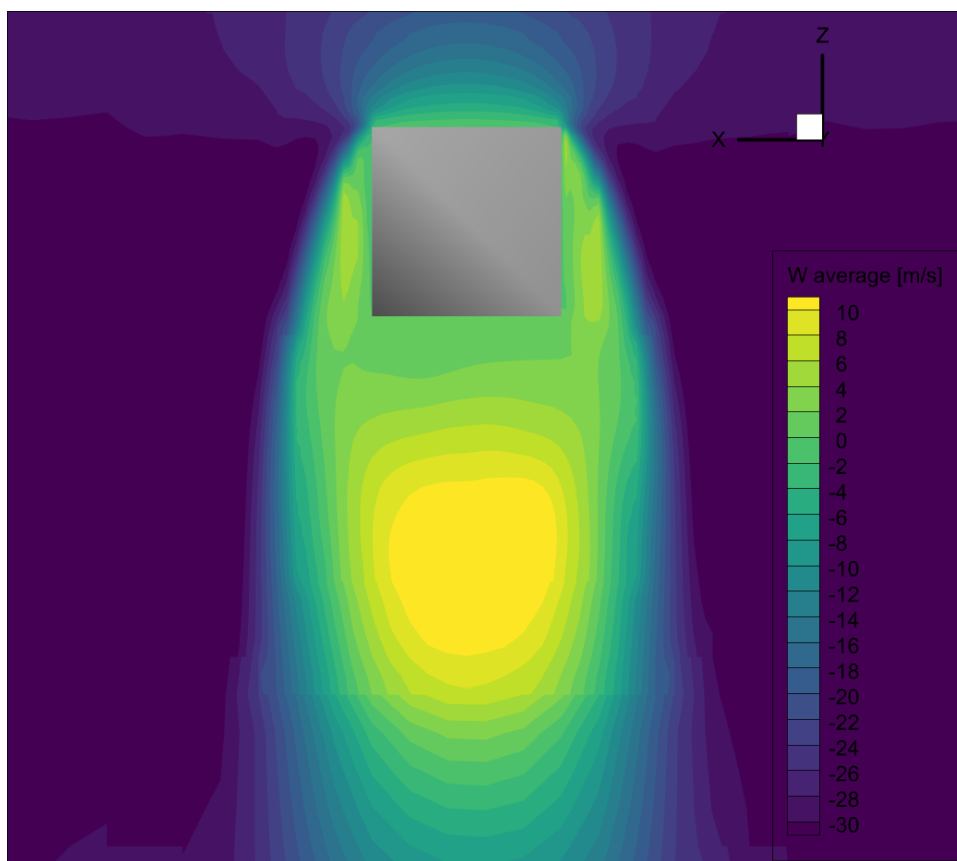


Figure 107: Contour of time-averaged longitudinal velocity for a 30m/s headwind simulation over a 10m cube using CGE/VTM along a midplane at  $y = 5$  m (top view).

Aft of the cube, the von Karman streak shown in Figure 83(e) is only observed during *time-accurate* solutions. Contours of *time-averaged* longitudinal velocity instead display a symmetrical distribution aft of the obstacle, as evident in Figure 107. It was surmised that the piece-wise ellipse-based and infinite vortex model used for the backwards step separation would be applicable here as well, with the primary difference being the starting height from where the velocity predictions would be made. In the previous section, velocity predictions were made from  $\hat{y} = 0$  m given that this coordinate represents the ground surface. The  $\hat{y}$  axis is introduced to represent vertical axis used by the model. For the backwards step separation considered in the previous section, the vertical axis used by the model corresponded with the CGE/VorTran-M  $y$  axis shown in Figure 107, and thus  $\hat{y} = y$ . In the current side surface separation case,  $\hat{y} = x$ . Unlike the backwards step separation case, no ground is present in the current case, but a symmetry condition exists along the  $x = 0$  m plane. From manual tuning, start heights of  $0.4H$  and  $0.25H$  for the ellipse-based and infinite vortex models, respectively, were found to produce reasonable agreement with prediction from the CGE/VorTran-M simulation as shown in Figure

109 and Figure 110, where velocity distributions at various span locations for  $x > 0$  m (left) are shown. Large discrepancies of approximately 15 m/s are noted at  $z = -6$  m in the above core and transition regions, but those are deemed acceptable since it is unlikely that the aircraft would be flying at that location. Offsets between model CGE/ VorTran-M predictions remain  $\leq 3$  m/s at the remaining span locations. Similar results are obtained for the  $x < 0$  m (right) region (not shown for brevity). With regards to parametrization, the results presented for the side flow separation are promising in that we were able to obtain adequate predictions for this flow feature by simple modifications to the inputs of the analytical vortex models for the backwards step flow separation.

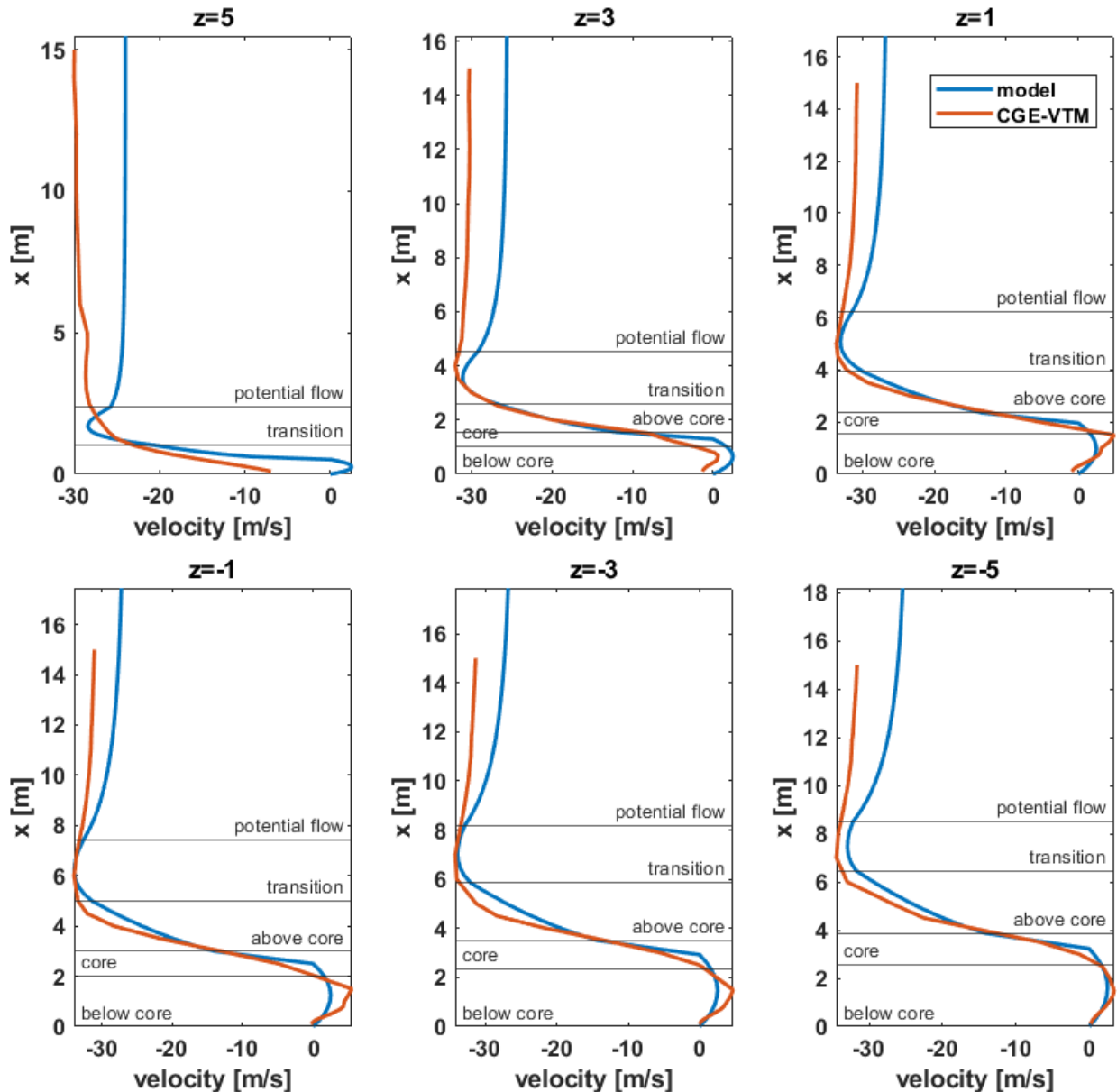


Figure 108: Longitudinal velocity profile over left side surface of cube ( $x = 5$  m);  $x$  distance plotted relative to surface; CGE data queried in 0.1 m increments at every span location.

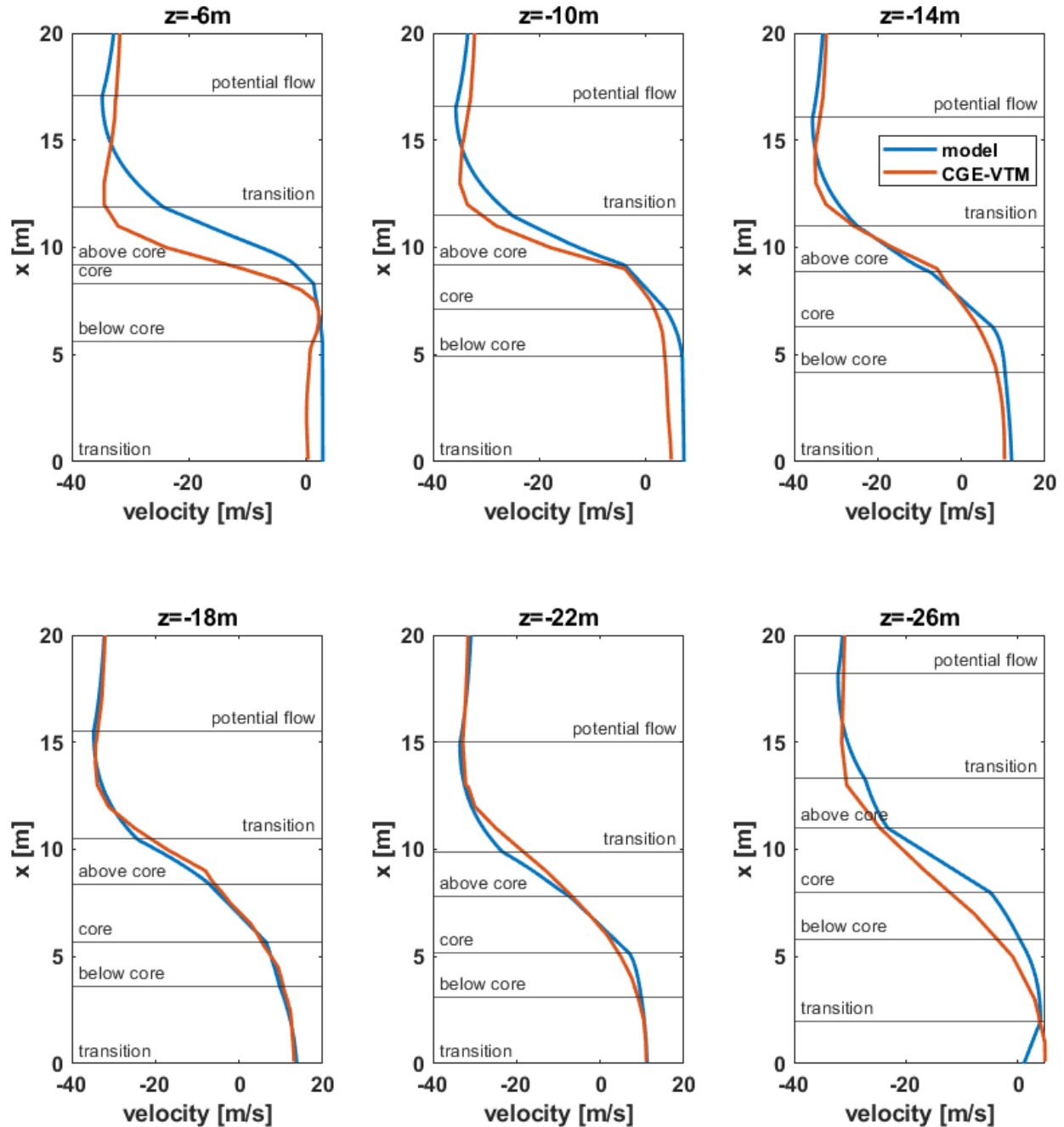


Figure 109: Longitudinal velocity profile at various span locations aft of the obstacle for  $x > 0$  m along the  $y = 5$  m plane using the slanted ellipse-based vortex model for  $-6 \text{ m} \geq z > -26 \text{ m}$  and the infinite vortex model for  $z = -26 \text{ m}$ ; CGE/VorTran-M data queried in  $0.1 \text{ m}$  increments at every span location.

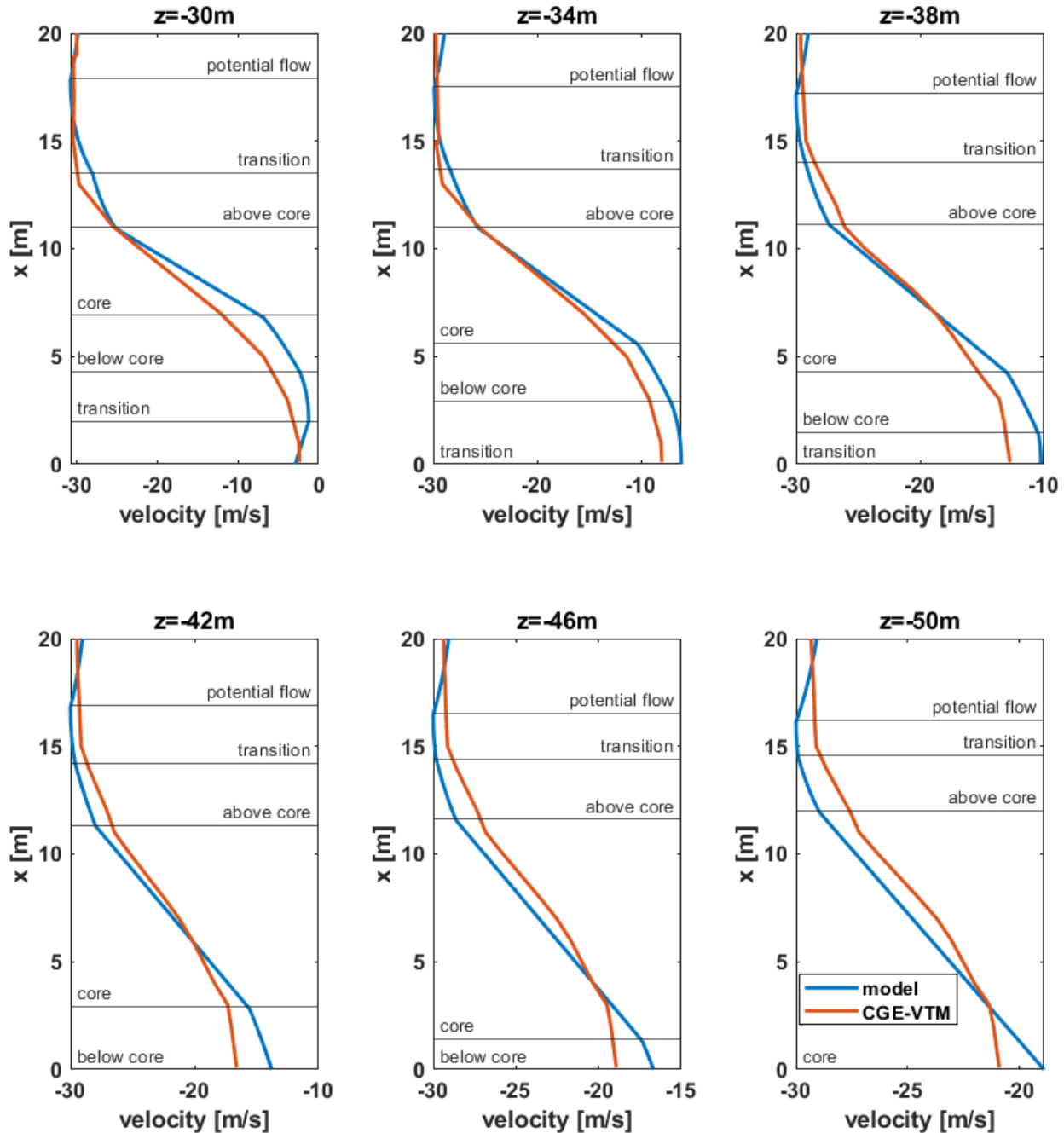


Figure 110: Longitudinal velocity profile at various span locations in the  $-30\text{ m} \geq z \geq -50\text{ m}$  range aft of the obstacle for  $x > 0\text{ m}$  in the  $y = 5\text{ m}$  plane using the infinite vortex model; CGE/VorTran-M data queried in  $0.1\text{ m}$  increments at every span location.

#### Corner flow separation modeling

For headwind flow conditions, edge vortices develop due to pressure gradients between side and top surfaces of the obstacle as illustrated in Figure 83(f). These edge vortices closely resemble roof corner vortices that the model was derived for, and thus, the original model equations shown in Table 2 were found to be applicable. Model inputs consist of the (1) the vortex radius ( $h$ ), (2) the location of point M ( $R_M$ ), and (3) the velocity at point M ( $U_M$ ). Model

parameter values and the corresponding velocity profile are shown in Table 7 and Figure 111, respectively, for the dashed 45° line shown in Figure 112. Good agreement is noted between the model predictions and those from CGE/VorTran-M.

Table 7: Vortex model parameter values.

| Parameter   | Value |
|-------------|-------|
| $h$ (m)     | 1.5   |
| $R_M$ (m)   | 6.0   |
| $U_M$ (m/s) | -32.8 |

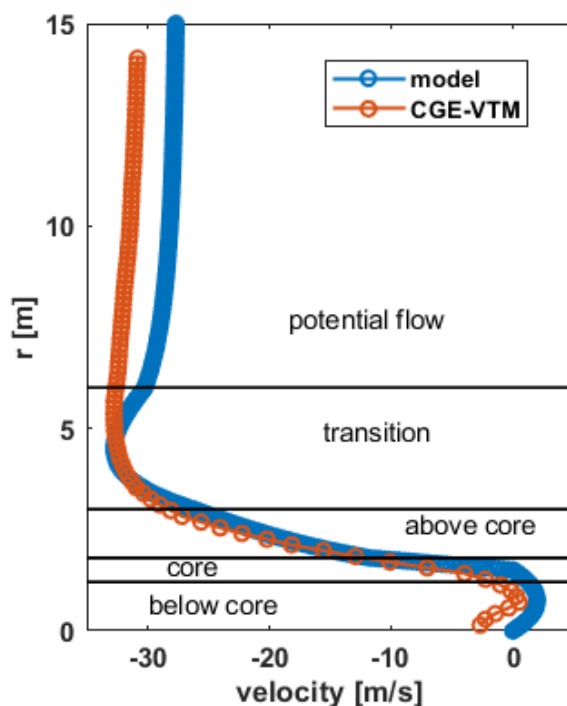


Figure 111: Longitudinal velocity profile along a 45° radial line from the corner of the cube.

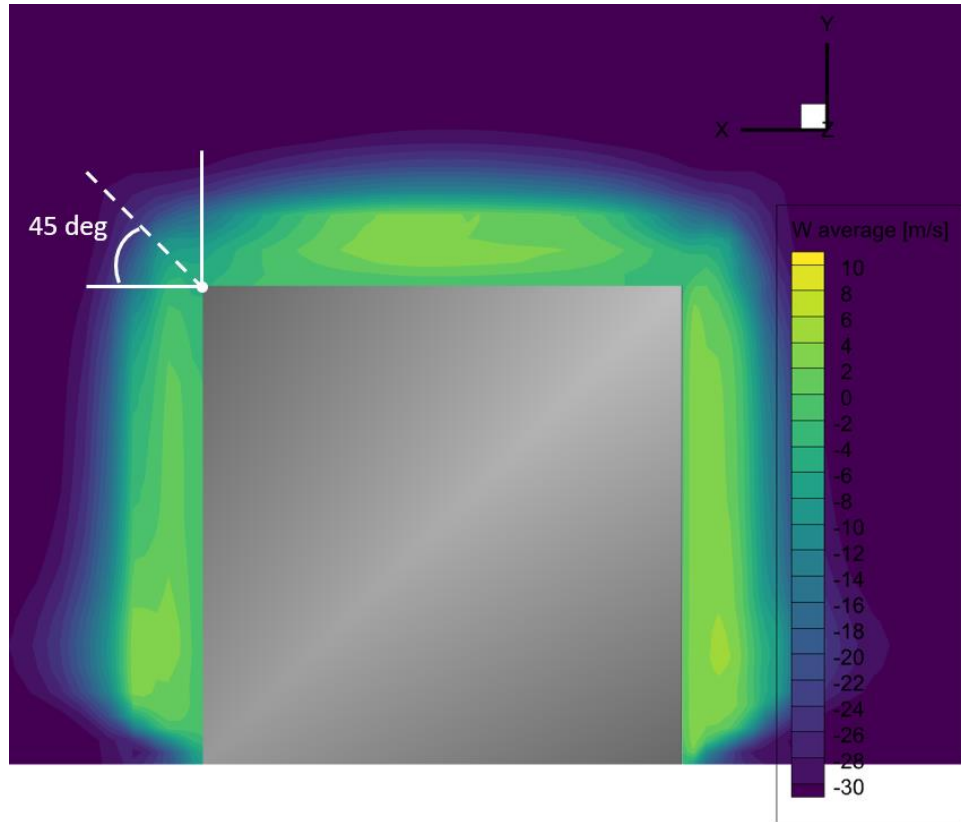


Figure 112: Contour of time-averaged longitudinal velocity for a 30 m/s headwind simulation over a 10 m cube along  $z = 0$  m plane; dashed line represents line along with corner vortex model fitting was initially performed.

In a similar manner to the top surface separation, the 1D prediction along the  $45^\circ$  line was extended to cover the entire 2D region over the surface ( $-5 \leq z \leq 5$  m). Here however, a function for the radial location of the vortex center  $r_0(z)$  was also derived in addition to functions for the vortex radius  $h(z)$ , location of vortex rotation point  $R_M(z)$ , and velocity  $U_M(z)$ , which represent standard inputs of the vortex models:

$$h(\bar{z}) = 0.07\bar{z} + 1, \quad (17)$$

$$R_M(z) = 0.24\bar{z} + 3.2, \quad (18)$$

$$V_M(\bar{z}) = -0.0014\bar{z}^2 + 0.251\bar{z} + 1.004, \quad (19)$$

$$r_0(\bar{z}) = \min[-0.024\bar{z}^2 + 0.456z - 0.46, h(\bar{z})] \quad (20)$$

where  $\bar{z} = z_{LE} - z$ , and where the leading-edge span location  $z_{LE} = 5$  m. From Eq. (20), the vortex center position can result in the vortex overlapping with the obstacle but is constrained to always being in contact with the edge. That is, there cannot be a gap between the vortex and the edge. This definition allowed for behavior at the  $z = 5$  m (leading edge) location to be adequately captured, as evident in Figure 113. At this location, the vortex center is located “inside” the obstacle along the  $45^\circ$  line. As flow progresses further downstream over the obstacle, both the location of the vortex center and vortex size increase radially in Figure 113, in a manner that parallels the expected physical behavior of the corner edge vortices. The model

thus allows for physical insights into the flow phenomenon. Note that leading edge flow profiles for top and side surfaces separations can also potentially be captured by introducing  $r_0(z)$  as an additional input to the fitting models.

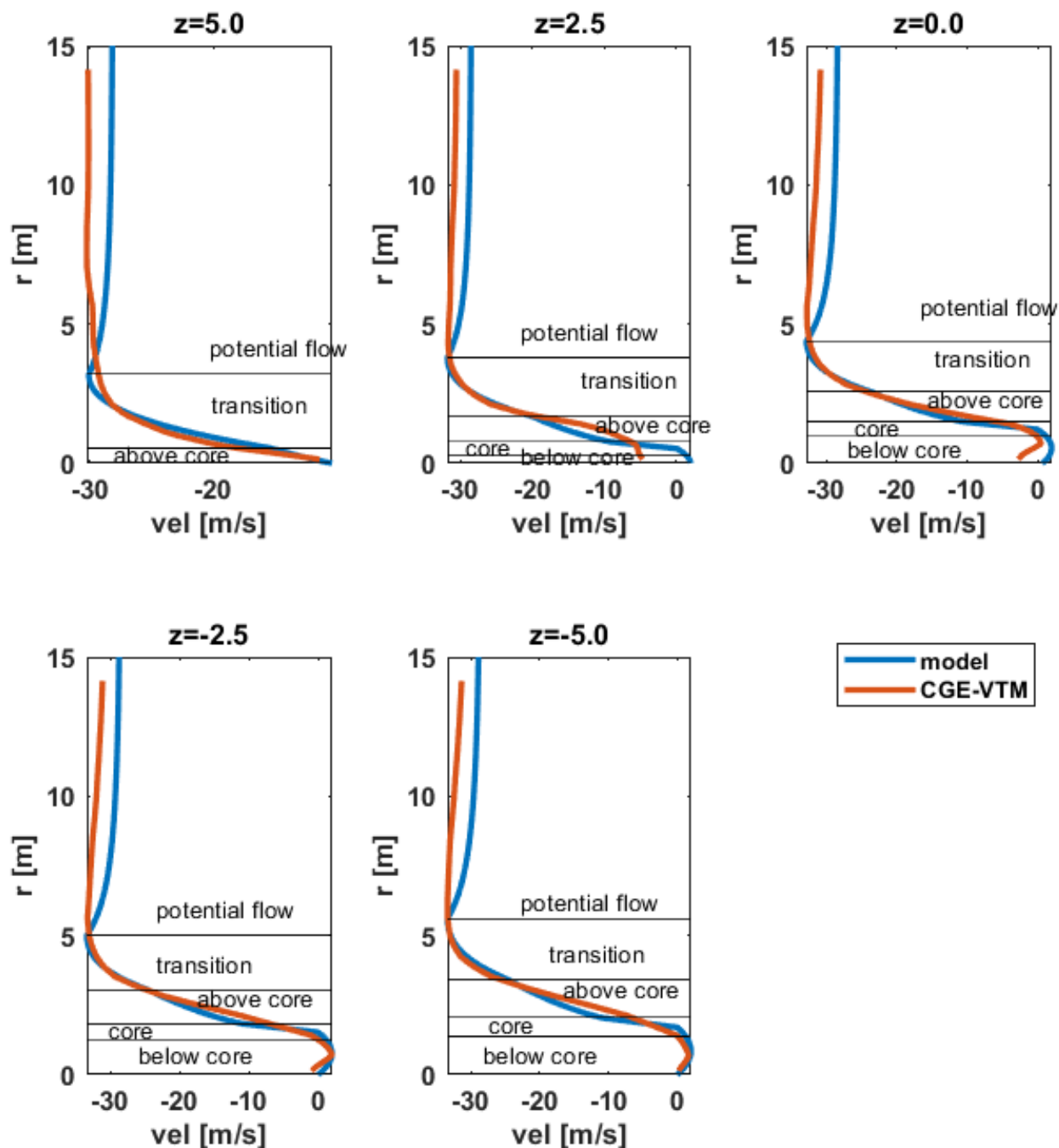


Figure 113: Longitudinal velocity profile along a  $45^\circ$  radial line from the corner of the cube at various span locations over the obstacle; CGE/VorTran-M data queried in 0.1 m increments.

## **Conclusions**

Work to date has focused on investigating the type of disturbance fields experienced by Naval aviators, and how to define the relevant primary flow structures and time scales. Approaches for developing simplified representations have been investigated, and as the effort progresses, we will commence developing representative surrogate flow fields that will be used in the flight simulations described above.

## **References**

1. Smith, M.J., A. Jones, A. Grubb, and J. Lefebvre. *Identification and Quantification of the Role of Turbulence in Aircraft/Ship Aerodynamics*. ONR Annual Review. 2020. Arlington, VA.
2. Silva, M.J., D.A. Wachspress, D.P. Gaublumme, E.W. Hayden, T.S. Davis, and T. Fean. *The Role of Modeling & Simulation in the Mitigation of V-22 Tiltrotor Formation Flight Wake-Induced Roll-off*. 72nd Annual Forum of the American Helicopter Society. 2016. West Palm Beach, FL.
3. Whitehouse, G.R. and R.E. Brown. *Modelling a Helicopter Rotor's Response to Encounters with Aircraft Wakes*. in 28th European Rotorcraft Forum. 2002. Bristol, UK.
4. Whitehouse, G.R. and R.E. Brown. *Helicopter Rotor Response to Wake Encounters in Ground Effect*. in 59th Annual Forum of the American Helicopter Society. 2003. Phoenix, AZ.
5. Whitehouse, G.R. and R.E. Brown, *Modelling the Mutual Distortions of Interacting Helicopter and Aircraft Wakes*. Journal of Aircraft, 2003. **Vol. 40**(No. 3): p. pp. 440-449.
6. Whitehouse, G.R. and R.E. Brown, *Modelling a Helicopter Rotor's Response to Wake Encounters*. The Aeronautical Journal, 2004. **Vol. 108**(No. 1079): p. pp. 15-26.
7. Whitehouse, G.R., *Helicopter Response to Vortex Encounters in the Near-Airfield Environment*, 2003, PhD Thesis, Department of Aeronautics Imperial College London.
8. ADS-33E-PRF. *Aeronautical Design Standard Performance Specification Handling Qualities Requirements for Military Rotorcraft*. in US Army Aviation and Missile Command, Aviation Engineering Directorate. 2000. Redstone Arsenal, Alabama.
9. Keller, J.D., D.A. Wachspress, and J.C. Hoffler. *Real Time Free Wake and Ship Airwake Model for Rotorcraft Flight Training Applications*. 71st Annual Forum of the American Helicopter Society 2015. Virginia Beach, VA.
10. Keller, J.D., D.A. Wachspress, and J.C. Hoffler. *Pilot Evaluation of a Real-Time Free Wake Model in a Navy Rotorcraft Fleet Trainer*. 73rd Annual Forum of the American Helicopter Society 2017. Fort Worth, TX.
11. Polsky, S.A. *A Computational Study of Unsteady Ship Airwake*. in 40th AIAA Aerospace Sciences Meeting & Exhibit. 2002. Reno, NV: AIAA 2002-1022.
12. Keller, J.D., G.R. Whitehouse, A.H. Boschitsch, J. Nadal, J. Jeffords, and M. Quire. *Computational Fluid Dynamics for Flight Simulator Ship Airwake Modeling*. in Interservice/Industry Training, Simulation, and Education Conference (I/ITSEC) 2007. 2007. Orlando, FL.
13. Padfield, G., *Helicopter Flight Dynamics: The Theory and Application of Flying Qualities and Simulation Modeling*. AIAA Educational Series, ed. J.S. Przemieniecki. 1996.
14. Payne, P.R., *Helicopter Dynamics and Aerodynamics*. 1959, New York: The Macmillan Company.
15. Howlett, J.J., *UH-60A Black Hawk Engineering Simulation Program - Volume I - Mathematical Model*, 1981, NASA-CR-166309.
16. Keller, J.D., D.A. Wachspress, J.C. Hoffler, N.J. Kachman, and J. Nichols. *Application of a Real Time Free Wake Induced Velocity Model in a Naval Rotorcraft Flight Trainer*. in 67th Annual Forum of the American Helicopter Society 2011. Virginia Beach, VA.

17. Forsythe, J.R., C.E. Lynch, S. Polsky, and P.R. Spalart. *Coupled Flight Simulator and CFD Calculations of Ship Airwake using HPCMP CREATE™-AV Kestrel*. 53rd AIAA Aerospace Sciences Meeting. 2015. Kissimmee, FL.: AIAA-2015-0556.
18. Beal, T.R., *Digital simulation of atmospheric turbulence for Dryden and von Karman models*. Journal of Guidance, Control, and Dynamics, 1993. **Vol. 16**(No. 1): p. pp. 132-138.
19. Dreier, M.E., *The Influence of a Trailing Tip Vortex on a Thrusting Rotor*, 1977, Penn. State University.
20. Padfield, G.D. and G.P. Turner, *Helicopter Encounters with Aircraft Vortex Wakes*. The Aeronautical Journal, 2001. **105**(1043): p. pp. 1-8.
21. Leishman, J.G., *Principles of Helicopter Aerodynamics*. 2nd ed. 2006, Cambridge, UK: Cambridge University Press.
22. Johnson, W., *Helicopter Theory*. 1980, Princeton, NJ: Princeton University Press.
23. Padfield, G.D. and G.P. Turner. *Helicopter Encounters with Aircraft Vortex Wakes*. in *25th European Rotorcraft Forum*. 1999. Rome, Italy.
24. Padfield, G.D., B. Manimala, and G.P. Turner. *A Severity Analysis for Rotorcraft Encounters with Vortex Wakes*. in *28th European Rotorcraft Forum*. 2002. Bristol, UK.
25. Turner, G.P., G.D. Padfield, and M. Harris, *Encounters with Aircraft Vortex Wakes: The Impact on Helicopter Handling Qualities*. Journal of Aircraft, 2002. **39**(5): p. pp. 839-849.
26. Wachspress, D.A., T.R. Quackenbush, and J.D. Keller. *Rotor Wake Evolution and Vortex Wake Encounter Analysis Methods for Terminal Area Operations*. in *AHS 58th Annual Forum*. 2002. Montreal, Canada.
27. Bilanin, A.J., M.E. Teske, and G.G. Williamson, *Vortex Interactions and Decay in Aircraft Wakes*. AIAA Journal, 1977. **15**(2): p. pp. 250-260.
28. Bilanin, A.J., M.E. Teske, and J.E. Hirsh, *Neutral Atmospheric Effects on the Dissipation of Aircraft Vortex Wakes*. AIAA Journal, 1978. **16**(9): p. 956-961.
29. Teske, M.E., T.R. Quackenbush, A.J. Bilanin, and D.A. Wachspress. *Vortex Roll-up Merging and Decay with the UNIWAKE Computer Program*. in *Aircraft Wake Vortices Conference*. 1991. Washington, D.C.
30. Quackenbush, T.R., M.E. Teske, and A.J. Bilanin. *Computation of Wake/Exhaust Mixing Downstream of Advanced Transport Aircraft*. in *AIAA 24th Fluid Dynamics Conference*. 1993. Orlando, FL: AIAA-1993-2944.
31. Quackenbush, T.R., M.E. Teske, and A.J. Bilanin. *Dynamics of Exhaust Plume Entrainment in Aircraft Vortex Wakes*. in *34th Aerospace Sciences Meeting & Exhibit*. 1996. Reno, NV: AIAA-1996-0747.
32. Teske, M.E., T.R. Quackenbush, A.J. Bilanin, and R.I. Sykes. *Long-Time Aircraft Vortex Wake Predictions and Plume Evolution Consequences*. in *28th AIAA Fluid Dynamics Conference*. 1997. Snowmass, CO: AIAA-1997-1843
33. McKillip Jr., R.M., D.A. Wachspress, J.D. Keller, G.R. Whitehouse, T.R. Quackenbush, and A.H. Boschitsch. *Distributed Multi-Vehicle Simulation Including High-Order Airwake Representation*. 60th Annual Forum of the American Helicopter Society 2004. Baltimore, MD.
34. Whitehouse, G.R., A.H. Boschitsch, and J.D. Keller. *Development and Testing of a Lagrangian-Eulerian CFD Analysis for Moving Body and Interactional Aerodynamics Analysis*. 2019 AIAA SciTech Forum. 2019. San Diego, CA: AIAA-2019-2117.

35. Whitehouse, G.R., A.H. Boschitsch, J.D. Keller, and D.A. Wachspress, *Next Generation of Fast Moderate Fidelity Methods for Interactional Aerodynamics*, 2018, Continuum Dynamics, Inc. CDI-TR-18-4: Final Report under ONR contract N00014-14-C-0014.
36. Tinney, C.E. and L.S. Ukeiley, *A Study of a 3-D Double Backward-Facing Step*. Experiments in Fluids, 2009. **Vol. 47**(No. 3): p. pp. 427-438.
37. Herry, B., *Aerodynamic Study of a 3D Backward Facing Double Step Applied to Safer Launch and Recovery of Helicopters on Ships*, 2010, PhD Thesis, Université de Valenciennes et du Hainaut Cambrésis.
38. Wilkinson, C.H., M. Roscoe, and G. VanderVliet. *Determining Fidelity Standards for the Shipboard Launch and Recovery Task*. AIAA Modeling and Simulation Technologies Conference and Exhibit. 2001. Montreal, Canada: AIAA-2001-4062.
39. Mantey, D., *Ride the Air Wake*, in *Product Design and Development* 2011, Advantage Business Media.
40. Carpenter, P.J. and B. Fridovich, *Effect of a Rapid Blade-Pitch Increase on the Thrust and Induced-Velocity Response of a Full-Scale Helicopter Rotor*, 1953, NACA-TN-3044.
41. Hansen, A.C., J.A. Peterka, and Cermak J.E., *Wind-Tunnel Measurements in the Wake of a Simple Structure in a Simulated Atmospheric Flow*, 1975: NASA-CR-2540.
42. Becker, S., H. Lienhart, and F. Durst, *Flow Around Three-Dimensional Obstacles in Boundary Layers*. Journal of Wind Engineering and Industrial Aerodynamics, 2002. **Vol. 90**(No. 4-5): p. pp. 265-279.
43. Forrest, J.S., I. Owen, G.D. Padfield, and S.J. Hodge, *Ship-Helicopter Operating Limits Prediction Using Piloted Flight Simulation and Time-Accurate Airwakes*. Journal of Aircraft, 2012. **Vol. 4**(No. 4): p. pp. 1020-1031.
44. Driver, D.M., H.L. Seegmiller, and J.G. Marvin, *Time-Dependent Behavior of a Reattaching Shear Layer*. AIAA Journal, 1987. **Vol. 25**(No. 7): p. pp. 914-919.
45. Eaton, J.K. and J.P. Johnston, *Review of Research on Subsonic Turbulent-Flow Reattachment*. AIAA Journal, 1981. **Vol. 19**(No. 9): p. pp. 1093-1100.
46. Eaton, J.K. and J.P. Johnston, *Low Frequency Unsteadiness of a Reattaching Turbulent Shear Layer*, in *Turbulent Shear Flows 3*. 1982, Springer. p. pp. 162-170.
47. Abbott, D.E. and S.J. Kline, *Experimental Investigations of Subsonic Turbulent Flow Over Single and Double Backward-Facing Steps*. ASME Journal of Basic Engineering, 1962. **Vol. 84**(No. 3): p. pp. 317-325.
48. De Brederode, V.A.S.L., *Three-Dimensional Effects in Nominally Two-Dimensional Flow.*, 1974, PhD Thesis, Imperial College of Science, Technology and Medicine.
49. Spazzini, P.G., G. Iuso, M. Onorato, N. Zurlo, and G.M. Di Cicca, *Unsteady Behavior of Back-Facing Step Flow*. Experiments in Fluids, 2001. **Vol. 30**(No. 5): p. pp. 551-561.
50. Kim, J., S.J. Kline, and J.P. Johnston, *Investigation of a Reattaching Turbulent Shear Layer: Flow Over a Backward-Facing Step*. Journal of Fluid Engineering, 1980. **Vol. 102**(No. 3): p. pp. 302-308.
51. Ma, X. and A. Schröder, *Analysis of Flapping Motion of Reattaching Shear Layer Behind a Two-Dimensional Backward-Facing Step*. Physics of Fluids, 2017. **Vol. 29**(No. 11).
52. Hoerner, S.F., *Fluid-Dynamic Drag*. 1965, New York, NY: Hoerner Fluid Dynamics.
53. Yu, D. and A. Kareem, *Parametric Study of Flow Around Rectangular Prisms Using LES*. Journal of Wind Engineering and Industrial Aerodynamics, 1998. **Vol. 77-78**: p. pp. 653-662.

54. Okajima, A., *Strouhal Numbers of Rectangular Cylinders*. Journal of Fluid Mechanics, 1982. **Vol. 123**: p. pp. 379-398.
55. Sousa, J.M.M., *Turbulent Flow Around a Surface-Mounted Obstacle Using 2D-3C DPIV*. Experiments in Fluids, 2002. **Vol. 33**(No. 6): p. pp. 854-862.
56. Hunt, J.C.R., C.J. Abell, J.A. Peterka, and H. Woo, *Kinematical Studies of the Flows Around Free or Surface-Mounted Obstacles; Applying Topology to Flow Visualization*. Journal of Fluid Mechanics, 1978. **Vol. 86**(No. 1): p. pp. 179-200.
57. Yakhot, A., H. Liu, and N. Nikitin, *Turbulent Flow Around a Wall-Mounted Cube: A Direct Numerical Simulation*. International Journal of Heat and Fluid Flow, 2006. **Vol. 27**(No. 6): p. pp. 994-1009.
58. Kim, K.C., H.S. Ji, and S.H. Seong, *Flow Structure around a 3-D Rectangular Prism in a Turbulent Boundary Layer*. Journal of Wind Engineering and Industrial Aerodynamics, 2003. **Vol. 91**(No. 5): p. pp. 653-669.
59. Larose, G.L. and A. D'Auteuil, *On The Reynolds Number Sensitivity of The Aerodynamics of Bluff Bodies With Sharp Edges*. Journal of Wind Engineering and Industrial Aerodynamics, 2006. **Vol. 94**(No. 5): p. pp. 365-376.
60. Healey, J.V., *Establishing A Database for Flight in the Wakes of Structures*. Journal of Aircraft, 1992. **Vol. 29**(No. 4): p. pp. 559-564.
61. Kareem, A., *Measurements of Pressure and Force Fields on Building Models in Simulated Atmospheric Flows*. Journal of Wind Engineering and Industrial Aerodynamics, 1990. **Vol. 36**(No. 1): p. pp. 589-599.
62. Polsky, S. and C.W.S. Bruner. *A Computational Study of Unsteady Ship Airwake*. NATO Research and Technology Organisation Applied Vehicle Technology Panel Symposium. 2003. Leon, Norway: .
63. Cherry, B.E. and M.M. Constantino. *The Burble Effect: Superstructure and Flight Deck Effects on Carrier Air Wake*. American Society of Naval Engineers Launch & Recovery Symposium 2010. 2010. Arlington, VA.
64. Sezer-Uzol, N., A. Sharma, and L.N. Long, *Computational Fluid Dynamics Simulations of Ship Airwake*. Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering,, 2005. **Vol. 219**(No. 5): p. pp. 369-395.
65. Forrest, J.S. and I. Owen, *An Investigation of Ship Airwakes Using Detached-Eddy Simulation*. Computers and Fluids, 2010. **Vol. 39**(No. 4): p. pp. 656-673.
66. Lee, D., N. Sezer-Uzol, J.F. Horn, and L.N. Long, *Simulation of Helicopter Shipboard Launch and Recovery with Time-Accurate Airwakes*. Journal of Aircraft, 2005. **Vol. 42**(No. 2): p. 448-461.
67. Polsky, S.A. and D.S. Miklosovic. *CFD Study of Bluff Body Wake From a Hangar With Comparison to Experimental Data*. 29th AIAA Applied Aerodynamics Conference. 2011. Honolulu, HI: AIAA-2011-3351.
68. Polsky, S.A. and C. Bruner. *Time-accurate Computational Simulations of an LHA Ship Airwake*. in *18th AIAA Applied Aerodynamics Conference*. 2000. Denver, CO: AIAA 2000-4126.
69. Rajagopalan, G., D. Schaller, A.J. Wadcock, G.K. Yamauchi, J.T. Heineck, and M.J. Silva. *Experimental and Computational Simulation of a Model Ship in a Wind Tunnel*. 43rd AIAA Aerospace Sciences Meeting and Exhibit. 2005. Reno, NV: AIAA-2005-1347.
70. Kumar, K.V., M.P. Mathew, S.N. Singh, S.S. Sinha, and R. Vijayakumar. *Experimental Investigation of Flow Over The Flight Deck of a Generic Aircraft Carrier*. Warship 2018:

- Procurement of Future Surface Vessel*. 2018. London, UK: Royal Institution of Naval Architects.
71. Cvengros, M., *CFD Simulations of Unsteady LHA Ship Airwake in OpenFOAM*, 2018, Royal Institute of Technology: Stockholm, Sweden.
  72. Kelly, M.F., *The Development, Validation, and Integration of Aircraft Carrier Airwakes for Piloted Flight Simulation*, 2018, University of Liverpool: Liverpool, UK.
  73. Watson, N.A., M.F. Kelly, I. Owen, S.J. Hodge, and M.D. White, *Computational and Experimental Modelling Study of the Unsteady Airflow over the Aircraft Carrier HMS Queen Elizabeth*. Ocean Engineering, 2019. **Vol. 172**: p. pp. 562-574.
  74. Kozmar, H., *Surface Pressure on a Cubic Building Exerted by Conical Vortices*. Journal of Fluids and Structures, 2020. **Vol. 92**.
  75. Liu, T., H. Zhang, and D. Xin, *Experimental Study on Mitigating Extreme Roof Suctions by Passive Vortex Generators*. Journal of Wind Engineering and Industrial Aerodynamics, 2021. **Vol. 219**.
  76. Asghari Mooneghi, M. and R. Kargarmoakhar, *Aerodynamic Mitigation and Shape Optimization of Buildings: Review*. Journal of Building Engineering, 2016. **Vol.6**: p. pp. 225-235.
  77. He, J. and C.C.S. Song, *A Numerical Study of Wind Flow around the TTU Building and the Roof Corner Vortex*. Journal of Wind Engineering and Industrial Aerodynamics, 1997. **Vols. 67-69**: p. pp. 547-558.
  78. Kawai, H., *Structure of Conical Vortices Related With Suction Fluctuation on a Flat Roof in Oblique Smooth and Turbulent Flows*. Journal of Wind Engineering and Industrial Aerodynamics, 1997. **Vol. 69-71**: p. pp. 579-588.
  79. Kozmar, H., *Characteristics of Natural Wind Simulations in The TUM Boundary Layer Wind Tunnel*. Theoretical and Applied Climatology, 2011. **Vol. 106**(No. 1-2): p. pp. 95-104.
  80. Daley, W.H.I., *Flow Visualization of the Airwake Around a Model of a TAWARA Class LHA in a Simulated Atmospheric Boundary Layer*, 1988, Naval Postgraduate School: Monterey, CA.
  81. Sydney, A.J., J.P. Ramsey, and J.I. Milluzzo. *Time-Resolved PIV Measurements of Ship Motion and Orientation Effects on Airwake Development*. 35th AIAA Applied Aerodynamics Conference. 2017. Denver, CO: AIAA-2017-4230.
  82. Yuan, W., A. Wall, and R. Lee. *Simulations of Unsteady Airwakes Behind Ships in Motion*. 31st Congress of the International Council of the Aeronautical Sciences. 2018. Belo Horizonte, Brazil.
  83. Dooley, G., J. Ezequiel Martin, J.H.J. Buchholz, and P.M. Carrica, *Ship Airwakes in Waves and Motions and Effects on Helicopter Operation*,. Computers and Fluids, 2020. **Vol. 208**.
  84. Krebill, A., *Effect of Ship Motion on Ship Airwake Aerodynamics*, 2020, The University of Iowa: Iowa City, IA.
  85. Tai, T. *Effect of Ship Motion on DD-963 Ship Airwake Simulated by Multizone Navier-Stokes Solution*. Twenty-First Symposium on Naval Hydrodynamics. 1997. Trondheim, Norway.
  86. Keller, J.D., G.R. Whitehouse, and A.H. Boschitsch, *CFD Methodology Validation Report for Ship Airwake Model and Database*, 2007, CDI Report No. 07-04.

87. Woofield, A.A. and B.N. Tomlinson. *Ship Airwakes - A New Generic Model for Piloted Simulation*. NATO AGARD Symposium on flight simulation – Where are the challenges? 1995. AGARD-CP-577.
88. Kastner-Klein, P. and M.W. Rotach, *Mean Flow and Turbulence Characteristics in an Urban Roughness Sublayer*. Boundary-Layer Meteorology, 2004. **Vol. 111**(No. 1): p. pp. 55-84.
89. Whitehouse, G.R., A.H. Boschitsch, and P.V. Danilov. *Efficient Cartesian Grid CFD-Based Methods for Aeroelastic Analysis of Wind Turbines*. 2022 AIAA SciTech Forum. 2022. San Diego, CA: AIAA-2022-1017.
90. Whitehouse, G.R., A.H. Boschitsch, and B.S. Silbaugh. *Development and Testing of a Low Cost CFD Based Analysis for Surface Mesh Interrogation & Refinement for Applied Aerodynamics Applications*. 56th AIAA Aerospace Sciences Meeting, AIAA SciTech Forum. 2018. Kissimmee, FL: AIAA-2018-0541.
91. Klotz, L., S. Goujon-Durand, J. Rokicki, and J.E. Wesfreid, *Experimental Investigation of Flow Behind a Cube for Moderate Reynolds Numbers*. Journal of Fluid Mechanics, 2014. **Vol. 750**: p. pp. 73–98.
92. Banks, D. and R.N. Meroney, *A Model of Roof-Top Surface Pressures Produced by Conical Vortices: Model Development*. Wind and Structures, 2001. **Vol. 4**(No. 3): p. pp. 227–246.
93. Banks, D. and R.N. Meroney, *A Model of Roof-Top Surface Pressures Produced by Conical Vortices: Evaluation and Implications*. Wind and Structures, 2001. **Vol. 4**(No. 3): p. pp. 279–298.