Spring 2017

Ship Airwake Computational Modeling Method Verification

Gregory Dooley

SHIP AIRWAKE COMPUTATIONAL MODELING METHOD VERIFICATION

by

Gregory Dooley

A thesis submitted in partial fulfillment of the requirements
for graduation with Honors in the Mechanical Engineering

Pablo Carrica
Thesis Mentor

Spring 2017

All requirements for graduation with Honors in the
Mechanical Engineering have been completed.

H.S. Udaykumar
Mechanical Engineering Honors Advisor

Ship Airwake Computational Modeling Method Validation

by

Gregory Dooley

A thesis submitted in partial fulfillment of the requirements for graduation with Honors in the Department of Mechanical and Industrial Engineering

---

Pablo Carrica

Honors Thesis Supervisors

Spring 2017

All requirements for graduation with Honors in the Department of Your Major have been completed.

H.S. Udaykumar

Major Honors Advisor

Jon Kuhl

Associate Dean, College of Engineering

Table of Contents

Abstract

Computational Fluid Dynamics (CFD) simulations offer a feasible solution to resolving the behavior of ship airwake behavior in the interest of flight operations near a moving ship as opposed to large full scale experimental methods. Wind tunnel data has been collected for several wind-over-deck (WOD) conditions using the simple frigate shape (SFS2) conducted by the National Research Council of Canada. In order to test the ability of the CFD code REX to accurately simulate these conditions and provide meaningful data a comprehensive validation study was conducted. Wind tunnel data is provided for both a headwind and Green 45° scenario. Both were recreated and simulated using REX. Comparison of averaged flow velocities and turbulence intensities in the area of the ship's flight deck show strong agreement with the provided wind tunnel data. Differences in velocity magnitude and turbulence intensity were seen specifically in the areas of high turbulence but overall the flow behavior modeled using REX agrees with the trends seen in the wind tunnel data. The conclusion is thus drawn that REX can be used to accurately model the airwake behavior around a ship geometry under varying flow conditions.

Introduction

I. Background Information

There is a current effort at the University of Iowa to accurately model the influence of ship motions on ship aerodynamics. One part of this undertaking is using computational methods to accurately model these effects. Ship motions can affect the behavior of the airflow around the ship and its velocity relative to the ship's superstructure. Understanding these flow fluctuations is important due to the impact they have on aircraft during launch and recovery operations. The turbulence generated near a ship's superstructure during different maneuvers generates unsteadiness in the surrounding flow field. This poses a danger to those operating aircraft in this area. Being able to accurately model this airflow will provide the necessary data for simulating realistic flight conditions for pilot training and potentially increase overall operational safety for those aboard ships with aircraft operation. There is currently a very limited amount of high quality experimental data for ship airwakes under prescribed ship motions. The at-sea trials that have been conducted are very expensive to execute and are only capable of collecting limited amounts of data regarding ship and aircraft aerodynamics and their interaction. Also, the use of scale models to collect data suffers greatly from scaling effects, especially when trying to accurately measure small turbulent flow features. The use of computational fluid dynamics (CFD) offers a cost-effective alternative to experimental methods. CFD has the ability to be used for a wide range of applications without extremely costly modifications such as those encountered during physical experimentation. The use of computational methods also provides a more direct path for the implementation of generated data sets in training simulators. For the effort at the University of Iowa, the proprietary advanced naval hydrodynamics CFD code REX was used to run all computational simulations. The CFD results generated through the use of REX will eventually be compared to experimental results captured through the use of a large water flume at the University of Iowa. The use of these two methods will result in strong validation of the results as well as provide a validation data set for a given geometry. There are currently existing data sets generated through the use of wind tunnels, which is the current preferred method when it comes to studying ship airwakes. The ability of the CFD code REX to be used for modeling these ship airwake characteristics was tested through the completion of a comprehensive comparison study. The wind tunnel experiments presented by Forrest and Owen (2010), performed by the National Research Council of Canada (NRCC), were computationally modeled and simulated using REX. This comparison study is to serve as initial code validation for REX in the effort discussed above.

II. Objective

As a means to measure the accuracy of future airwake simulations a validation study was to be conducted comparing CFD results to existing wind tunnel data sets. The validation study is to consist of two different flow conditions, one with incoming airflow parallel (headwind) to the ship's hull and the other with airflow at 45° to the starboard side of the ship's geometry (Green 45°). Measurements for both cases will be taken in the same region of the model ship's flight deck. This area is based on the wind tunnel data set presented by Forrest and Owen (2010). It is expected that REX will be able to accurately recreate the results provided through experimental methods and will offer a more cost-effective way of conducting these studies while providing a broader range of results. The validation study is to serve as an example of the effectiveness of the REX code and CFD methods in general.

III. REX

REX is a proprietary CFD solver being continually developed by Professor Pablo Carrica's research group at the University of Iowa. REX utilizes unsteady Reynold-Average Navier-Stokes (URANS) simulations to generate solutions and has capabilities for detached eddy simulation (DES) and delayed detached eddy simulation (DDES). For this validation study DES simulations were used to generate solutions and the results were compared accordingly. REX simulations are conducted using structured body-fitted grids. The overset grids use Suggar or Suggar++ to compute all necessary overlap and domain connectivity information. Parallel processing of solutions is handled through the use of MPI-based domain decomposition tools with each decomposed block solved using one processor. The information generated by each process is transferred to others as necessary after each subsequent alternate-direction-implicit (ADI) iteration and any boundary condition enforcement. The pressure matrices include all information for overset interpolation and prescribed boundary conditions (Carrica, 2015). All resulting linearized algebraic matrices are solved using an implemented PETSc toolkit (Carrica, 2015).

Simulation Process

I. Solution Strategy

The wind tunnel data presented by Forrest and Owen (2010) was modeled using a structured overset grid system and simulated using the CFD solver REX, as discussed previously. Flow velocities and turbulence intensities generated using REX at specific locations were used to quantitatively compare the results generated using CFD methods to the experimental wind tunnel results. The report contains data for both headwind and Green 45° conditions, both of which were simulated and compared. For the wind tunnel experiment conducted by the NRCC a 1:100 scale model of the SFS2 geometry was used. The simple frigate shapes consist of simplified geometry that was developed as an easily repeatable benchmark case specifically for validating CFD codes (Forrest & Owen, 2010). The NRCC used hot-film anemometry to investigate the flow field around the scale model of SFS2. All of the wind tunnel experiments were conducted in a 2m x 3m low-speed wind tunnel at the Aerodynamics Laboratory of the NRCC. Boundary layer suction was also used to ensure a uniform incident velocity profile (Forrest & Owen, 2010). The data provided in Forrest and Owen (2010) included mean velocity as well as turbulence statistics measured at various locations. The wind tunnel experiments were all conducted at Reynolds number of 6.58 x 10^5 in air, normalized by free stream velocity and ship beam length (Forrest & Owen, 2010). All non-dimensional variables in REX are obtained using a reference length and velocity. For all REX simulations, the reference length was the full length of the ship and the Reynolds number used was computed accordingly. The reference velocity used for this calculation was the free stream velocity in the wind tunnel which was approximately 60 m/s.

$$Re = \frac{\rho\, U_{ref} L_{ref}}{\mu} \tag{1}$$

Although the overall goal is to generate full scale data sets, for this comparison study the scale model of SFS2, used for experimental data collection, was also used for all CFD simulations for a more direct and accurate comparison of data. All solutions were iterated to reach a steady-state solution that accurately represented the behavior of the airwake in the measurement zone for this comparison study. A time step of $\Delta t^* = 2.0 \times 10^{-3}$ was used for all simulations. This time step is normalized by the freestream velocity of the airflow and the ship length. Each of the simulations were run for at least at time equal to the flow traversing a distance equal to 15 ship lengths based on the speed of the free stream velocity. This based on the idea that some of the airflow features are slow to fully develop and occur at a very small frequency. Solutions were averaged over different time periods and reported to account for any consistent transient behavior in the solutions. Post processing techniques similar to this are discussed further in the corresponding section.

II. Grid Generation

The geometry of the SFS2 model was obtained from Quon and Smith (2015). The geometry of the ship was first modeled in CREO using the dimensions shown in Fig. A1. The generated 3D model of the ship was exported as an IGES file and uploaded to Gridgen to be used as the base to generate the overset grid system. Creating a structured grid system for a geometry that of all square features proved to be a difficult and time consuming process. It is worth mentioning that the CFD results also reported in Forrest and Owen (2010) utilized an unstructured grid which is typically much less difficult to generate. Once the overall geometry was defined in Gridgen, it needed to be scaled to the 1:100 scale model used in the NRCC wind tunnel experiments. After scaling the model to the correct size, all dimensions were normalized by the ship's overall length. This is done for modeling and simulation purposes. The ship's outer geometry was meshed using four separate structured overset grid systems. The overset grid system allows the flow characteristics around the ship to be computed. The overset grid system utilized also consists of a large relatively coarse background grid used to define the full computation domain and impose the boundary conditions, and a refinement grid located over the ship's flight deck were all measurements were taken. All grid blocks used to resolve the flow near the surface of the ship have a wall spacing of $y^+ = 5E\text{-}5$ with an expansion ratio of 1.3. The four grids used to cover the ship's hull consist of a hull, top surface, bow cap, and superstructure block. The background volume was specified to cover a surrounding area 2.5 ship lengths long, 1.2 ship lengths wide, and 0.8 ship lengths high. The background grid allows the behavior of the flow around the ship to fully develop and be properly resolved. Based on the fact that all measurements will be compared along the same line shown in Fig A.3, a refinement grid was generated in this area to ensure all small turbulent flow features can be resolved properly. These six grid blocks make up the full system used for the REX simulations. The use of multiple grid blocks requires the coupling of solutions between blocks in the overlapping of regions. All the necessary overlap and domain connectivity information (DCI) was computed by the code Suggar++. The body-fitted volume grids were difficult to generate due to the square faces that make up the geometry of the SFS2 model. This issue is discussed by Forrest and Owen (2010) when it is mentioned that this type of geometry is better suited for an unstructured mesh as opposed to the structured mesh used for simulations run by REX.

III. Grid Study

The headwind case was simulated using two different grid systems as a means to test the grid dependency of the final solutions. The coarse grid system shown in Fig. A.2 consist of a total of $8.8x10^6$ points, including the background and refinement grids. The coarse grid was refined by a factor of two in each direction in order to produce a fine grid system with a total of $70x10^6$ points. A refinement factor of two was used to ensure that the overall geometry of the grid system was not affected during the refinement process. Ideally, a medium-sized grid would also have been compared but, due to time constraints, this was not feasible and only two different grid sizes were used for this grid study. By using a factor of two in each direction the refinement tool used placed a new point in between each of the existing points ensure that the location of any sharp corners were not changed or rounded. For the overall grid layout only the spacing between points was changed. The goal of the grid study was to prove that the solutions being generated through REX are grid independent and not affected by the grid spacing. The results of this study can be found in the results section of this report for headwind case.

IV. Post Processing

For the comparison study, all measurements were taken along a line located at 50% of the deck length and hangar height which spans a length equal to two beam widths across the flight deck. The location of the line for which the data is reported over can be found in figure A.3. This location was used to represent the region closest to where a helicopter would hover before landing on the deck (Forrest & Owen, 2010). The data used for comparison was the wind tunnel results of mean velocity and turbulence intensity. Time histories of the forces imposed on the ship from the airflow were analyzed in order to measure the period over which different flow behaviors changed. Since Forrest and Owen (2010) did not report any information about how solutions were averaged or adjusted for transient behavior, a trial and error method was implemented to see how averaging over different periods of time affected the level of accuracy of the results produced by REX in comparison to the experimental results. It can be seen in the results section of this report that varying the number of solutions averaged and the time period over which they are averaged did result in smoother flow behavior but did not significantly change the behavior at any measured point. The directional turbulence intensities were resolved by calculating the root mean squares (RMS) of the velocity components for the same periods of time steps used for calculating the velocity averages. The resultant average turbulence intensity at each point was calculated by using

$$x_{rms} = \sqrt{\frac{1}{n}(x_1^2 + x_2^2 + \cdots + x_n^2)} \tag{2}$$

Equation 2 was used to calculate the turbulence intensity in the u, v, and w directions. The exact locations of measurements used during the wind tunnel experiments were replicated by using a linear interpolation zone within TecPlot. Line graphs were generated for each of the different cases and results for direct comparison to the experimental results.

Results & Discussion

I. Headwind Case

During the wind tunnel experiment the velocity and turbulence intensity were measured and calculated at 11 different locations across the line shown in figure A.3 in the appendices. Due to the capabilities of CFD the solutions could be resolved over the entire line of measurement. The values of the NRCC wind tunnel experiments were plotted on the same line plots as the CFD results for direct comparison. It can be seen in Fig. B.1 and Fig. B.2 that both CFD cases show a velocity magnitude at each location that is lower than that of the experimental results. This behavior is especially prominent near the center of the lateral line across the ship's flight deck. The fine grid results do seem to more accurately resolve the non-symmetric behavior of the flow velocity magnitude that is seen in the experimental results. Both grid solutions do very closely show the same trend seen in the experimental results for the velocity magnitude across the ship's flight deck. The velocity components were also plotted and compared in order to better understand the source of any differences seen in the flow velocities between the CFD results and experimental results and develop a deeper understanding of how the flow behavior changes across the flight deck. It can be seen in Fig. B.3 and Fig. B.4 the CFD simulations for both the fine and coarse grid were able to very accurately resolve the flow velocity in the w-direction. The plots show that near the center line there is noticeable downdraft introduced into the flow field that is not seen near the edge of the lateral line. It can also be seen that the velocity in the u-direction for the CFD simulations is significantly lower than that measured during wind tunnel experiment. The largest differences in velocity are seen in the u and v directions. The CFD results again accurately follow the same trends shown in the experimental results shown in the experimental results but do report lower velocities in certain locations. Turbulence intensity is used to measure the fluctuations in the flow. Unsteady flow is represented by higher turbulence intensities and steady undistributed flow is represented by lower turbulence intensities. The turbulence intensity was plotted in Owen and Forrest (2010) as both a point resultant and a directional component. It can be seen in Fig. B.5 and Fig. B.6 that both the fine and coarse grids were able to very accurately resolve the behavior of the turbulence and intensity along the ships flight deck. Differences are seen in the areas where turbulence intensity begins to increase significantly. Although the CFD model does not match the experimental values directly, it can be seen that the computational model is able to predict large increases in turbulence with good spatial accuracy. The individual turbulence components for both the CFD fine and coarse grids, compared to the experimental values, can be found in Fig. B.7 and Fig. B.8. These figures show that overall agreement in turbulence intensity for the headwind case is good. The fine grid was very accurate in resolving the turbulence intensity in the lateral and vertical directions but did calculate a turbulence intensity in the longitudinal (v) direction to be significantly lower than that of the wind tunnel results near the center of the measurement line. The coarse grid was able to more accurately resolve the turbulence components for the u direction, in comparison to the fine grid, but did produce large results in the v direction. Overall, after plotting the directional turbulence intensities for the coarse and fine grid and them comparing them to the experimental data, it is shown that the computational results were able to accurately match the trends seen in the wind tunnel data with some areas of magnitude discrepancies.

II. Green 45° Case

The airflow velocity behavior for the Green 45° case is significantly different than that of the headwind case. Much larger flow velocities are measured in the w direction in comparison to the headwind case as shown in figure C.1. Also, the wind speeds do not share the same asymmetric behavior seen in the headwind case along the line across the flight deck. All wind speed data from the wind tunnel experiments are measured with respect to the direction of the free stream velocity. This is due to the fact that the actual wind direction was not changed for these experiments but the ship's orientation was adjusted accordingly (Forrest & Owen, 2010). The CFD data is reported in this same manner for direct comparison. Figure C.1 shows a direct comparison between the directional flow velocities predicted using CFD and those measured by the NRCC. As seen in the headwind case, CFD was able to able to generate the same trend seen in the experimental data and capture major flow features but was unable to directly reproduce the same wind speeds measured in the wind tunnel. The CFD solutions shown in figure C.1 are averaged over the time steps 9000-15000. This averaging is based on a frequency behavior observed on the force imposed on the ship in the y-direction, this time represent three periods of this behavior (2000 time steps). This averaging technique was used in order to account for any transient behavior in the flow field, which appeared to be much more prevalent in this case as opposed to the headwind case. The largest difference in wind speed data can be seen at the port side of the ship in the u direction. The wind speed predicted using CFD is almost 50% lower than that measured experimentally. Part of this is due to the fact that the flow is approaching the ship from a slightly different angle than 45° for the wind tunnel experiments, this produces higher u velocities as well as lower v directional velocities (Forrest & Owen, 2010). It is also believed that this flow angle discrepancy contributed significantly to the amount of flow velocity introduced in the w direction, this is shown by the CFD result predicting a much a higher velocity in this direction on the starboard side of the ship's deck.

The turbulence intensity components were also measured, using the same time steps used previously, by calculating the RMS of the individual velocity components. For easier comparison to the experimental data, the components of turbulence were plotted separately and can be found in figures C.2, C.3, and C.4. The computational data generated for the turbulence intensity in u-direction was very similar to that from the wind tunnel data. The turbulence intensity is relatively high on the port side of the ship and then decreases drastically on the starboard side of the ship to practically zero, signaling that the location is in the free stream flow and away from any disturbed flow generated by the ship. This same trend is seen in the v and w turbulence component plots. This trend shows the drastic effect the ship's body has on the level of turbulence found in the flow around the ship. The computational simulations did produce turbulence intensities in the v and -directions that follow the trends seen in the experimental data but did produce turbulence magnitudes that were different. From these figures, it can also be seen that the maximum turbulence intensity measured for the Green 45° was significantly higher (≈ 40%) than that seen in the headwind wind case. This is believed to be due to significantly more mixing of the airflow over the flight deck when the angle of attack is changed from the head wind case. This is important information to note when trying to quantify how the ship's operational conditions affect the behavior of the airflow over its flight deck.

III. Discussion

The use of CFD simulations for full scale simulation offers a more accessible method for generating operational data sets in comparison to large full scale experimentation. Understanding the airflow fluctuations behind a ship that launches and recovers aircraft can provide important information for pilot training as well as help ship builders design features with this information in mind. At-sea trials have been done in order to quantify the behavior of the airflow but are very expensive and provide limited amounts of data. Scale model experiments have also been done but do present a certain degree of bias error from scaling effects. To ensure that computational models can accurately resolve a ship's aerodynamic behavior under prescribed motions the CFD models need to be compared to existing experimental data sets. This preliminary comparison study does not include the simulation of ship motions based on the experimental data sets readily available. Even without motions, this static wind tunnel comparison can still be used to confirm if the CFD code REX is capable of accurately resolving the airflow behavior around a structure during varying conditions. The by Forrest and Owen (2010) is limited and does not mention any type of uncertainty analysis in the data sets, though it mentions possible sources of error during wind tunnel experimentation, such as a discrepancy in the angle of flow for the Green 45°, this is where the largest difference between the computational and experimental measurements are seen for the mean flow velocities. The difference in flow direction could have had a large effect on the magnitude of flow velocity seen in each direction. For a stronger comparison, more detailed experimental information would be needed to ensure that the computational simulations are directly comparable to the experimental method being used. This study serves as a strong baseline understanding of the capabilities of REX and offers direction on how to setup and post-process futures simulations.

Conclusion

I. Study Conclusions

The overall objective of this comparison study was to verify that the CFD code REX can for prediction of the characteristics of a ship’s airwake under varying conditions. To test this, wind tunnel experiments conducted by the NRCC, using the simple frigate geometry (SFS2), were simulated using a structured grid system of the geometry and REX. Two different flow conditions were tested, a headwind case and a Green 45° case (starboard side at 45°). Using the results discussed previously it can be concluded that REX is capable of modeling the flow behavior in the flight deck area of SFS2 under varying conditions. The results for the flow velocity and turbulence from REX accurately match the same trends seen in the wind tunnel experimental data for both flow conditions. There were inaccuracies when it came to predicting the same magnitude for wind speed and turbulence intensity for some of the directional components and conditions. These inaccuracies were especially prominent in the area directly behind the ships superstructure for the headwind case and on the port side of the ship for the Green 45° case. These differences coincide with the areas of highest turbulence intensity which typically have very significant time-dependent flow features. Without detailed knowledge of the period over which experimental data was collected, it is very difficult to recreate time-dependent results. For the headwind case a grid study was also conducted in order to ensure a grid-independent solution head be achieved. Based on the similarity between results produced by the fine grid and coarse grid it is confirmed that the solutions generated by REX are grid independent. It is noteworthy that the fine grid was able to resolve smaller flow feature not seen in the coarse grid especially in the areas of high turbulence. This aspect should be taken into consideration when designing a grid system for future simulation and deciding the target level of detail in the solution to be achieved.

II. Acknowledgements

I would like to thank Professor Pablo Carrica for allowing me to explore this topic during my time as an undergraduate research assistant at IIHR – Hydroscience & Engineering, University of Iowa. Also, thank you to Juan E. Martin, IIHR Research Engineer, for all his support and recommendations throughout the entire length of this study. Without his help the completion of this project would not have been possible.

References

Carrica, P. (2015). REX User's Manual. 002. Retrieved from http://www.iihr.uiowa.edu/pcrg/rex

Forrest, J., & Owen, I. (2010). An investigation of ship airwakes using Detached-Eddy Simulation. *Computers & Fluids*, *39*, 656-673.

Quon, E., & Smith, M. (2015). Advanced data transfer strategies for overset computational methods. *Computer & Fluids*, *117*, 88-102.

Zhe-Ming, G., & Chang-Meng, L. (2016). *Airwake characteristics simulations with different turbulence models*. Paper presented at the International Conference on Advanced Materials and Computer Science (ICAMCS 2016).

Appendices

A. Simulation Setup

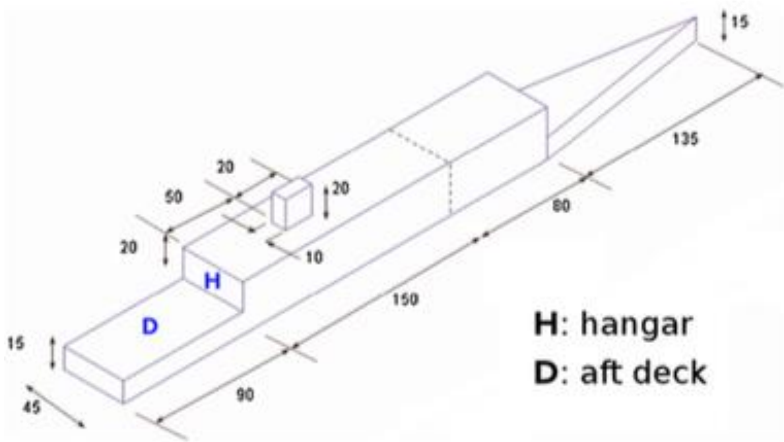

Figure A.1: SFS2 full scale dimension in ft. (Quon & Smith, 2015).

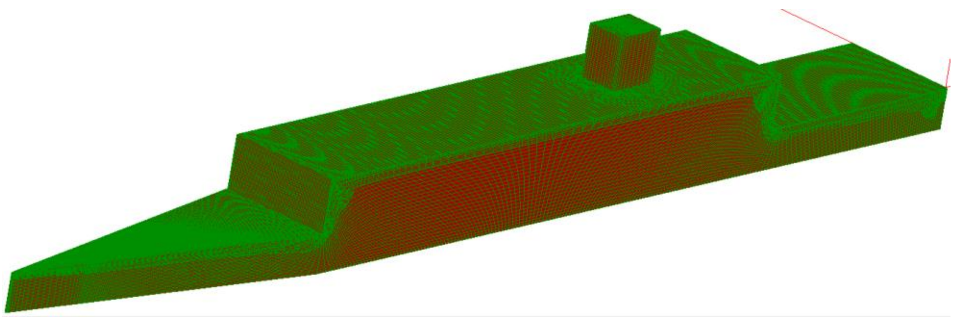

Figure A.2: Overset grid system used to model the surface of SFS2 geometry.

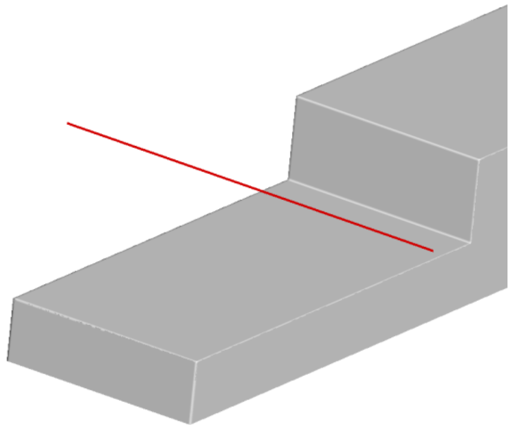

Figure A.3: Line over which all measurements were taken and used for comparison.

B. Headwind Case Results

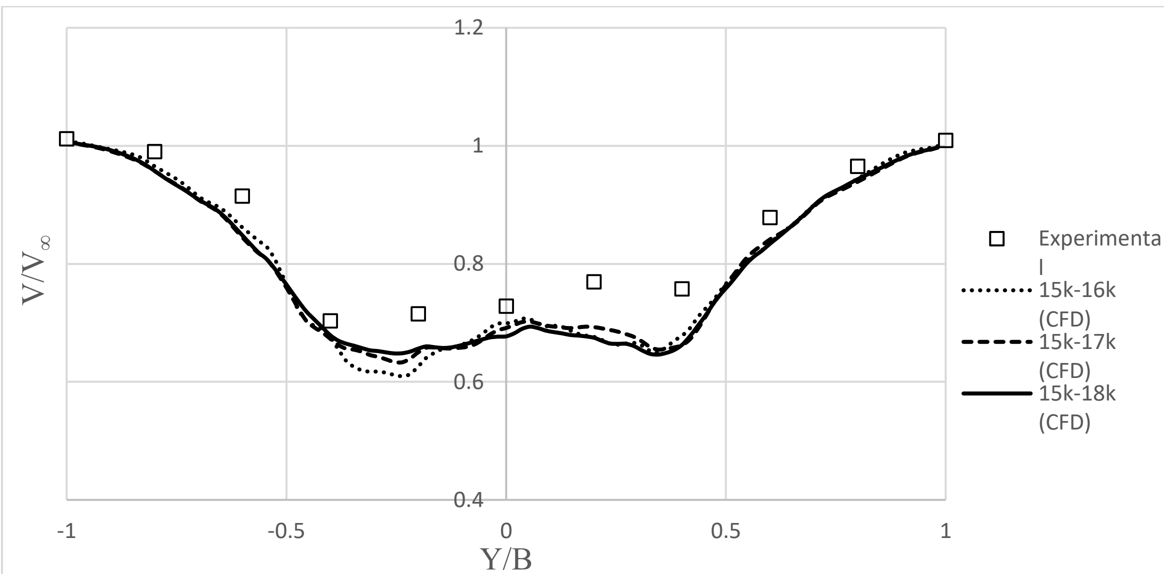

Figure B.1: Comparison of velocity magnitude on lateral line over the flight deck for coarse grid averaged over three different groups of time steps. Velocity and lateral position are normalized by the free stream velocity and the ship's beam, respectively.

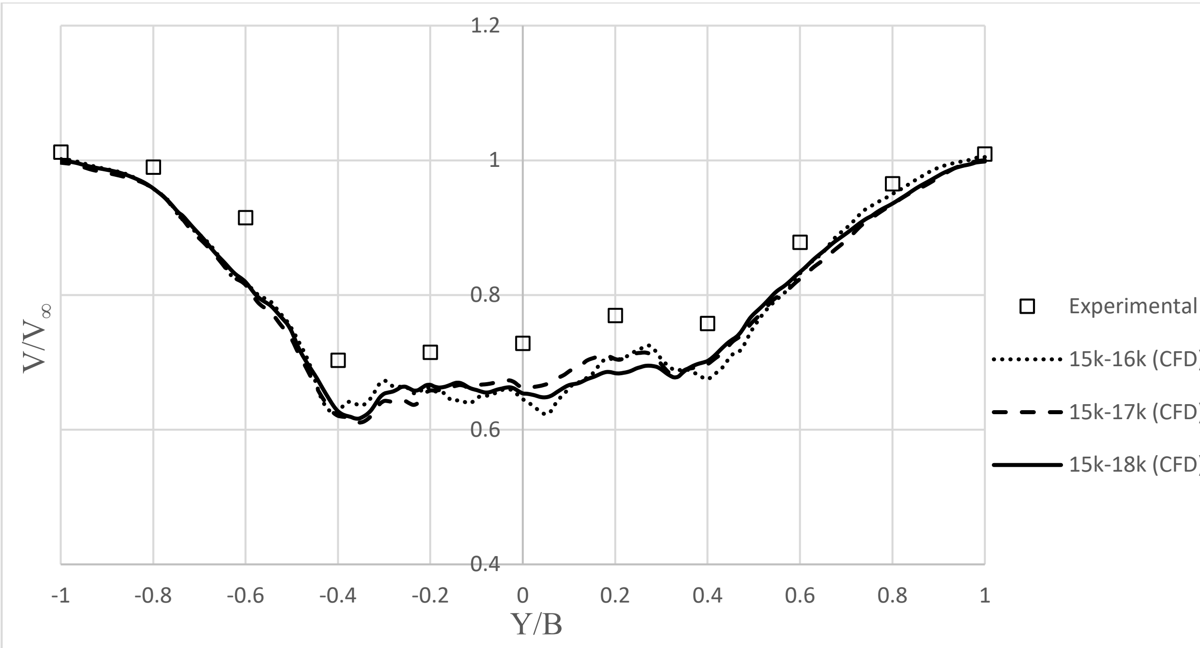

Figure B.2: Comparison of velocity magnitude on lateral line over the flight deck for coarse grid averaged over three different groups of time steps. Velocity and lateral positions are normalized by the freestream velocity and the ship's beam, respectively.

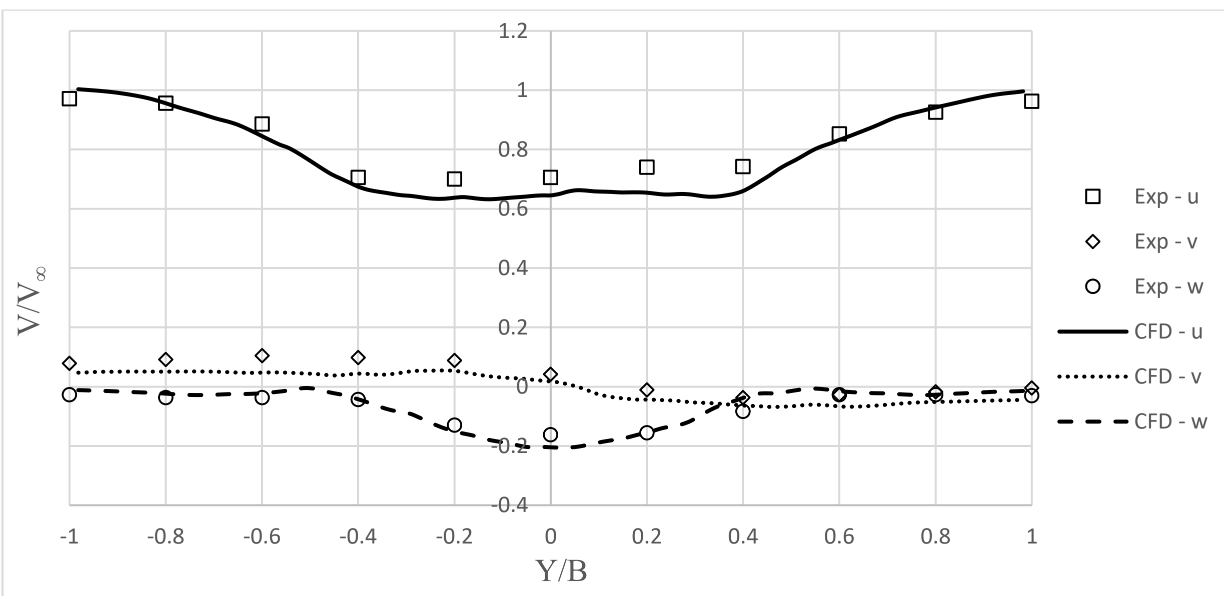

Figure B.3: Comparison of velocity components on lateral line over the flight deck for the coarse grid. CFD velocities were averaged from 15000-18000 Δt^*. Velocity and position are normalized by the freestream velocity and the ships beam, respectively.

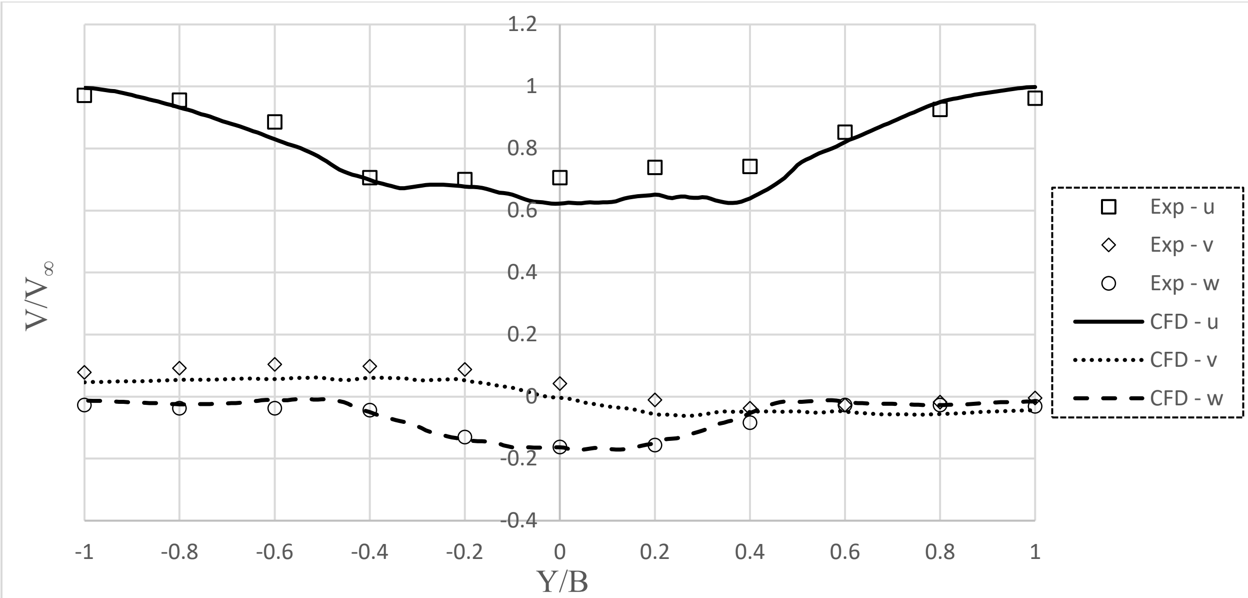

Figure B.4: Comparison of velocity components on lateral line over the flight deck for the fine grid. CFD velocities were averaged from 15000-18000 Δt^*. Velocity and position are normalized by the freestream velocity and the ship's beam, respectively.

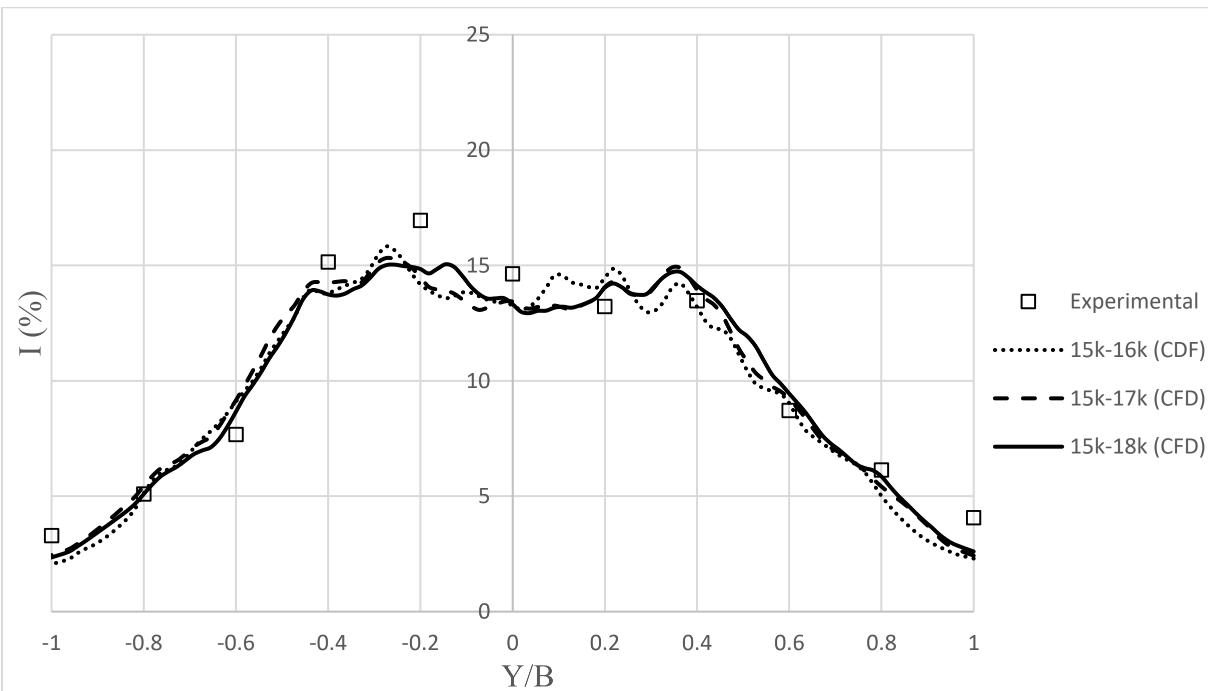

Figure B.5: Comparison of average turbulence intensity on lateral line over the flight deck for the coarse grid calculated over three different groups of time steps. Intensity and lateral position are normalized by the free stream velocity and the ship's beam, respectively.

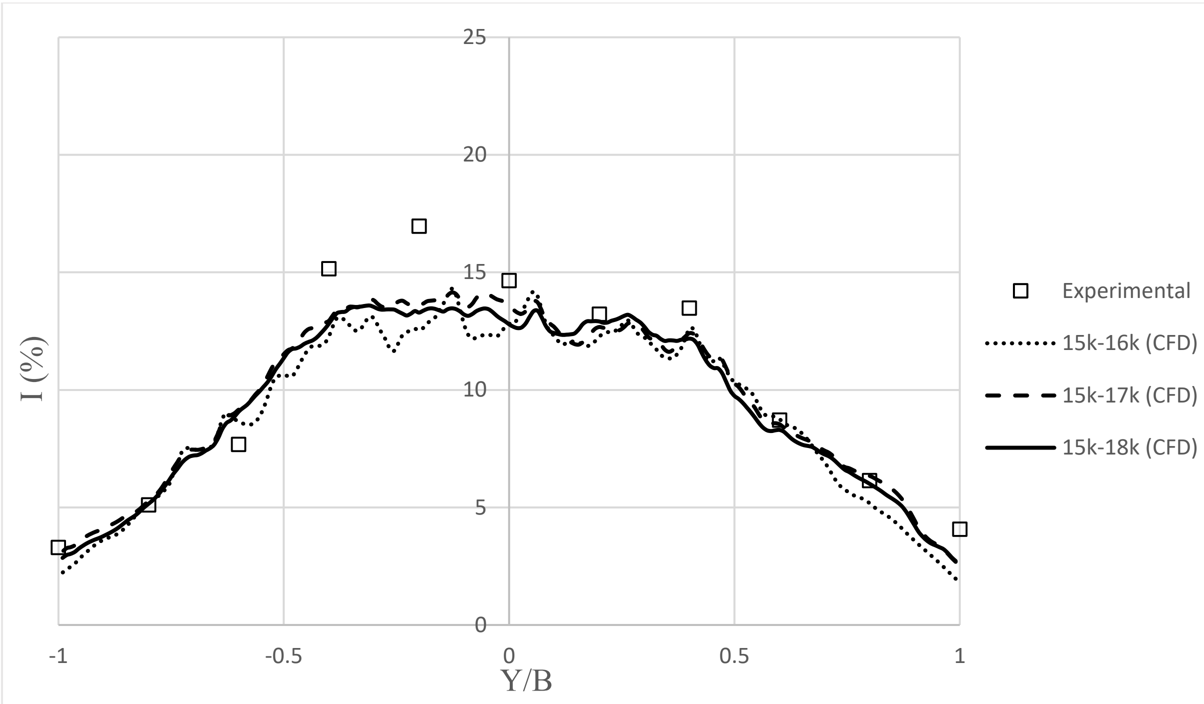

Figure B.6: Comparison of average turbulence intensity on lateral line over the flight deck for the fine grid calculated over three different groups of time steps. Turbulence intensity and lateral position are normalized by free stream velocity and the ship's beam, respectively.

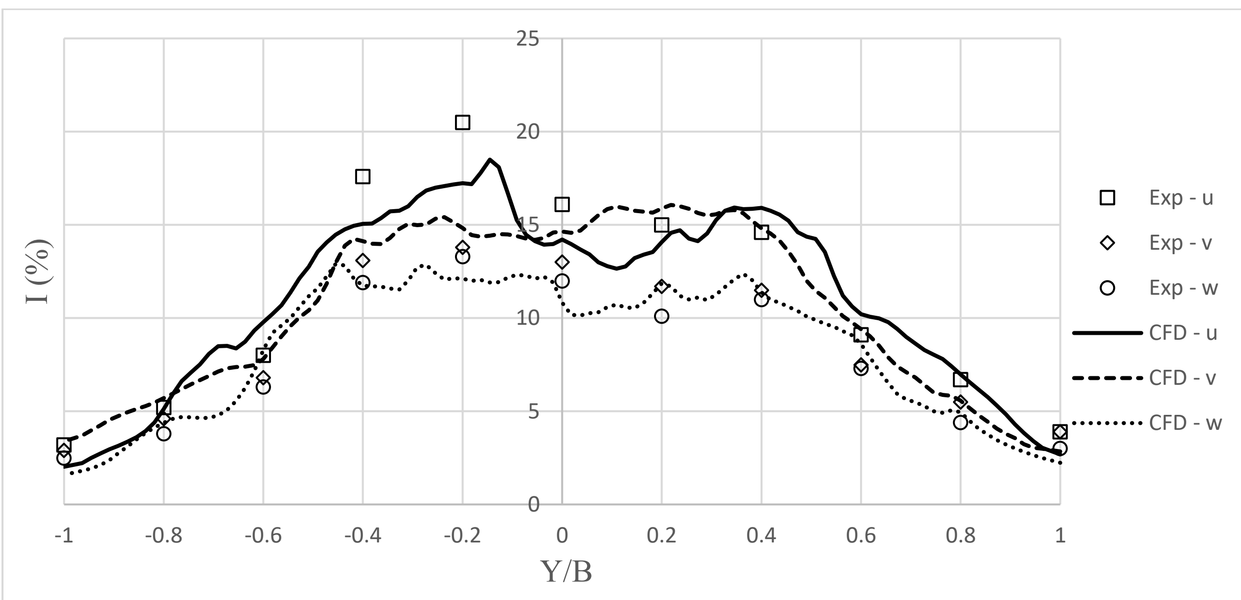

Figure B.7: Comparison of turbulence components on lateral line over the flight deck for the coarse grid. CFD RMS values were calculated from 15000-18000 Δt*. Turbulence intensity and lateral position are normalized by free stream velocity and the ship's beam, respectively.

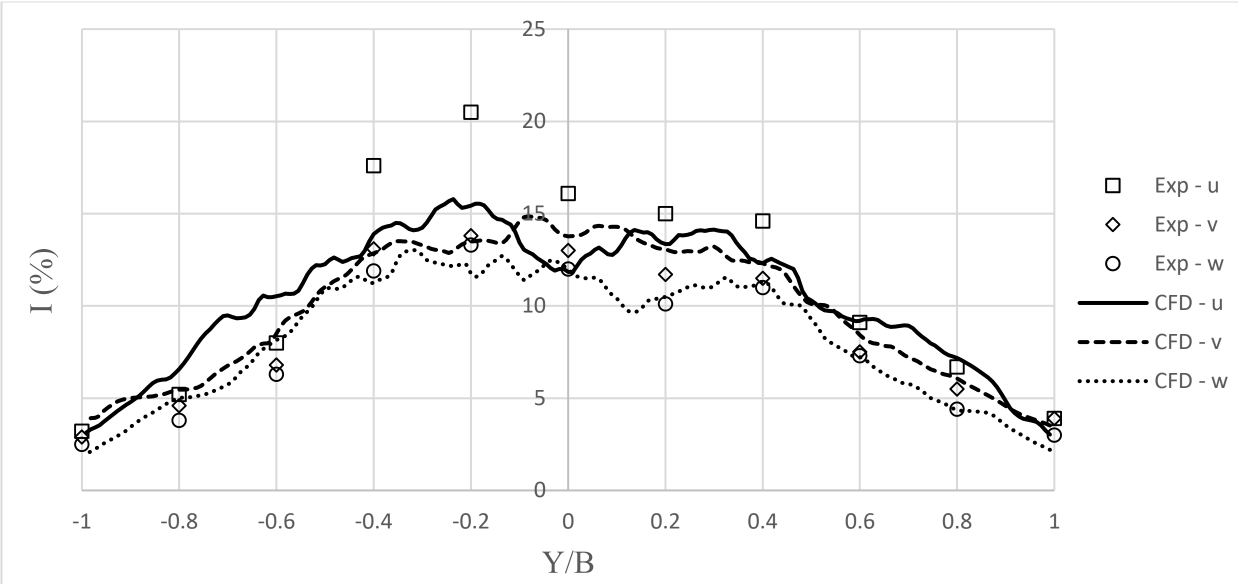

Figure B.8: Comparison of turbulence intensity components on lateral line over the flight deck for the fine grid. CFD RMS values were calculated from 15000-18000 Δt*. Turbulence intensity and lateral position are normalized by free stream velocity and the ship's beam, respectively.

C. Green 45° Case Results

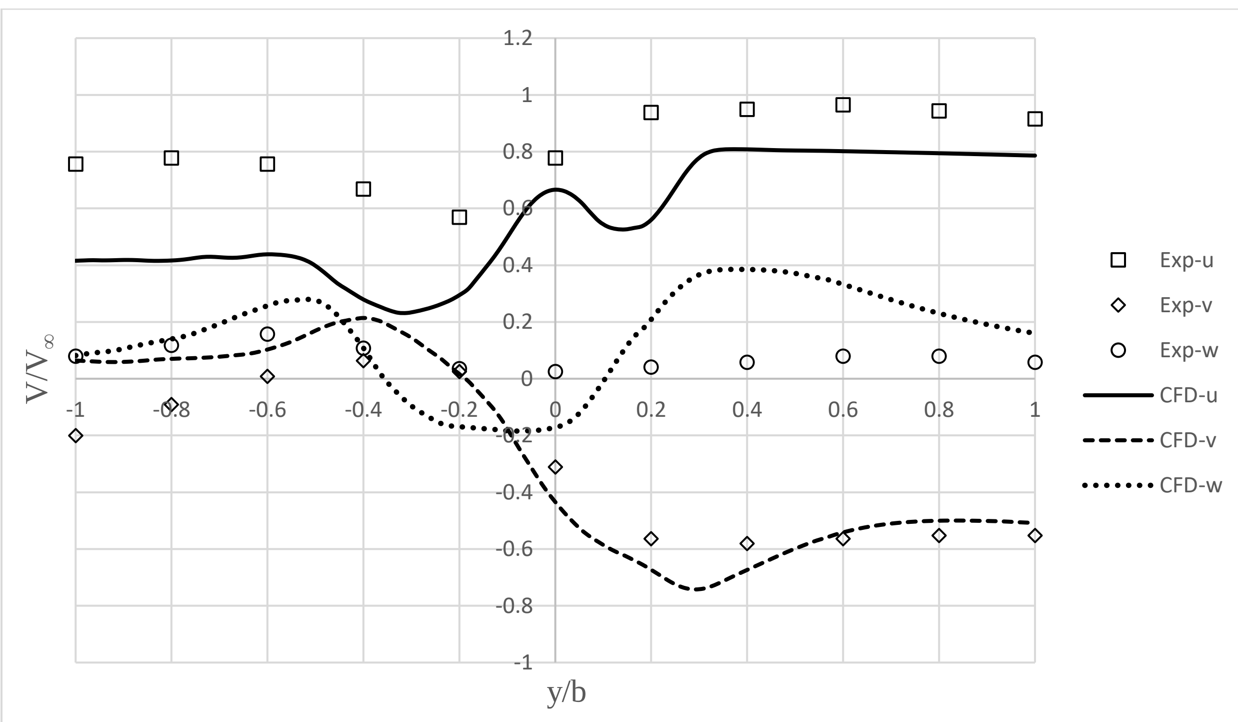

Figure C.1: Comparison of velocity components on lateral line over the flight deck. CFD velocities were averaged from 9000-15000 Δt*. Velocity and position are normalized by the freestream velocity and the ships beam, respectively.

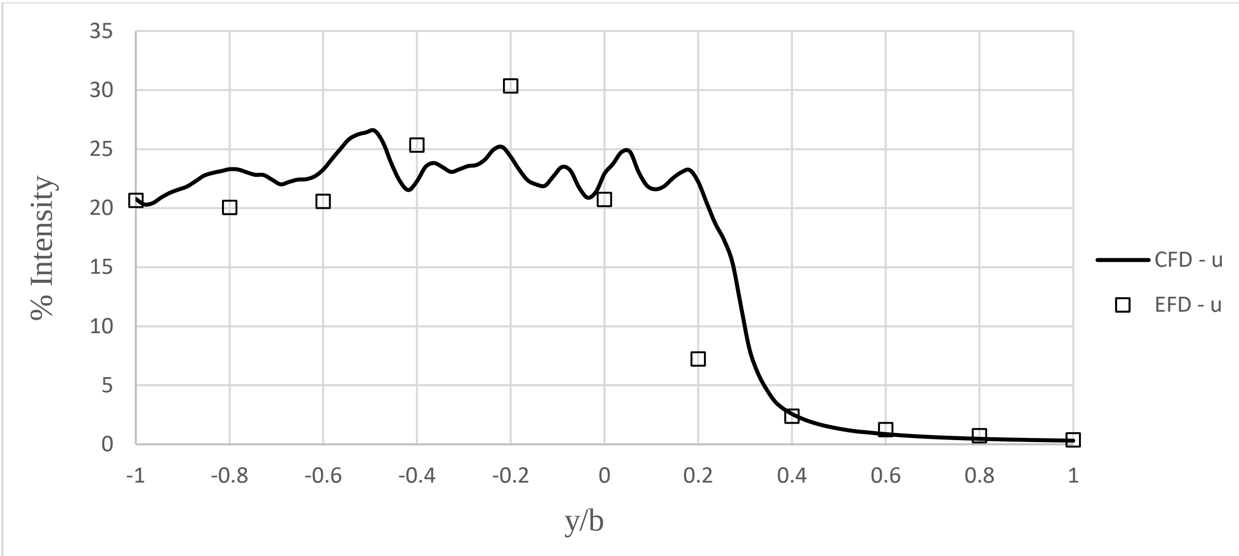

Figure C.2: Comparison of turbulence intensity in the u-direction on lateral line over the flight deck. CFD turbulence intensities were calculated from 9000-15000 Δt*. Turbulence intensity and lateral position are normalized by free stream velocity and the ship's beam, respectively.

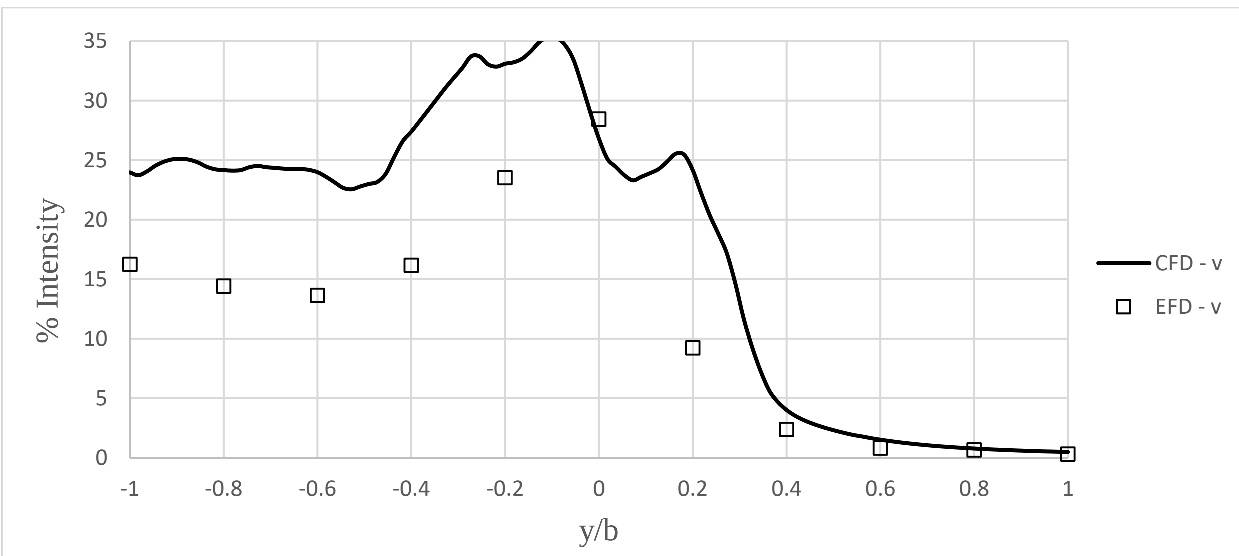

Figure C.3: Comparison of turbulence intensity in the v-direction on lateral line over the flight deck. CFD turbulence intensities were calculated from 9000-15000 Δt*. Turbulence intensity and lateral position are normalized by free stream velocity and the ship's beam, respectively.

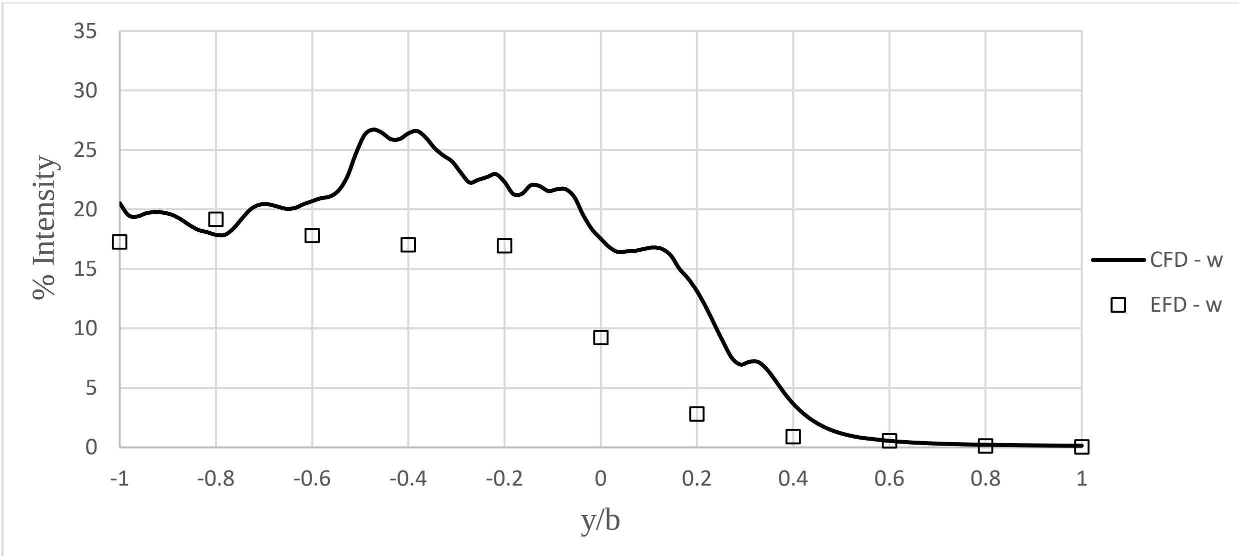

Figure C.4: Comparison of turbulence intensity in the w-direction on lateral line over the flight deck. CFD turbulence intensities were calculated from 9000-15000 Δt^*. Turbulence intensity and lateral position are normalized by free stream velocity and the ship's beam, respectively.