REPUBLIC OF TURKEY

YILDIZ TECHNICAL UNIVERSITY

GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

A NUMERICAL INVESTIGATION OF FLOW AROUND SUPERSTRUCTURE OF A SURFACE COMBATANT

Sinem ÖKSÜZ

MASTER THESIS

Department of Naval Architecture and Marine Engineering

Naval Architecture and Marine Engineering Program

Advisor

Asst. Prof. Ali DOĞRUL

February, 2020

REPUBLIC OF TURKEY

YILDIZ TECHNICAL UNIVERSITY

GRADUATE SCHOOL OF NATURAL AND APPLIED SCIENCES

A NUMERICAL INVESTIGATION OF FLOW AROUND SUPERSTRUCTURE OF A SURFACE COMBATANT

A thesis submitted by Sinem ÖKSÜZ in partial fulfillment of the requirements for the degree of MASTER OF NAVAL ARCHITECTURE AND MARINE ENGINEERING is approved by the committee on 20.02.2020 in Department of Naval Architecture and Marine Engineering, Naval Architecture and Marine Engineering Program.

Asst. Prof. Ali DOĞRUL

Yıldız Technical University

Advisor

Approved By the Examining Committee

Asst. Prof. Ali DOĞRUL, Advisor
Yıldız Technical University ______________________

Assoc. Prof. Seyfettin BAYRAKTAR, Member
Yıldız Technical University ______________________

Assoc. Prof. Ömer Kemal KINACI, Member
İstanbul Technical University ______________________

I hereby declare that I have obtained the required legal permissions during data collection and exploitation procedures, that I have made the in-text citations and cited the references properly, that I haven't falsified and/or fabricated research data and results of the study and that I have abided by the principles of the scientific research and ethics during my Thesis Study under the title of Location Analysis of The Emergency Service Centers Of a Case Company supervised by my supervisor, Asst. Prof. Ali DOĞRUL. In the case of a discovery of false statement, I am to acknowledge any legal consequence.

Sinem ÖKSÜZ

Signature

ACKNOWLEDGEMENTS

The ship air wake around a frigate is investigated in this study. Following by verification and validation study, it is carried out a parametric study that analyses geometry effects on the airflow region.

I would like to thank my advisor Asst. Prof. Ali DOĞRUL who consults and shares his acknowledgments and experience with me.

I would like to express my deepest appreciation to Dr. Yiğit Kemal DEMİREL who is my supervisor at the University of Strathclyde for all his help and support in Glasgow.

I would like to thank especially my friend Y. Kaan İLTER for his help and sharing his technical experience with me during the analyzing process in Glasgow.

Moreover, I would like to thank Prof. Fahri ÇELİK and Assoc. Prof. Seyfettin BAYRAKTAR for their support during my MSc life at Yıldız Technical University. I am really grateful to them for their valuable ideas.

Also, many thanks to Asst. Prof. Ferdi ÇAKICI and Res. Asst. Emre KAHRAMANOĞLU for their patience and kindness during the analyses process.

I also would like to thank Res. Asst. Soonseok SONG for his help during the simulation analysis process.

I would like to thank the University of Strathclyde for the provision of the ARCHIE-WeSt high-performance computing facilities.

Lastly, I would like to express my deepest appreciation to my family, my friends, and my uncle Cemal ÖKSÜZ for their endless support. I am grateful to them for their support and belief during my education life.

Sinem ÖKSÜZ

TABLE OF CONTENTS

LIST OF SYMBOLS

ρ	Density
U_{mean}	Headwind speed
P	Pressure
h1,2,3	Grid lenghts
r	Grid refinement factor
N	Mesh cell number
ε	Difference between grid numbers
ϕ1,2,3	Desired value
R	Grid convergence condition
μ	Dynamic viscousity
h	Hangar height
Re	Reynolds number
Fn	Froud number
δ	Boundary layer thickness
u	Velocity at x-direction
v	Velocity at y-direction
ν_T	Turbulence eddy viscosity
ω,ε	Specific rate of dissipation
σ,β	Closure coefficients

LIST OF ABBREVIATIONS

CFD	Computational Fluid Dynamics
DES	Detached Eddy Simulation
LHA	Landing Helicopter Assault
LNG	Liquified Natural Gas
MILGEM	Milli Gemi (National Ship)
RANS	Reynolds Average Navier Stokes
SFS	Simple Frigate Shape
SFS2	Simple Frigate Shape 2
SHOL	Safe Helicopter Operation Limits
SRF	Shortened Research Frigate
TTCP	The Technical Cooperation Team

LIST OF FIGURES

LIST OF TABLES

ABSTRACT

A Numerical Investigation Of Flow Around Superstructure Of A Surface Combatant

Sinem ÖKSÜZ

Department of Naval Arhitecture and Marine Engineering

Master Thesis

Advisor: Asst. Prof. Ali DOĞRUL

In this study, it is purposed to modify ship air wake to reduce instabilities airflow region in frigates which have helidecks. "What is the effect of hangar height to ship air wake?" and "Is it possible to get better aero dynamical flight region over the hangar deck by adding chamfer geometry on the hangar decks?" questions were searched. Two different hangar level and two chamfer shape were investigated to contribute scientific researches on the naval area.

The main step is to determine the air wake character firstly. To execute this, computational fluid dynamics method was used. The open-source geometry simple frigate shape 2 (SFS2) was chosen to perform the parametric study. Firstly, a previous literature study was chosen to validate results by comparing velocities. The different mesh configurations and turbulence models were tried to get closer results. Star CCM+ commercial software was used to model geometry and domain. The Reynolds Averaged Navier Stokes method was used due to time and computer processor limits. The uncertainty analyses were made to determine appropriate mesh configuration. After getting very close velocity gradients, the four new geometries were constituted as two different hangar height and two different hangar edge angles. It was seen that the airflow over the deck changed dramatically by changing the hangar height and adding chamfer. The velocity gradients and vorticity fields were compared with original SFS2 geometry results. It is concluded that to constitute better aerodynamically frigate design is possible by changing hangar design. It is purposed to ensure safer helicopter landing and take-off operations, reduction in fuel consumptions, and save time for transportation and critical operations such as search and rescue.

Keywords: Ship aerodynamics, CFD, hangar deck, frigate

ÖZET

Askeri Bir Geminin Üst Binası Etrafındaki Akışın Sayısal Olarak İncelenmesi

Sinem ÖKSÜZ

Gemi İnşaatı ve Gemi Makineleri Mühendisliği Bölümü

Yüksek Lisans Tezi

Danışman: Dr. Öğr. Üyesi Ali DOĞRUL

Bu çalışmada helikopter güvertesine sahip fırkateynlerdeki hava akımı bölgesinin dengesizliklerini azaltmak için gemi üstbinası etrafındaki hava akımı bölgesinin modifiye edilmesi amaçlanmıştır. "Hangar yüksekliğinin gemi hava izine etkisi nedir?" ve "Hangar güvertelerine pah geometrisi ekleyerek hangar güvertesi üzerinde daha iyi aerodinamik uçuş bölgesi elde etmek mümkün müdür?" sorularıcevaplanmaya çalışıldı. Denizcilik alanında bilimsel araştırmalara katkıda bulunmak için iki farklı hangar seviyesi ve iki pah şekli araştırılmıştır.

Ana adım, öncelikle gemi hava izi karakterini belirlemektir. Bunu gerçekleştirmek için hesaplamalı akışkanlar dinamiği yöntemi kullanılmıştır. Parametrik çalışmayı gerçekleştirmek için açık kaynaklı geometri basit fırkateyn şekli 2 (SFS2) seçildi. İlk olarak, hızları karşılaştırarak sonuçları doğrulamak için bir literatür çalışması seçildi. Farklı mesh konfigürasyonları ve türbülans modelleri kullanılarak daha yakın sonuçlar elde etmeye çalışıldı. Geometri ve etki alanını modellemek için Star CCM + ticari yazılımı kullanıldı. Zaman ve bilgisayar işlemcisi sınırları nedeniyle Reynolds Ortalama Navier Stokes yöntemi kullanıldı. Belirsizlik analizleri uygun ağ konfigürasyonunu belirlemek için yapılmıştır. Literatür değerlerine en yakın hız gradyanları elde edildikten sonra, dört yeni geometri; iki farklı hangar yüksekliği ve iki farklı hangar kenarı açısı olacak şekilde oluşturuldu. Hangar yüksekliğini değiştirip pah ekleyerek güverte üzerindeki hava akışının büyük ölçüde değiştiği görüldü. Hız gradyanları ve girdap alanları orijinal SFS2 geometri sonuçları ile karşılaştırıldı. Hangar tasarımını değiştirerek daha iyi aerodinamik özelliklere sahip fırkateyn tasarımı oluşturmanın mümkün olduğu sonucuna varıldı. Daha güvenli helikopter iniş ve kalkış operasyonları, yakıt tüketiminde azalma ve arama ile kurtarma ve kritik kurtarma operasyonları için zamandan tasarruf sağlamak hedeflenmiştir.

Anahtar Kelimeler: Gemi aerodinamiği, HAD, hangar güvertesi, fırkateyn

1
INTRODUCTION

Frigates are various sized warships that are used in military operations such as search and rescue, combat, helicopter operations, etc. These ships have a significant role in naval forces. It is thought the frigate name came from originally Italian; they designed in 16th century as speed and light ships to ensure maneuverability. First frigate models were seen in Eighty Years' War between Netherlands countries [1]. The French frigate ship with gun equipment can be seen in the Figure 1.1.

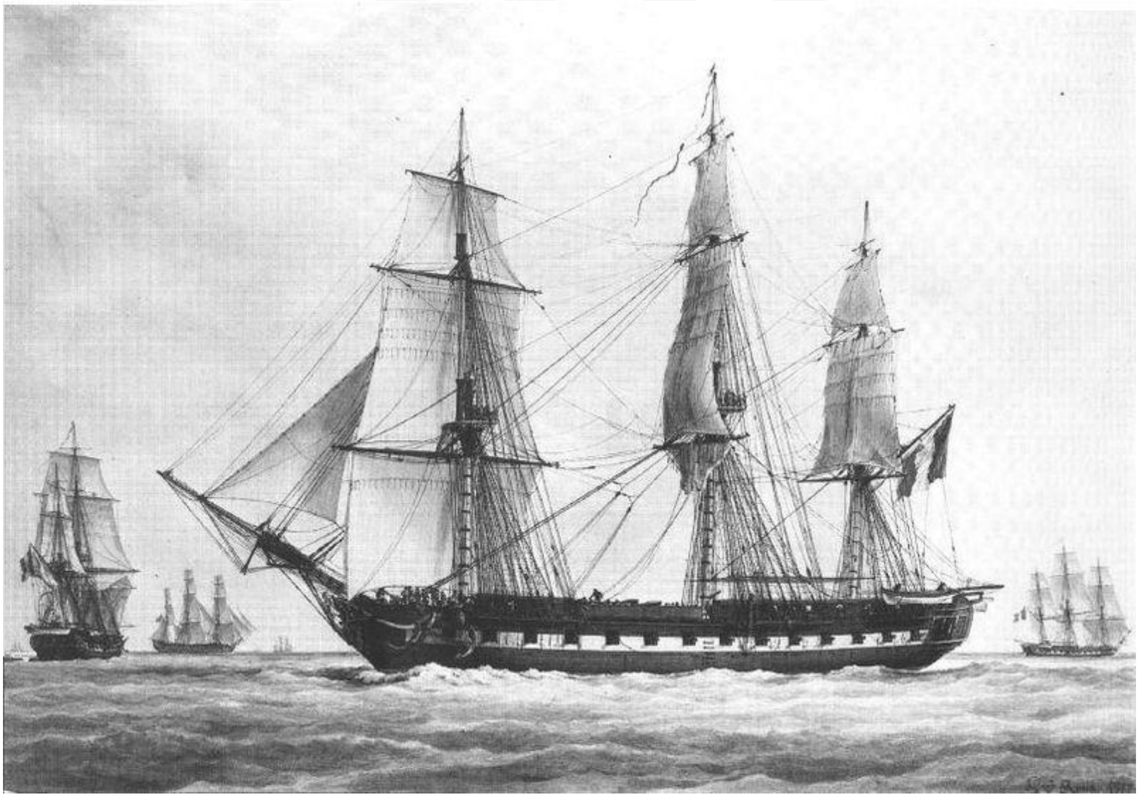


Figure 1.1 French frigate Penelope from 1806 [1]

The first examples of these ships have sails, by developing technology they evolved into modern frigate designs.

In Turkey, MILGEM project purposes to ensure national ship designs by using national sources [2]. The frigates are classified as three different tonnages which are named Gabya, Barbaros and Yavuz class [3]. These classes are consisting of 16 frigates; moreover, all G-class ships are equipped with integrated command control systems. Barbaros and G-class type frigates are developed with guided missile and radar systems. The one of Barbaros class frigate is seen in Figure 1.2.

Figure 1.2 Barbaros class frigate TCG Oruç Reis [4]

These ships have flight decks which ensure to do multipurpose operations such as search and rescue operations, using special radar systems to detect submarine targets, and also can carry missiles and bombs. The airflow region over the flight deck is called ship air wake. The interaction between ship air wake and helicopter is called a dynamic interface. It is critical to constitute aerodynamically stable hangar decks in frigates. To ensure that, The Technical Cooperation Team (TTCP) has been established in 1957. This researcher group study about ship-helicopter dynamic interface. Due to commercial or military purposes, it is hard to share actual ship dimensions and analyze results. With this purpose, simple frigate shape constituted which is a basically bluff body with hangar deck. Then the bow part added to this geometry and it is called a simple frigate shape 2. TTCP is a program that investigates the dynamic interface between ship and helicopter to optimize ship superstructure aerodynamically and to provide helicopter operations safer. To ensure safer flight operation over the deck, SHOL (safe helicopter operation limits) are defined [5]. The graph presented in Figure 1.3 shows the safe flight limits for helicopters according to the wind direction and angles. Red means the port direction of the ship, and green means starboard side of the ship. With using red and green definitions, the angle values represent the wind direction.

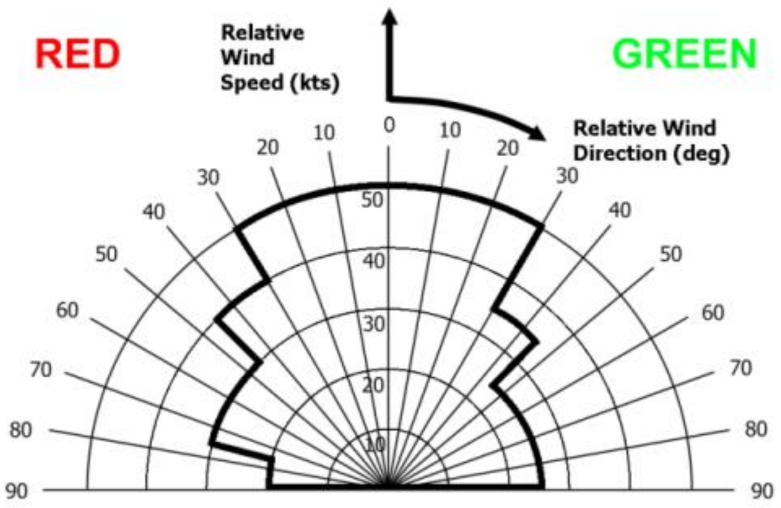


Figure 1.3 Ship helicopter operating limits (SHOL) [6]

The ship air wake unsteadiness causes the negative effects on the helicopter rotors which increase the fuel consumption and decrease the pilot control during landing operations. In the past, the helicopter accidents during landing and take-off operations were seen often when helicopters started to use with ships. High risk and high cost of experimental trials with full-scale ships and helicopter trials forced to researchers make model experiments and numerical studies. The experiments are generally executed in water or wind tunnel with the model scale of ships by using various force, velocity, pressure, momentum, etc. measure systems. The important point is to ensure geometric, dynamic and kinematic similarity rules to perform actual environment- ship conditions. Ship air wake cannot be generalized because different combinations have different characteristics. Every ship and helicopter combination needs to be modeled for various environmental conditions. For constructed ships, it is hard to ensure different test conditions and take pilot insights due to the risk of flight tests. On the purpose of analyse constructed or future design ships, both experiments and numerical methods are used by researchers. In recent years with increasing computing processor capacity, numerical methods have become a wide study area. It is possible to verify numerical methods by using previous experimental study results and make amendments in using program codes.

1.1 Literature Review

In order to analyse ship air wake and to amend ship-helicopter dynamic interface, the studies have done by researchers by using experimental and numerical methods. These studies have done for frigates which have helicopter hangar deck or some simple frigate geometries. SFS and SFS2 geometries commonly have used to validate CFD methods by comparing results with previous experiments or to understand and determine design change effects of ship geometric parameters.

To characterize ship air wake, wind tunnel experiments used since mid of 15th century. Generally, the full scale ships are modelled to keep geometric, dynamic and kinematic similarity rules. There is a certain similarity ratio between model and full scale ships which gives the less error in experiments. The experiments can be executed as one or two phase (free surface effect), ship movements (roll, pitch, and yaw), and helicopter rotor motions (flight simulations). The measurement methods have also their own error percentages due to different conditions (temperature, pressure, and humidity), researchers (inattention, wrong measure etc.), measurement devices and negligence. However, the experimental results are good agreement with real ship conditions and used to validate computational fluid analyses [7]. Blendermann studied about wind effects on different ship shapes. He classified ships as multiform, rectangular and unsymmetrical shape ships and he compared the angle of attacks, yaw moments according to wind tunnel experiments [8]. These results helped to understand wind effects on the ships and characterize unknown situations depending on obtained data. Haddara also studied about to analyse wind effects on various ship types and offshore structures such as container ships, drill ships, tankers, cruise ships and offshore supply vessels [9]. He used four different methods to estimate wind load including Blendermann`s method and concluded the most reliable method is experimental method in that time.

By developing software, it is possible to model ship air wake in computational methods. Theoretically, based on Navier-Stokes equations, the air flow can be modelled around the body. It is possible to analyze air flow with disturbances, movements, etc. which give idea to design ship more aerodynamically in the design process. It makes to possible to calculate all risks with different environment conditions and cost during this situations (fuel consumption). The important point is validation in numerical studies. To ensure this, CFD results were checked by based on previous experiments and the error between these studies are calculated. All ship and helicopter types with different environmental conditions have different air wake behaviours, so CFD studies had significant role in this area. In Tattersall's study, the ship air wake effects with rotor movements both were investigated by using CFD codes [10]. Nguyen and his researcher team focused on a container ship air drag with port and starboard direction winds [11]. Firstly, they mapped the container air wake by using CFD. Then, different geometric designs were applied to a container scale model in the towing tank and the wind fan was used to generate oblique winds. The containers on the ship deck cause the undesired load and moments and dependently increases air drag of containers [12]. In another study of this team, they tried to reduce air resistance by making geometric modifications over the container superstructure. They used four different coverage parts (full-side, front side, front half side, and center wall) to cover spaces between containers and compared velocity and pressure fields. It was seen that these added parts reduced the air drag dramatically [13]. The general logic of their studies is covering gaps and directing air over the whole superstructure by generating minimum resistance. These methods reduced the vortexes and moments significantly and contribute to constituting to enhance the aerodynamic shape of the container ship. Majidian and Azarsina also studied about container ship air drag reduction [14]. They used four different CFD simulations such as sea surface, symmetrical body, rotating plate and image method. By using these methods, the force and moments were compared with experimental data.

Yoonsik`s research team also studied to reduce air drag of a container ship but differently, they designed part over the bow (rounded and edged bow part) and gap protectors to lead airflow over the superstructure of the ship [15]. Andersen also studied to examine wind load on the container ships [16]. It was purposed to reduce fuel consumption by decreasing air drag around the container superstructure. Wnek studied to investigate air flow over LNG Carrier and floating platform models [17]. It was used both experimental and CFD methods. The CFD results were compared with wind tunnel test results to validate study.

Brizzolara investigated the negative pressure area over the flight deck due to wind heeling moments [18]. This zone causes the suction over the deck. Yelland studied to understand the effect of wind direction and speed on the wind flux over the deck [19].

The design changes in hangar shape and dimensions directly affect the turbulence, velocity and pressure fields in the flow region. Hangar geometry has sharp edges which cause the separation in the flow, velocity rotations and vortex shedding. It is possible to constitute more stable flow region by changing hangar shape. In Wang's study, the hangar geometry was changed as different angles to show turbulence map changes near ship air wake. The hangar angle was increased 10 degrees from 0^0 to 180^0 and the optimum angle was determined as 90^0 for this design [20].

Owen's researchers group studied to modify hangar edges by using added parts and geometric modifications. The effects` in the turbulence area of chamfer, flap, cylinder, saw-tooth, and tabs geometries were investigated. They found that the chamfer method is future design by comparing with several designs [21]. Ship air wake effects directly pilot workload, launch and recovery operations. At this point, the hangar geometry was tried to better air wake conditions to execute safer helicopter operations. As a further study of Owen`s group, the effects of hangar modifications on pilot workload were investigated using flight simulator. It was seen that the chamfer and the flap reduced turbulence significantly. On the other hand, the vertical modifications increased the turbulence and dependently pilot workload during flight operations. Air wake were generated, flight mechanics were modelled and pilot simulator was used. It was seen that by changing superstructure design, it is possible to modify air wake [22].

Yuan's study, the SFS2 model was used to validate wind tunnel experiments using model scale in both methods. Then, the CFD method was used for a petrol frigate to determine environmental (headwind and green 45 wind) condition effects on helicopter operations [23].

Forrest and Owen`s research group have focused on the helicopter piloted simulations to amend pilot flight experience. Firstly, they mapped the velocity gradient and turbulence intensity of the ship air wake, and then FlightLAB software used to simulate flight with a pilot. The pilot`s view took into account to determine better flight conditions [24].

Sysms used Lattice-Boltzman method to analyze flow region around SFS and SFS2 geometries [25]. This method gives to capture turbulence area and ensure to visualize flow topology accurately. In another study of Sysms`s, the air wake region around patrol frigate was investigated. He mapped the vortex region and velocity rotation area over the flight deck [26]. He modelled frigate and domain both sea and air (two phases), the ship air wake cases were compared according to different yaw angles.

In Forrest and Owen`s researchers group compared experimental and numerical models using Detached Eddy Simulation [27]. By this method, eddies in turbulence area were also taken into account, for this reason, the smaller time steps are needed to catch small eddies. In this study, time step was taken as 4E-5 with a large circular domain. In marine engineering, ships have similar geometries, which have adversely aerodynamic effects. It originates from the other design problems have priority comparing with an aerodynamic shape. On the other hand, the aerodynamic design has significantly affected LHA (landing helicopter assault) ships, naval ships and generally directly related with speed of the ship. Service speed has critical effect on the costs and power of the ship. Due to these reasons, the researchers have studied ship air wake effects and tried to minimize the adverse effect of the ship structures. Wind tunnel and water tunnel experiments have used to model ship and environment conditions and the results are convincing with actual situations [23]. Therefore, these experiments have used as a reference to validate CFD results in most studies. In recent years, ship/helicopter dynamic interface has studied by many researchers [28], [29].

In Zan`s study, both ship and helicopter effects were investigated together and also the assumption of the atmospheric layer and without atmospheric layer were taken into account. It was found that the exhaust gas over the deck causes to decrease helicopter power. The main reason for this situation, increasing air temperature affects the reduction in air density and aerodynamic performance of the helicopter. Also, the effects of the ship`s roll and pitch angles were investigated [30]. In another study, funnel exhaust gas temperature effect on turbine inlets were investigated. The funnel exhaust gas has increased the inlet air temperature which causes the efficiency loss on turbines. Therefore, researchers made some changes on funnel outlet to lead air flow and added part changes on turbine inlet to prevent to enter hot air into turbines. They had good results and increased the efficiency of the turbines in this study [31]. In another study of Singh`s team, they investigated the funnel exhaust gas behaviour through ship deck [32]. Since the effective parameters are complicated for this study, they constituted 112 cases to understand effective parameters and exhaust flow characteristics. They chose a specific ship type and simplified it as four different cases. They used both experimental and numerical methods to execute different superstructure configurations with regarding yaw angle. In another study, helicopter and ship dynamic interface were investigated using both CFD and water tunnel experiments [7]. The aim of the study is verifying to CFD results in real situations to use future design ships with better aerodynamic features. In another study, UK Naval Ship, Queen Elizabeth was used as a model in different wind conditions [33]. The turbulence map of the ship was investigated and compared both model and full-scale situations. In some studies, the identify air wake as general, the simply frigate model were chosen [34]. The effects of different wind conditions were studied on the ship air wake. Some researchers used different turbulence methods such as Detached Eddy simulation to define more accurate turbulence model [27]. Moreover, in some universities, the researchers used their own software to avoid errors in CFD models and get more reliable results [7]. On the other hand, some researchers used more than one software to compare results and eliminate errors [35]. In helicopter/ ship dynamic interface studies, it is common to use flight simulator to define pilot workload [29]. They defined the ship air wake firstly and apply this model into the FlightLAB. Then by taking the pilots comments about flight experience, they investigated turbulence locations over the deck. In another study, the effects of ship sizes were investigated using a patrol vessel, which has helicopter-landing area. Three ship sizes were used to compare with turbulence intensity, mean forces and moments in air wake region [36]. Kaaria's study, it was aimed to decrease the bad weather conditions effect on the ship/helicopter interface. To achieve this purpose, different added parts were used on the side of the ship. Basic ship model and added-part models were investigated in water channel. It was seen that the added parts have reduced the aerodynamic load on the helicopter and also pilot workload [37]. In some studies, special situations like velocity burst and gusts were investigated due to adverse effects on helicopter hangar area [38]. The main point was to determine the cause and location of gusts on the deck, which inconvenience to landing and recovery operations. In another study, ship/helicopter interface was investigated with modelling rotor blades using two different assumptions which were actuator disc model and Reynolds averaged Navier Stokes equations [39]. In these studies, ship models were chosen simply frigate to standardize air wake region generally. Also, in some studies it was focused on different wind angles [40]. The air wake area and turbulence model also change with different wind conditions. Geometric modifications effects to reduce adverse effects of airflow region on helicopter operations were examined in the water tunnel in Kaaria's another study [41]. Airdyn is 1:54 scale model of Merm EH-101 helicopter which can measure forces and moments caused by ship air wake turbulence. SRF (Shortened Research Frigate) was used as the ship model. It was possible to change helicopter location according to the turbulence area to measure forces and moments from these regions. Wind directions are represented as red and green which are shown in the figure 1.3. This graph represents ship-helicopter operating limits (SHOL) as direction and angles. Red and green describe port and starboard respectively on naval terminology. It shows the safe regions for flight operations over the deck. By using this test setup, different hangar modifications were applied on SRF geometry, forces and moments were measured to understand turbulence change inflow region. Also, aerodynamic disturbances between specific values were measured and geometric design effects on disturbance reduction were investigated by focusing G30-G45 wind directions especially. Similarly, Owen's researcher group studied about ship air wake modification by adding geometric parts to hangar deck area [42]. It was seen that notch and two different flap geometries have different workload on the pilot during flight operations. All of them reduced the unsteadiness of the flow region. To compare flight experiences, FlightLAB simulation was used. These studies were purposed to estimate the general wind effect map for ships and to amend flight conditions over the deck. In some of these studies were assumed the flow region as a single phase, which is easier to calculate and take less time with the computer process. However, for desired solutions, free surface and air assumption will give more realistic results. Moreover, ship movement is another significant effect in air wake problems. It is a common assumption to consider flow region independence from ship movements, which are roll, pitch, and yaw. In general, all ship/ helicopter interface problems have investigated using flight simulations of pilot experience, which is main part of the study. The objective results and changes for amending flow region can be determined after this part of study. Most of the studies have focused on operating ships. It means that the possible change on ships cost higher than in-design process.

It is concluded that ship air wake studies can be divided into three according to sea-ship motion, ship air wake modelling and combined ship-helicopter studies.

Firstly, in some studies were focused on ship movement effects on ship air wake. In this method, sea states (different sea environmental states) and ship motion (roll, pitch and yaw) are modelled to investigate the effects of ship motions on the ship air wake [43]. Basically, the ship locates in the specific coordinate system and for defined axis, the motion defines with a desired period. Theoretically, Navier-Stokes equations and Newton equations have solved together to understand motion effect.

Secondly, it is neglected the ship movement effects and focused on ship geometry since it directs the air flow and shapes the air wake. By using this method, ship geometry effects have investigated with parametric studies. Ship geometry has certain edges and separation areas which produce turbulence area and instability in the ship air wake. These parameters directly affect atmospheric boundary layer, mean velocity profile, and turbulence intensity. To get more stable air wake region and ensure safer flight region for helicopters, it is aimed to design the optimum ship geometry by changing these parameters i.e. hangar height, funnel height and location etc. [44].

Thirdly, some studies have both ship air wake analyses and pilot simulators to understand flight experiences. In these studies, the helicopter rotor effects also take into account using flight simulators or test flights. These cases have more variables which make the problem complicated and increase the solution time. However, these studies give opportunity to simulate flight. The rotor motion and the adverse effects on the helicopter rotor can be investigated using this method [28].

To sum up, ship-helicopter investigations are performed as experimental and numerical methods. The main point in these studies is to understand airflow behavior over the deck and effects during landing and take-off operations. To amend ship-helicopter dynamic interface, different geometry trials have been tried by researchers. The ship aerodynamic is taken into account as a reference. By using different ship geometries, the impacts of various parameters, the flow directions, and sea conditions were investigated. In general, it is seen that sharp edges have adverse effects on the air wake. It causes the velocity rotations in-flight region and turbulence eddies which has instability on helicopter rotors. It is seen that the essential parameters which affect ship air wake characteristics are as follows:

- Hangar Height
- Atmospheric Boundary Layer
- Mean Velocity

- Turbulence Intensity Level
- Turbulence Length Scale

Hangar height is the main parameter which directs airflow over the flight deck. In this study, the effect of hangar height on the ship air wake will be investigated. Also, the hangar edges have critical importance to determine the shape of velocity vectors over the flight deck. Moreover, it is possible to change the velocity gradient in this area.

1.2 General Motivations

The background theory of visualizing ship air wake is to understand the general characteristics of air over the ship deck and the effects of helicopter flight. Over the decades, the researchers focused on the different ship-helicopter scenarios to model general rule. Ship air wake directly affects flight conditions and if the ship superstructure design becomes more aerodynamic structure, these landing and flight operations can be performed more safely and economically.

Turbulence over the deck affects directly the helicopter rotor, and it costs more fuel and more effort for pilots. Hence, the desired design is to achieve a superstructure to reduce turbulence and ensure a more stable ship air wake.

First of all, the general problem is to understand air wake behavior. To ensure this, early in the 15th century it was made experimental studies. However, due to the significant cost and take more time, with developing technology, it is possible to make numerical analyses using commercial CFD software.

First CFD studies were validated by using previous experimental studies that were made in generally wind tunnels. After ensuring closer results and decreasing error percentages, the computational methods have been used for new ship designs which have a significant effect to reduce turbulence over the ship deck.

The studies are divided into three categories:

1. Ship geometry (without motion (yaw, roll, pitch) just design changes in geometric parameters)

2. Ship and helicopter dynamic interface (with using ship geometry, rotor motion also is considered as a parameter in the studies)

3. Ship motions (Ship air wake investigation with considering ship motion to design environmental conditions)

It has significant importance to prevent accidents over the deck and ensure safer helicopter operations. In this study, ship geometry effects will take into account to map ship air wake.

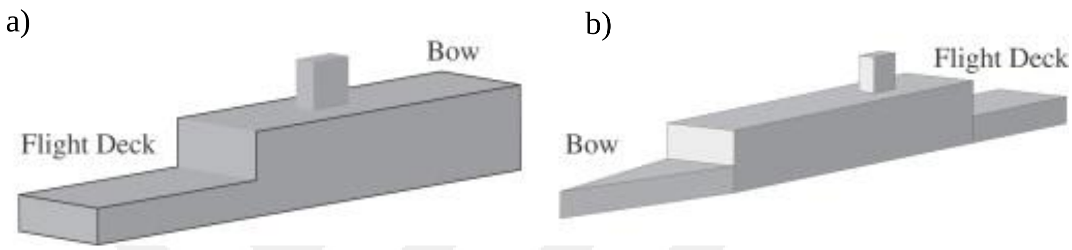


Figure 1.4 a) Simple Frigate Shape (SFS) and b) Simple Frigate Shape 2 (SFS2) [25]

Since the shipping sector is competitive, it is hard to get merchant ship geometries. To solve this problem, TTCP (The Technical Cooperation Program) researches determine a simple frigate model which has basic parts as hull, mass or mast, bow, and funnel. The several studies have been made with this geometry to understand air wake behavior and constitute general theories which can reduce air wake instability.

This geometry consisted of SFS in the 1980s firstly, and then the bow part added in 1990s. It is seen that hangar height directly affects the turbulence intensity and velocity profile over the deck. The effect of different hangar heights will investigate firstly. Moreover, some of the studies made with SFS2 geometry shows that some geometric added parts and design changes reduce turbulence over the deck. The chamfered design is chosen to investigate with two different angles to analyze ship air wake also. After these analyses, it is aimed to choose a proper height and chamfer angle to determine better air wake conditions.

1.3 Main Aims and Objectives

The main aim of the present study is to reduce the mean velocity and turbulence over the flight deck of the ship using CFD methods. It is purposed to investigate flow characteristics on the superstructure of the ship, to investigate the effects of different parameters of the ship air wake, to propose design changes in the superstructure of the ship to execute safe flight operations by carrying out the literature review.

The bluff body causes the separation on the ship air wake. This contributes the turbulence locations and discontinuity in the flow. The upcoming wind direction, ship motion, the structural elements on the deck is the main effects of air wake characteristics. In general, it contributes to increase eddy viscosity in the ship-helicopter dynamic interface, which is critical to execute helicopter landing and recovery operations for desired conditions. The turbulent flow is more common in the environment and it causes unsteadiness in ship air wake. The wind velocity behind the ship is desired as low as possible to avoid undesired wind loads on helicopter rotor. In designing phase, it is a fundamental process to model ship air wake to determine forces and moments on the helicopter rotor and minimize these forces by design changes.

An operating naval ship model was analyzed using CFD software to determine flow region around the superstructure body. As a beginning, the ship body were considered in a single phase (air) and ship movements' effects were neglected. For headwind condition the ship air wake were mapped. The mean velocity, pressure, and turbulence viscosity at this region were calculated using CFD software based on Navier-Stokes equations. With these equations, it is possible to calculate pressure gradient on air flow region. Using these pressure values, speed and drag coefficient can calculate easily with using computer software. The main purpose is reduced velocity rotations and vorticity area for reducing turbulence in flow region. Effects of sharp edges, bluff geometry, and location of deck hardware will be investigated to optimize ship's effectiveness. By making changes in design, the change of turbulence effects will be compared.

1.4 Hypothesis

The frigates have their own characteristic air wake depends on different superstructure shapes which affect directly the landing and take-off operations of helicopters. The pilot workload depends on the turbulence area over the flight deck.

It is thought if the frigate superstructure design can be changed to reduce turbulence effects and unsteadiness over the flight deck, it will amend the pilot control on this critical area, also will reduce the accident possibilities. To reach this purpose, a parametric study was run by using simple frigate shape 2. It is expected to reduce turbulence area and velocity changes over the flight deck.

2
METHODOLOGY

Ship-helicopter investigations can be studied by using experimental or numerical methods. The experimental method requires more time than the CFD method. Due to the model construction period, measurement systems, and other test apparatus, the set-up process takes long time and it is more expensive. In the past, model experiments were common to execute closer results with actual cases. Model experiments give the chance of change conditions flexibility by comparison with full-scale experiments. It has a higher risk of accidents with full-scale ship and helicopters and hard to control helicopter over the deck with high-speed wind conditions. Numerical methods started to use first to validate experiments and actual data. Since this method reduces the calculation time and provides to test various scenarios in a short time, it is preferable than experiments. In this study, the flow region of a vessel is investigated using computational fluid dynamic methods. These methods are based on Navier Stokes equations. Using these equations, it is possible to determine pressure and velocity values in flow regions. These methods can simplify as steady and unsteady states. Steady state does not depend on time and the flow qualities do not change with time. Unsteady state defines the flow region, which change with time. Also, the flow region defines as viscous flow and non-viscous flow in some situations. To estimate fluid behavior in flow region, iteration methods are used to calculate all desired values. The important point in this method, the previous results and last results compare regularly and calculation process continues until error value less than a specific value. There are many softwares, which enables to calculate these complicated equations. In this study, Star CCM+ is used to define specific ship geometry and to analyse flow region around superstructure. It has critical effect on calculation to define flow region correctly in these softwares. The flow region is divided desired control volume parts, which ensure to map all volume visually in program. The mesh quality should ensure to define all features of body. The smaller cells are always good to get closer results to actual situation. However, the calculation time and needed processor of computer should be risen comparatively. Some of simplifications and assumptions could be made as beginning. According the received data, the calculation will be close to the real situation. The meshing area identifies the problem solution area, which means to model critical area for the air flow.

Simple Frigate Shape (SFS2) geometry is an open source basic ship geometry which ensures to make researches and share the knowledge about new findings. In this study, the model scale is 1:100 and the domain is rectangular which has velocity inlet and pressure outlet, the other regions and ship is adjusted as wall (no-slip). The simulations were started from k-epsilon turbulence model. The results were compared with literature data and it was seen that the difference between validation and data are very high. Then, the mesh cell sizes were decreased, the flow type was changed as steady flow. The results are converged but not so close. The effects of different turbulence models have tried to understand effects on the ship air wake. K-omega and Spalart Allmaras models were run and it was seen that Spalart Allmaras model gave the closest results with literature. Since the flow is aerodynamic, this model is more capable to solve this problem. In the literature study, detached eddy simulation was used and the time step is 4E-5. In this study, the Reynolds Average Navier Stokes (RANS) method was used due to time limits. This method is not capable to solve small eddies. With these reasons, the flow region was solved as steady by using Spalart Allmaras for new geometries. Since the RANS method is not sensitive to solve eddies in the turbulence area as DES method, the results are assumed as a good agreement with literature.

2.1 Approach

It should be following an analytical approach to solving a problem using physical and mathematical problems. The main step is to construct the mathematical models properly. By making some assumptions and idealizations it is ensured to simplify and generalize the problem. The physical properties are used to identify conditions and main and auxiliary equations are used to reach the solution [45].

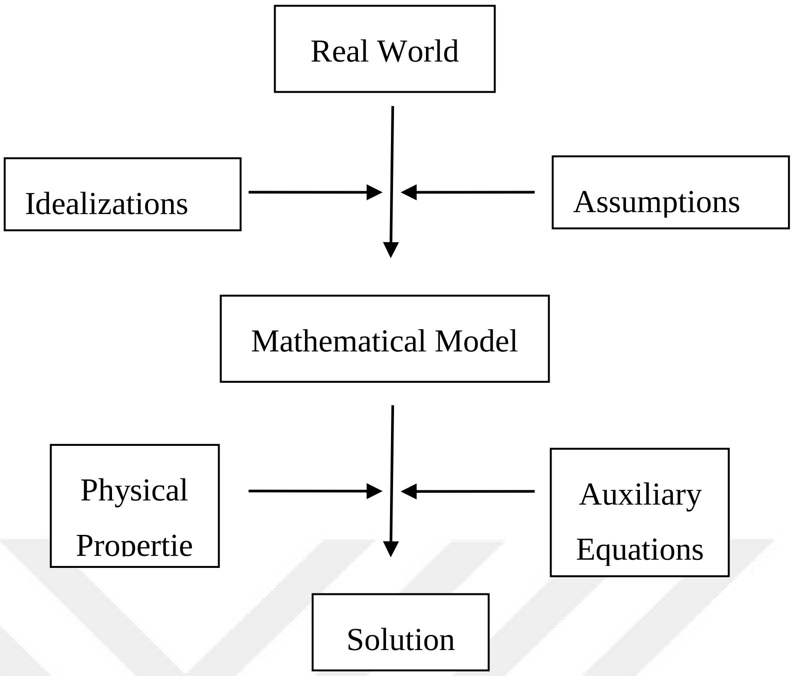


Figure 2.1 Solution steps for a problem using analytical approach [45]

The Reynolds Averaged Navier Stokes (RANS) method is used to solve air flow region over the hangar deck. The unsteady analyses was carried out by using GCI method which developed by Roache [46]. The flow chart shows the steps of this study below.

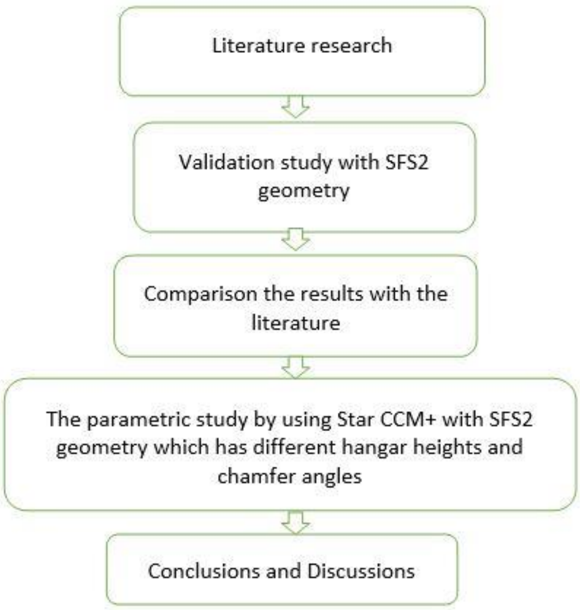


Figure 2.2 Methodological order of this study

2.2 Theoretical Background

In this study, Star CCM+ is used to define specific ship geometry and to analyze flow region around superstructure. It has critical effect on calculation to define flow region correctly in these softwares. The flow region is divided desired control volume parts, which ensure to map all volume visually in program. The mesh quality should ensure to define all features of body. The smaller cells are always good to get closer results to actual situation. However, the calculation time and needed processor of computer should be risen comparatively. Some of simplifications and assumptions could be made as beginning. According the received data, the calculation will be close to the real situation. The meshing area identifies the problem solution area, which means to model critical area for the airflow, the Reynolds Average Navier Stokes (RANS) method is used due to time limits.

2.2.1 RANS Method

The airflow over the deck is solved using RANS method that solves on Navier-Stokes equations. The flow is assumed as incompressible and steady in 3-dimensions. The continuity equation is:

$$\frac{\partial U_i}{\partial x_i} = 0 \tag{2.1}$$

The continuity equation symbolizes the mass conservation in the flow region depends on Cartesian coordinate velocity components.

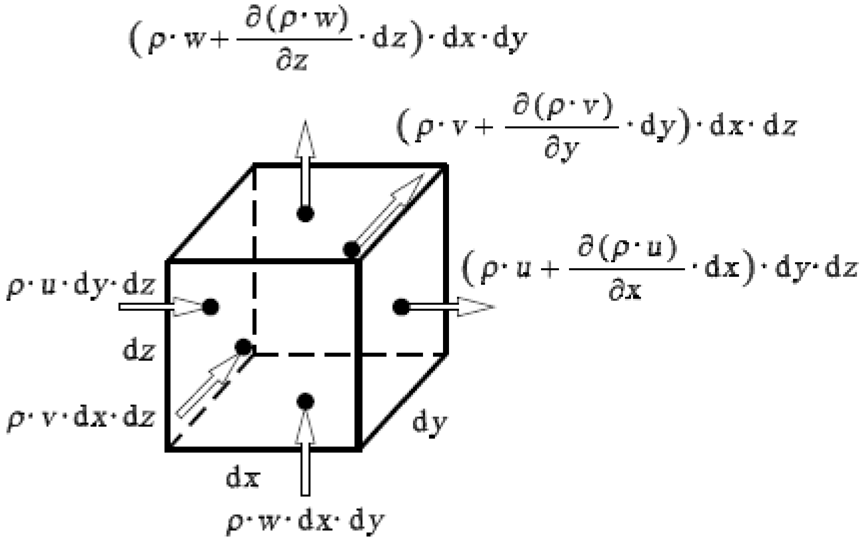


Figure 2.3 The volume element in the flow region which shows mass conservation in three dimensions [47]

In the figure 2.3, the x velocity enters to volume by passing through dz and dy surfaces. Also, momentum is conserved in flow region as mass flow. The momentum entering and leaving the volume unit can be written with a formula by velocity gradient and pressure gradient depending on location. And the momentum equation is given as below [48]:

$$\frac{\partial U_i}{\partial t} + \frac{\partial \left(U_i U_j\right)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\nu\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right)\right] - \frac{\partial \overline{u_i' u_j'}}{\partial x_j} \tag{2.2}$$

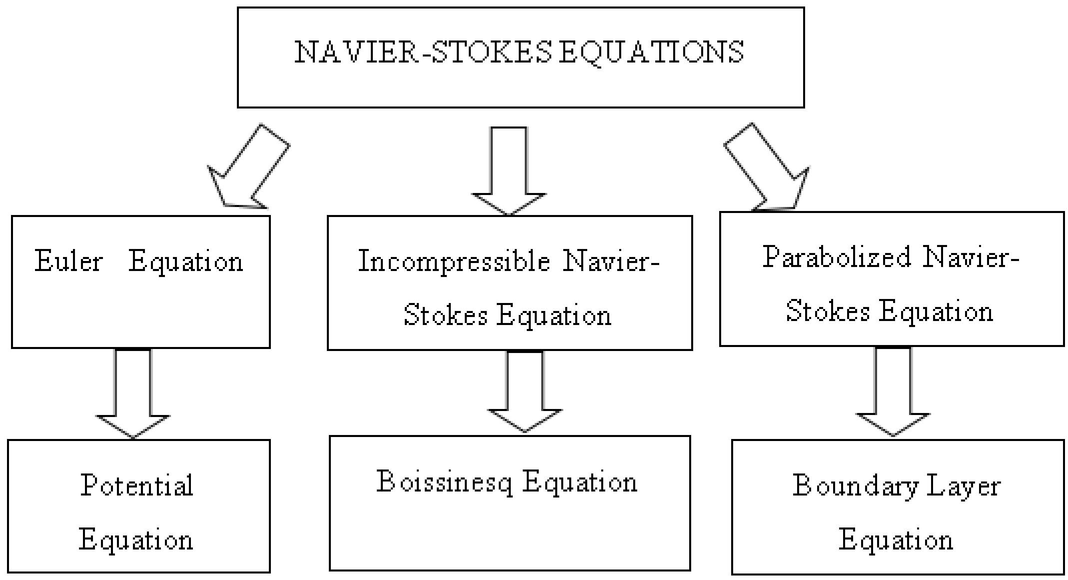


Figure 2.4 The model equations obtained by simplifying Navier-Stokes equation [47]

The Navier Stokes equations can be simplified by making assumptions in flow problems. If the flow is in-viscid, it changes into the Euler equation. For this assumption, if the flow is irrotational, it is simplified into the potential flow. The flows which has constant density gives the incompressible Navier-Stokes equation. Moreover, if the density is depending on temperature, it is formulazed with Boussinesq equation. The parabolized Navier-Stokes equations is used at large Reynolds numbers which the boundary layer thickness is so small comparing by body dimensions and can be neglected [47].

2.2.2 Uncertainty Assessment

The GCI method is used to conduct an uncertainty study [49], [50], [51]. The main aim of this method is to investigate the refinement of the mesh of the simulation. Three different mesh configurations (coarse, medium, fine) are created to calculate the required values.

$h1<h2<h3$

The refinement factor is calculated using obtained values GCI formulas that are given in Chapter 3.1 in details. The refinement factor (r) is recommended to be greater than 1.3 to assume the error of the mesh refinement is acceptable [52].

Grid refinement factor is defined as:

$$r_{21} = \frac{h_2}{h_1} \qquad r_{32} = \frac{h_3}{h_2} \tag{2.3}$$

In this study, refinement factor is defined as:

$$r_{21} = \left(\frac{N_1}{N_2}\right)^{1/3} \qquad r_{32} = \left(\frac{N_2}{N_3}\right)^{1/3} \tag{2.4}$$

After checking the refinement factor, the difference between the numerical results is used for calculation of grid convergence condition (R).

$$\varepsilon_{21} = X_2 - X_1 \qquad \varepsilon_{32} = X_3 - X_2 \tag{2.5}$$

$$R = \frac{\varepsilon_{21}}{\varepsilon_{32}} \tag{2.6}$$

2.2.3 Turbulence Models

k-ω , k-ε and Spalart Allmaras turbulence models are used in this study. k-ω model is a two equation turbulence model which has k, turbulence kinetic energy and ω as specific rate of dissipation [53]. Both parameters symbolize differential equations. Kinematic eddy viscosity is given with this formula,

$$, \nu_T = k / \omega \tag{2.7}$$

Kinetic energy, k;

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=\tau_{ij}\frac{U_i}{\partial x_j}-\beta^* k\omega+\frac{\partial}{\partial x_j}\left[\left(\nu+\sigma^*\frac{k}{\omega}\right)\frac{\partial k}{\partial x_j}\right]$$

(2.8)

Specific rate of dissipation, ω;

$$\frac{\partial \omega}{\partial t}+U_j\frac{\partial \omega}{\partial x_j}=\alpha\frac{\omega}{k}\tau_{ij}\frac{U_i}{\partial x_j}-\beta\omega^2+\frac{\sigma_d}{\omega}\frac{\partial k}{\partial x_j}\frac{\partial \omega}{\partial x_j}+\frac{\partial}{\partial x_j}\left[\left(\nu+\sigma\frac{k}{\omega}\right)\frac{\partial \omega}{\partial x_j}\right] \quad (2.9)$$

k-ε model is also two equation turbulence model which explains k as turbulence kinetic energy and ε as dissipation rate.

Kinematic eddy viscosity formula,

$$\nu_T=C_\mu k^2 \,/\, \varepsilon \quad (2.10)$$

Turbulence kinetic energy, k is given as;

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=\tau_{ij}\frac{U_i}{\partial x_j}-\varepsilon+\frac{\partial}{\partial x_j}\left[\left(\nu+\nu_T/\sigma_k\right)\frac{\partial k}{\partial x_j}\right]$$

(2.11)

Dissipation rate, ε;

$$\frac{\partial \varepsilon}{\partial t}+U_j\frac{\partial \varepsilon}{\partial x_j}=C_{\varepsilon 1}\frac{\varepsilon}{k}\tau_{ij}\frac{\partial U_i}{\partial x_j}-C_{\varepsilon 2}\frac{\varepsilon^2}{k}+\frac{\partial}{\partial x_j}\left[\left(\nu+\nu_T/\sigma_\varepsilon\right)\frac{\partial \varepsilon}{\partial x_j}\right]$$

(2.12)

Spalart Allmaras model is one equation turbulence model. It is given as also with kinematic eddy viscosity,

$$\nu_T=\nu f_{\nu 1} \quad (2.13)$$

Eddy viscosity equation

$$\frac{\partial \nu}{\partial t}+U_j\frac{\partial \nu}{\partial x_j}=c_{b1}S\nu-c_{\omega 1}f_\omega\left(\frac{\nu}{d}\right)^2+\frac{1}{\sigma}\frac{\partial}{\partial x_k}\left[\left(\nu+\nu\right)\frac{\partial \nu}{\partial x_k}\right]+\frac{c_{b2}}{\sigma}\frac{\partial \nu}{\partial x_k}\frac{\partial \nu}{\partial x_k}$$

(2.14)

2.3 Numerical Modelling

The flow problem has been solved by Reynolds Averaged Navier Stokes method which based on Navier Stokes Equations [48]. These equations are based on mass and momentum conservation.

$$\frac{\partial \bar{v}_i}{\partial x_i} = 0 \tag{2.15}$$

$$\frac{\partial v_i}{\partial t} + \frac{\partial}{\partial x_j} v_j v_i + \frac{1}{\rho}\frac{\partial p}{\partial x_i} = \nu \frac{\partial^2}{\partial x_j^2} v_i - \frac{\partial}{\partial x_j} \overline{v'_j v'_i} \tag{2.16}$$

It is known that high-velocity differences and turbulent regions cause the problems on the helicopter flight operations. To understand the effects of hangar height and chamfer design on the airflow region, four different geometries are constituted. Hangar height is changed as ±%10 of total height. Hangar angle is changed as x/h= 0.75 and x/h=1 by constant y/h=0.25 based on Kääriä`s study [54]. The chamfer angles are α=14.03° and α=18.43° in order.

2.3.1 Boundary Conditions

The SFS2 geometry is located on the x-y-z coordinate system which has the origin at hangar midpoint. The frigate service speed is taken as 20 knots. The headwind velocity is set as 30 knots, 40 knots, and 50 knots during the x-axis. Depending on similarity theory, the similarity conditions were ensured for 1:100 scale model.

Similarity ratio:

$$\frac{L_s}{L_m} = \lambda_L \tag{2.17}$$

To ensure kinematic similarity condition, the Froude number similarity is given with formulas below [55], [56],

$$Fn_m = Fn_s \tag{2.18}$$

$$\frac{v_m}{\sqrt{gL_m}} = \frac{v_s}{\sqrt{gL_s}} \tag{2.19}$$

$$\frac{v_s}{v_m} = \sqrt{\frac{L_s}{L_m}} = \sqrt{\lambda_L} \tag{2.20}$$

The domain size is 12x12x6 m3 rectangular geometry. SFS2 geometry and tank bottom were set as wall. Tank inlet was set velocity inlet and tank outlet is pressure outlet. Both sides and top was set as symmetry. For constant gas density, the Spalart Allmaras Turbulence model was chosen for steady flow. The boundary layer on the deck was calculated by using boundary layer formulas which based on Reynolds number.

$$\mathrm{Re} = \frac{\rho L U}{\mu} \tag{2.21}$$

In this formula ρ defines fluid density, L is ship beam, U is headwind speed and μ is fluid dynamic viscosity. Based on this formula, the boundary layer is defined with this formula:

$$\delta = \frac{0.37x}{Re^{1/5}} \tag{2.22}$$

The boundary layer was calculated from this formula. Then, the number of prism layer was chosen as 5 and the boundary layer was set 0.0072m. The y+ values are compared to reach theoretical boundary layer thickness which is calculated from formula. The y+ values are higher than 40 for this boundary layer.

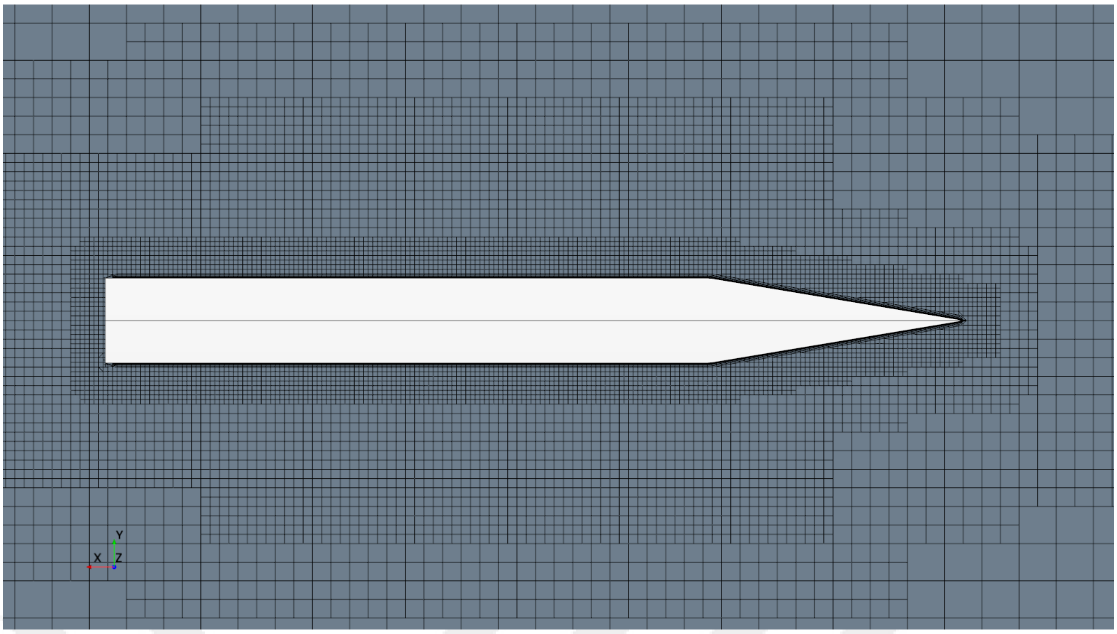


Figure 2.5 SFS2 Mesh Refinement Top View

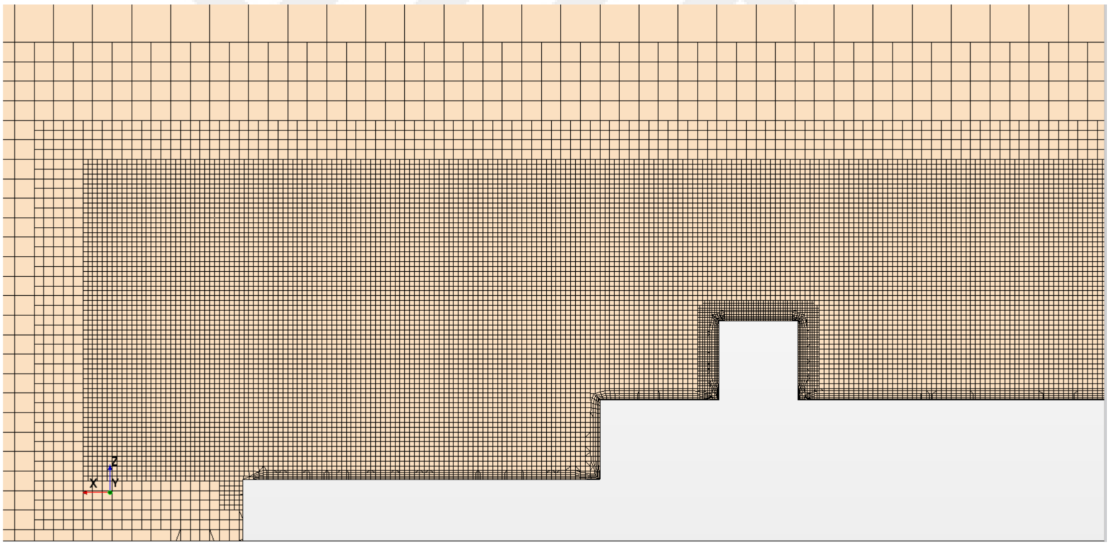


Figure 2.6 Hangar area mesh refinement

The measurements were taken at hangar height at the mid of the flight deck as can be seen in figure 20 below.

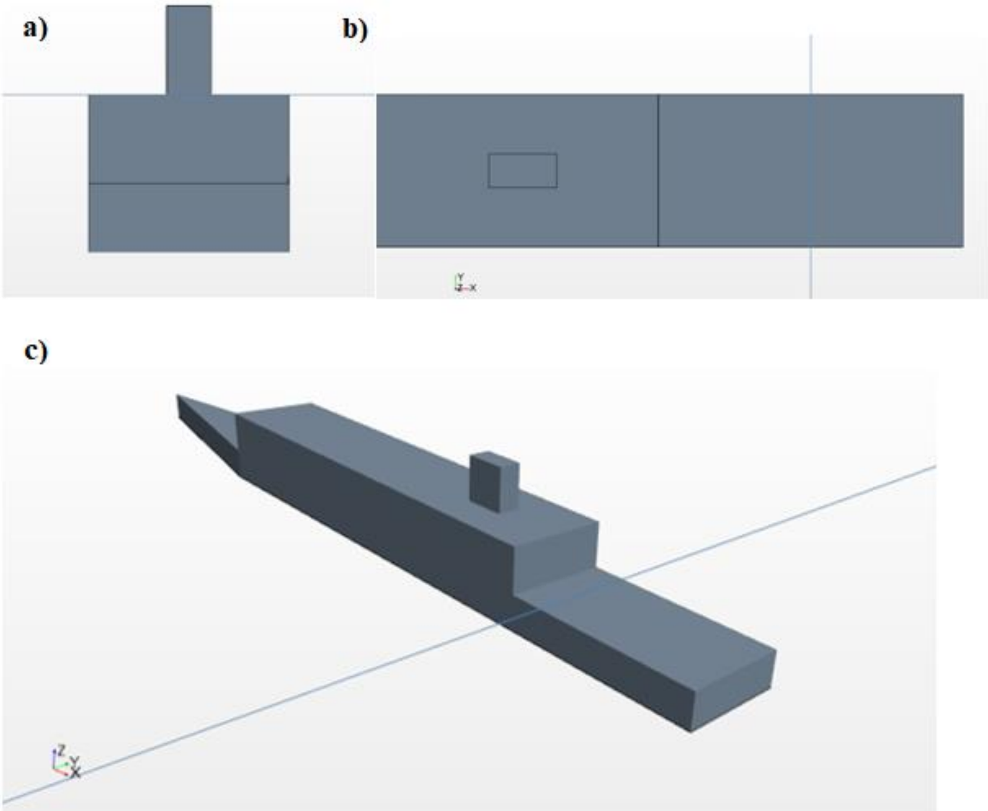


Figure 2.7 Measurement point back view (a), top view (b) and perspective view (c)

3
VERIFICATION AND VALIDATION

3.1 Geometry and Boundary Conditions

In this study, it is aimed to compare CFD results with the experimental and CFD study results [23]. The model scale SFS2 was used in the wind tunnel experiment. For headwind at 60 m/s, the velocity gradient has measured over the hangar deck. X axis was elongated during to ship length and y axis was elongated to ship beam. The coordinate system was located at the beginning of the hangar deck symmetrically. The measurements have recorded and mapped for x and y velocities at the 75% height of the hangar (z-axis) and 50% length of the hangar deck (x-axis). The velocities and locations have shown in the graphs as dimensionless. Moreover, another CFD study which was done by Owen`s research group also have shown and compared. It has seen that the general characteristic of air behavior and velocity magnitudes have good agreement to each other.

To validate SFS2 geometry with 1:100 scale, Star CCM+ software has used. The validation study has run firstly for the steady condition for converging faster at the unsteady conditions. Then, for 8 seconds simulation, using k-epsilon turbulence model, the simulation is arranged with finer mesh to get closer results. k-ω and Spalart-Allmaras model simulations were run as steady flow condition. 3-D model of the SFS2 geometry can be seen in Figure 3.1. Main dimensions of the SFS2 geometry are shown in Figure 3.2.

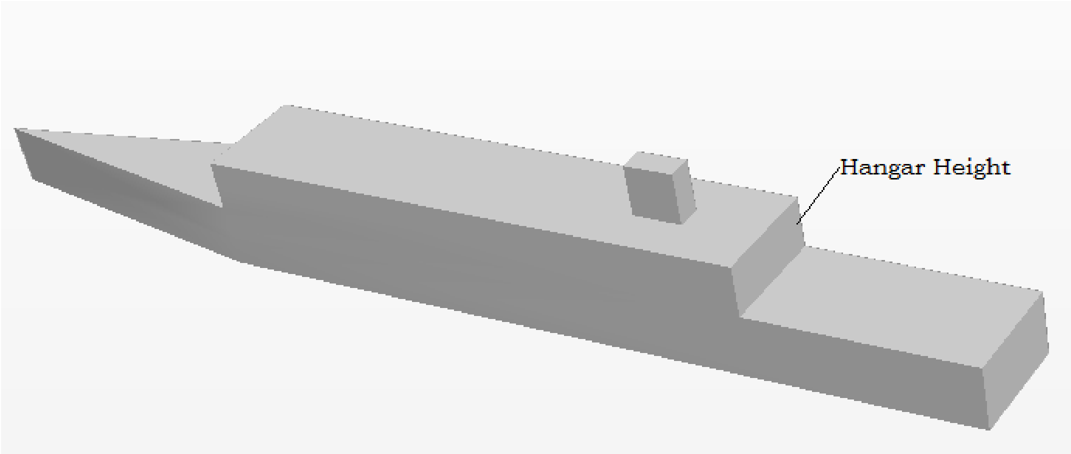


Figure 3.1 SFS2 geometry

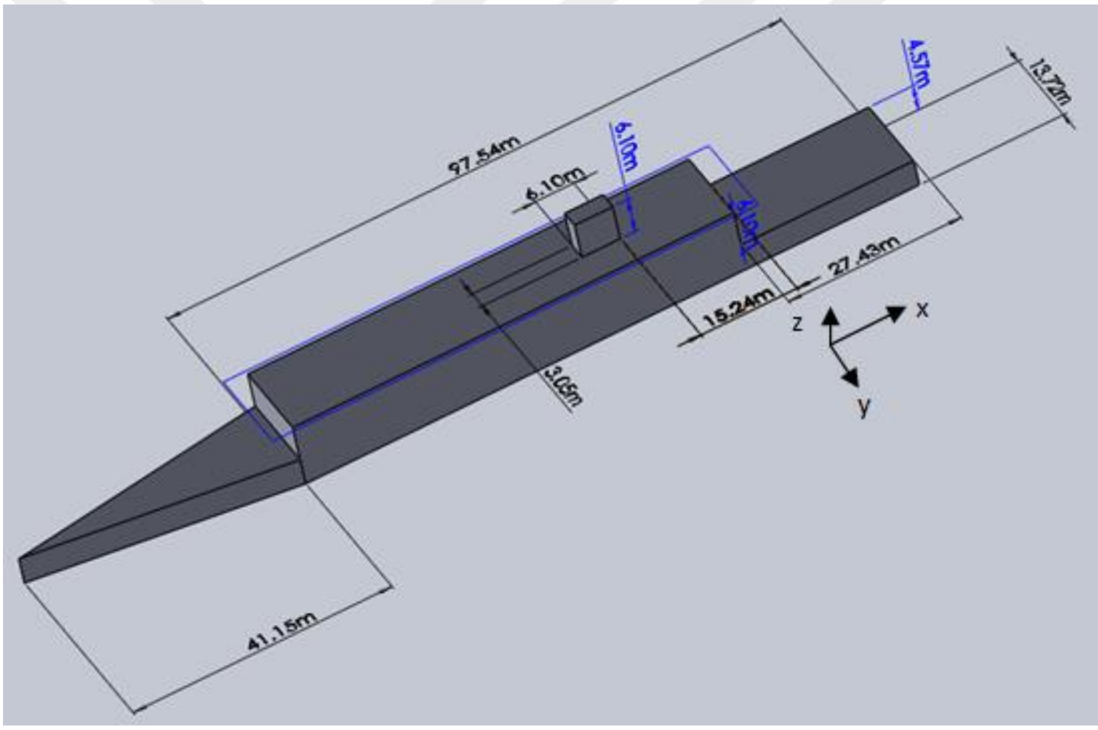


Figure 3.2 The dimensions of Simple Frigate Shape 2 (SFS2) geometry

The SFS2 geometry is located on the x-y-z coordinate system that has the origin at hangar midpoint. The headwind velocity is 60 m/s along x-axis. The domain size is 12x12x6 m3 rectangular geometry that can be seen in Figure 3.3. SFS2 geometry and tank bottom were set as no-slip wall. Tank inlet was set as velocity inlet and tank outlet as pressure outlet. Both sides and top were set as symmetry plane. Spalart-Allmaras turbulence model was chosen for modeling the turbulent and steady flow. The boundary layer on the deck was calculated and prism layer was applied near the wall boundaries.

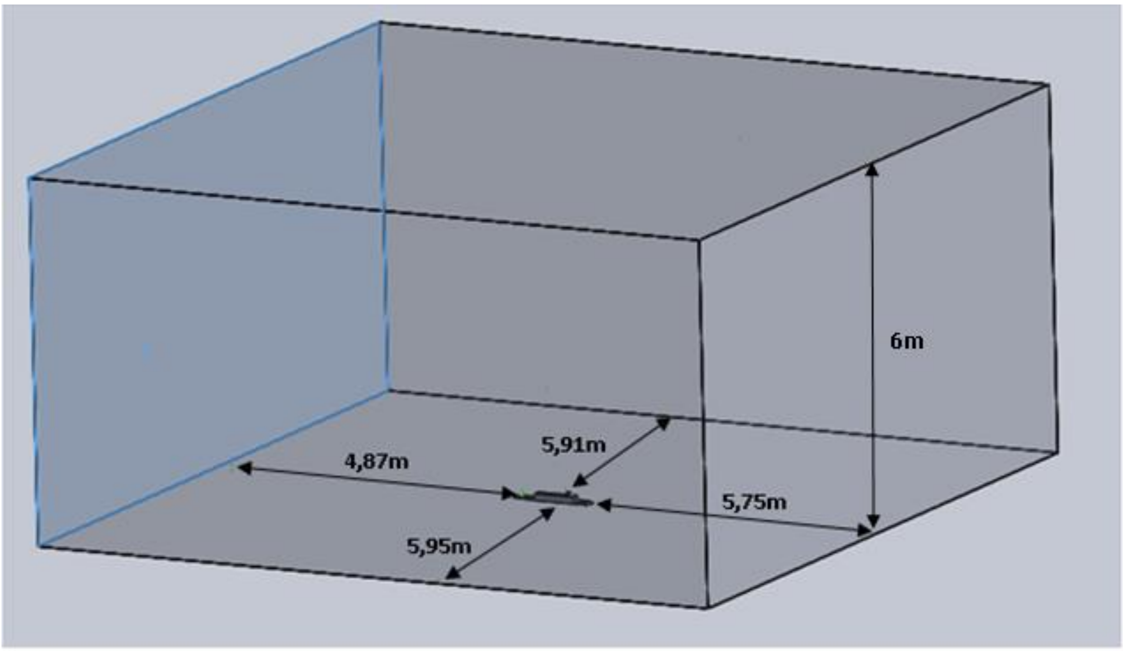


Figure 3.3 Computational domain around the SFS2 geometry

3.2 Verification Study

The uncertainty analyses were made to verification of CFD simulation in this study. To determine the mesh cell numbers` effect on the X-velocity component, the model with two meshes were simulated. The cell numbers and velocity ratios were given in Table 3.1. The results were compared with the original base size. With the higher cell number, the velocity values are getting closer to literature values. To calculate uncertainty, the X velocities at the same y-axes are compared.

Table 3.1 X velocity to headwind speed ratio for different mesh configurations

Mesh Configuration	Total Number of Cells	u/Umean	Difference
Coarse	492823	0.517661	%22,81
Medium	1162680	0.526735	%21,45
Fine	2984663	0.65336	%2,57
Experiment	-	0.670654	-

Table 3.2 Uncertainty analysis

N_1	2984663
N_2	1162680
N_3	492823
r	1.4142
ϕ_1	0.65336
ϕ_2	0.52673
ϕ_3	0.51766
R	13.9547
P	7.6054
φ_{ext21}	0.66313
e_a^{21}	19.3806

e_{ext}^{21}	1.474
GCI_{fine}^{21} (%)	1.87

In table 3.1, the velocity change with different mesh numbers was given for a specific point. By using the data from these mesh configurations, GCI calculations were made and the results were given in Table 3.2. The GCI value was found as %1.87 which means the velocity value change with mesh configuration is below 2%. Since the velocity value is closer to literature value in fine mesh configuration and GCI value is acceptable, the simulations were performed for this mesh configuration for parametric study. The fine mesh configuration can be seen in Figure 3.4 and Figure 3.5.

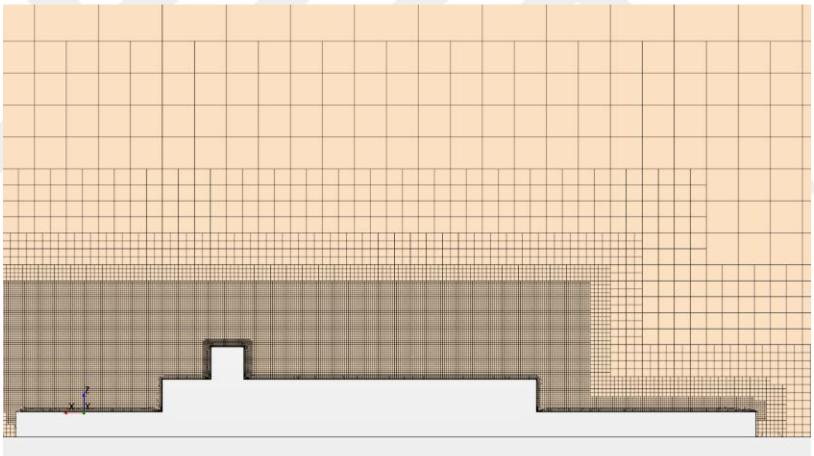


Figure 3.4 SFS2 Mesh Configuration front view

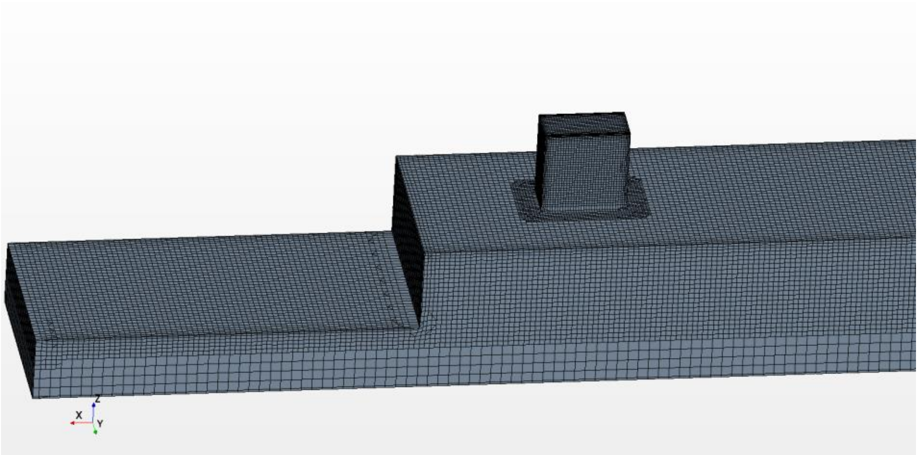


Figure 3.5 Local mesh refinements near hangar and funnel

3.3 Validation Study

The first analyses have run using k- epsilon turbulence model that is a general and most common model used in the flow problems. To capture sharp gradients in this region, the mesh cell numbers are increased to get high precision in air wake region. Base sell is decreased as base size/√2 and base size/2 respectively and the results are compared with literature and original base size. The other parameters depend on the percentage of base size. Since the convergence ratio is not the desired value, the other turbulence models are applied to the simulation. K-ω turbulence model gave very good results as it can be seen in figure 3.7. This turbulence model is generally used to solve shear stress near the walls. Lastly, the Spalart-Allmaras turbulence model is used in the simulation and the results are converged. Since the Spalart-Allmaras model is mostly used to aerodynamic problems, this model is chosen for the prevalent turbulence model in this study. As it can be seen in the graphs, in the mid of the deck, X velocity has the fluctuation in contrast to literature values. The RANS method has less sensibility to solve turbulence eddies in compared with Detached Eddy Simulation methods. The measurement point is shown in figure 3.6. The obtained X and Y velocity data can be seen in Figure 3.7and Figure 3.8.

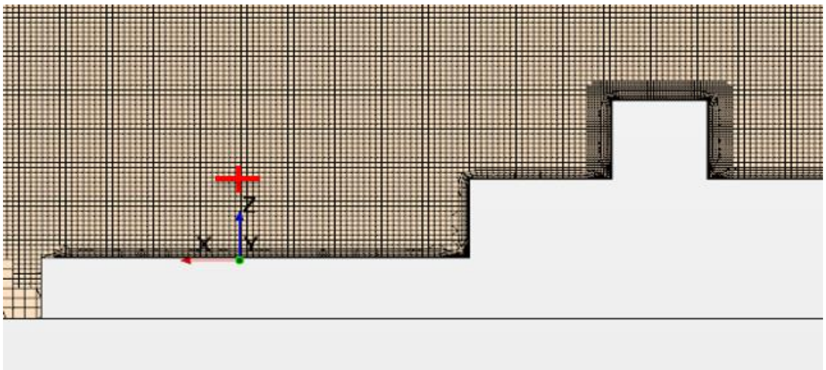


Figure 3.6 Measurement point over the flight deck

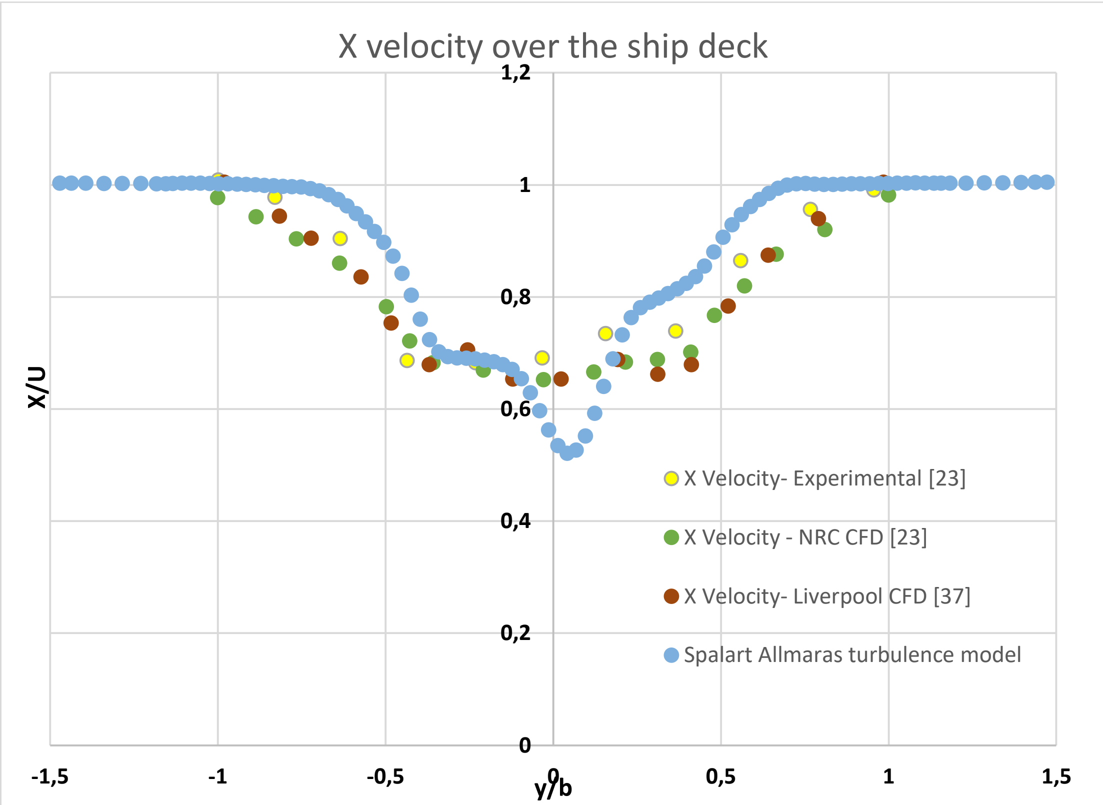


Figure 3.7 X velocity comparison between literature and present study

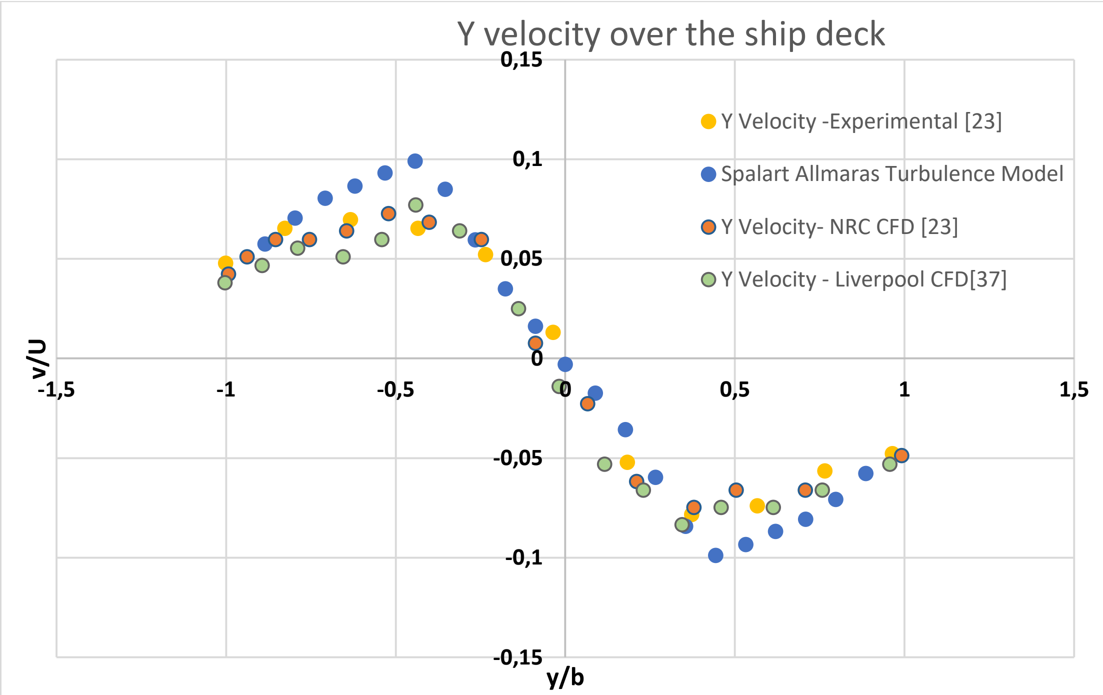


Figure 3.8 Y velocity comparison between literature and present study

3.4 Effects of Turbulence Models

The computational fluid dynamics methods consist of model geometry, grid structure, physical model and post-process. Since the geometry and grid structure cannot be changed, the physical model has a significant effect on the simulations. It is important to choose the turbulence model which ensures more accurate data to experimental results [57]. In this study three turbulence models were compared which are k-ε, k-ω and Spalart-Allmaras model. k-ε model is widely used to solve CFD problems [58]. It is a two-equation turbulence model and uses to solve far areas from the wall. This model has more accurate results to solve turbulent flows and eddies in the flow. k-ω model is a similar two-equation turbulence model and uses to solve the near the wall [59]. Spalart-Allmaras model is one equation turbulence model and generally is preferred to solve aerodynamic problems [60]. The comparison of different turbulence models and previous literature study data can be seen in Figure 3.9.

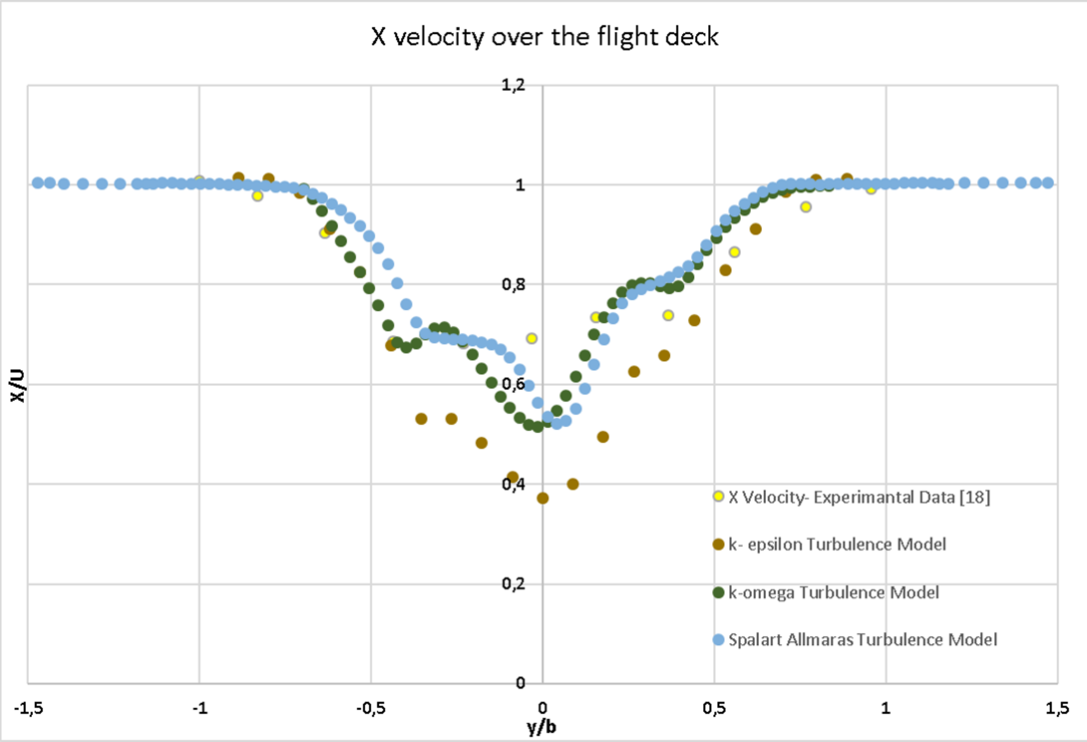


Figure 3.9 The comparison of different turbulence models at the midpoint of hangar deck

4
PARAMETRIC STUDY

4.1 Introduction

The frigate shape forms aerodynamic characteristic of the ship. The geometric specifications direct the air around the body and it directly affects the velocity, pressure and turbulence features in the critical areas. The constructed ship shapes give the chance of investigating and mapping ship air wake region and amending by making geometrical changes. The obtained results give to perspective to understand air behavior between ship and helicopter. It ensures to construct better aerodynamically superstructure ships in the design process.

In this study, the open-source geometry SFS2 was chosen to investigate various parametric effects on the ship air wake. As it is discussed in Chapter 3, the validation study was performed to validate the constructed CFD model. After comparison these obtained data with previous literature results, the verification study was carried out. In this part, the mesh cell sizes and turbulence model effects on the simulation were observed. Also, grid dependence analyses were performed to compare the convergence of velocity results.

The simulations were performed by using commercial CFD software Star CCM+ in the University of Strathclyde archives. The turbulence effects were modeled by using the Spalart-Allmaras turbulence model which is especially used in aerospace problems to solve the transport equation for kinematic turbulence viscosity. The Reynolds Averaged Navier Stokes method was used to solve the turbulent flow region around the SFS2 body. The SFS2 body is modeled in a 1:100 scale with the rectangular domain, by using structured block mesh which has 2.9millions of cells. The base size is 0.015m and all regions are the percentage of base size. To solve critical flow regions better, there is a smaller cell size around the funnel and hangar area.

4.2 Effect of Hangar Edge Angle

The chamfer dimensions were determined as x/h and y/h ratios which x-axis is elongated during ship length and y-axis are along ship beam, `h` is defined as hangar height. These ratios were based on Kaaria`s study, y/h ratio was assumed constant, only x/h ratio was changed and the angle effect was examined[21]. The detailed chamfer design is shown in figure 4.1 and design parameters are given in table 4.1.

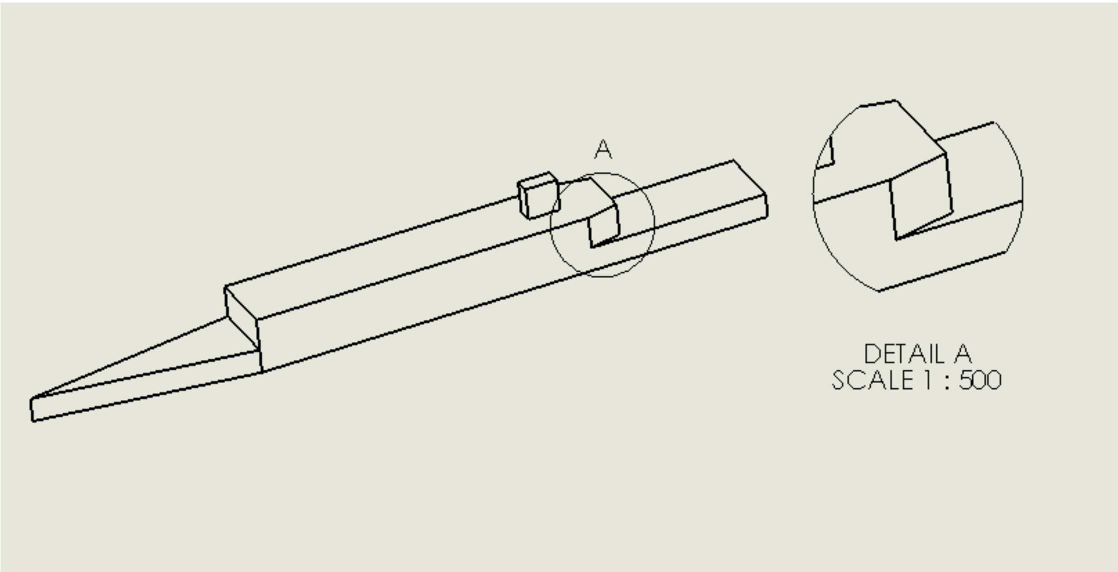


Figure 4.1 SFS2 geometry with chamfer design

Table 4.1 Chamfer geometry dimensions

	x/h	y/h	Angle ,α	h(feet)
Chamfer 1	1	0,25	$14{,}03^0$	20
Chamfer 2	0,75	0,25	$18{,}43^0$	20

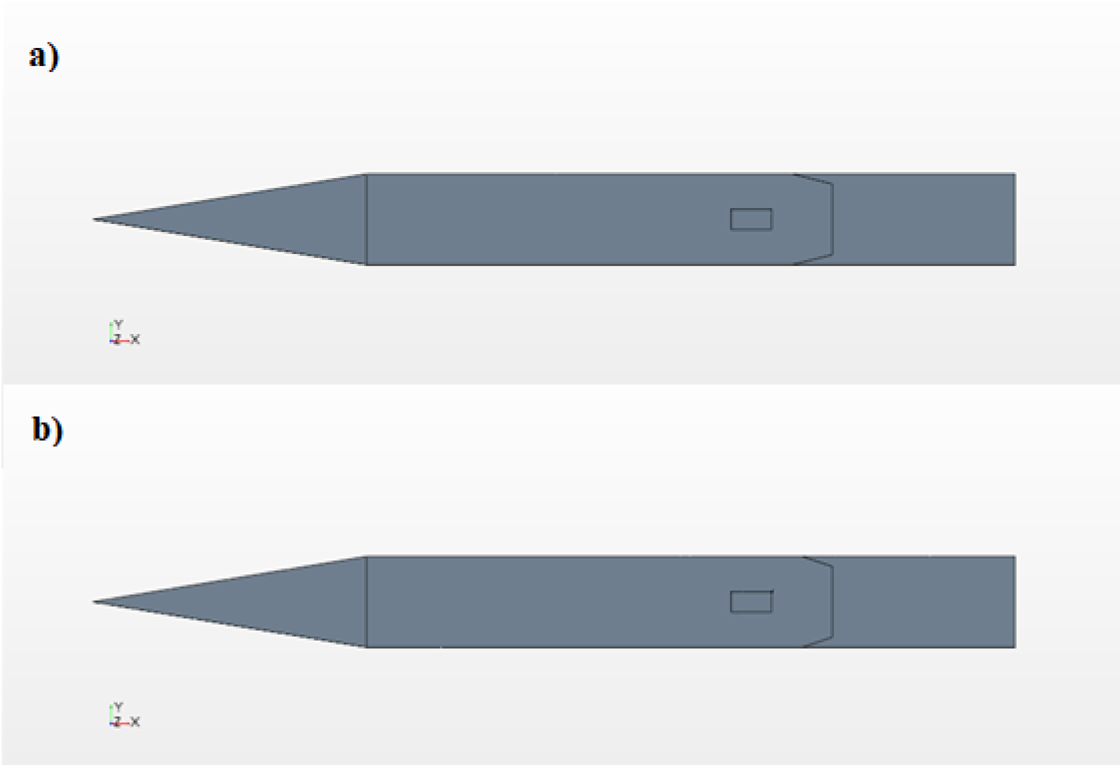


Figure 4.2 Chamfer 1 (a) and Chamfer 2 (b) top view

The simulations were performed for three different headwind conditions. Both X and Y velocity changes were compared with original geometry. It is seen that with increasing headwind speed X velocity is decreasing over the flight deck for both chamfer designs. However, for chamfer 2, the change of speed is seen as almost stable after 40 knots headwind.

The vorticity defines the velocity rotations which make instability in the flow region. The chamfer design effect on the vorticity and velocity in the flow region was shown in the scalar scenes below.

It is obviously seen that the chamfer design causes to decrease in vorticity area over the flight deck which has seen in figures 4.3, 4.5 and 4.7. The reduce on the vorticity is higher at Chamfer 1, however Chamfer 2 design shows increasing angle causes to increase vorticity again.

The velocity rotations can be seen for different headwind speeds in Figures 4.4, 4.6 and 4.8. The flow region after hangar edges has highly velocity rotations which have adverse effects on the helicopters. Chamfer 1 design significantly reduces the velocity rotations at the first mid of the hangar deck. But Chamfer 2 design makes the velocity rotations increase again.

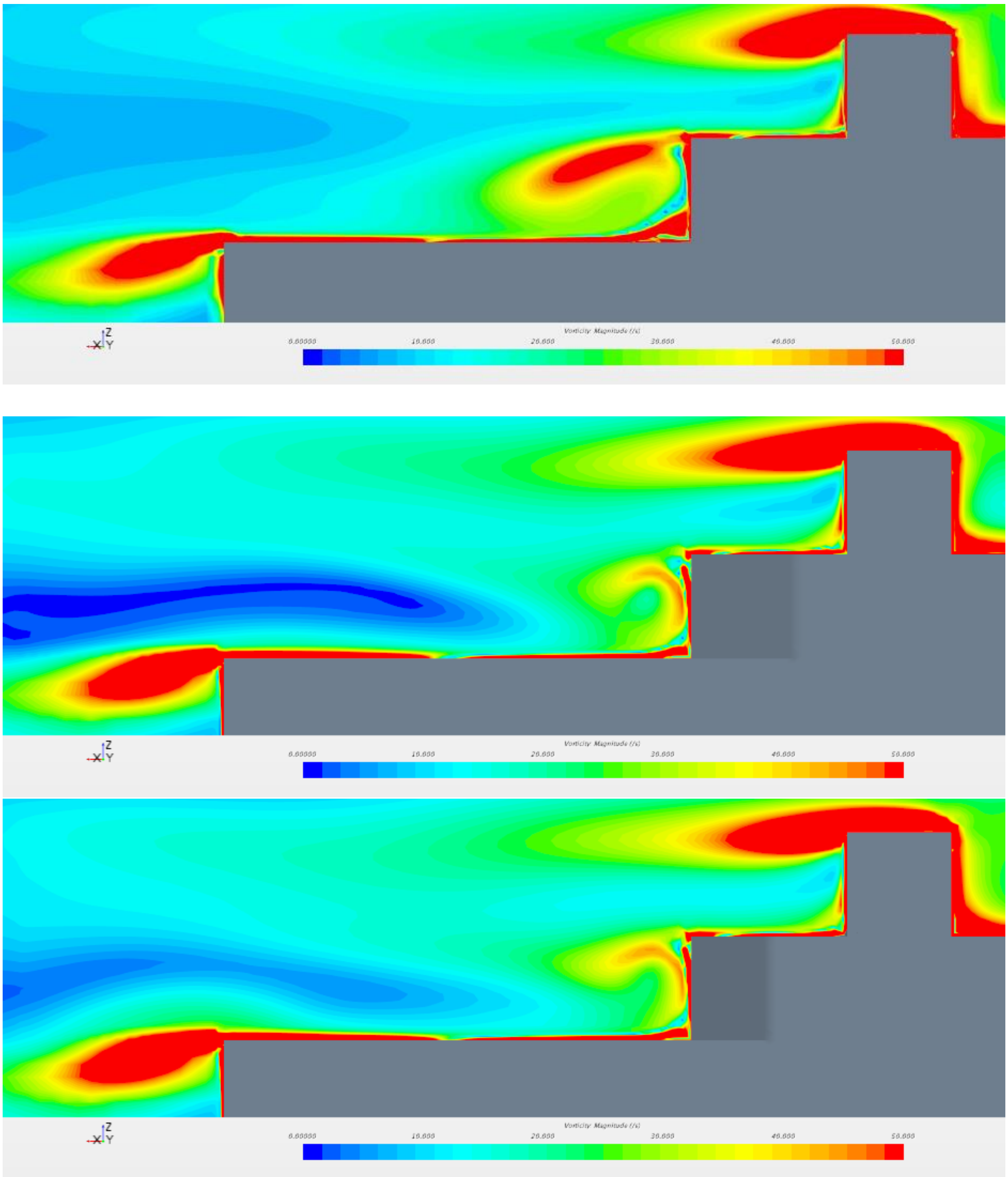


Figure 4.3 Vorticity comparison at 30 knots headwind for original geometry (top), chamfer 1 (middle) and chamfer 2 (bottom)

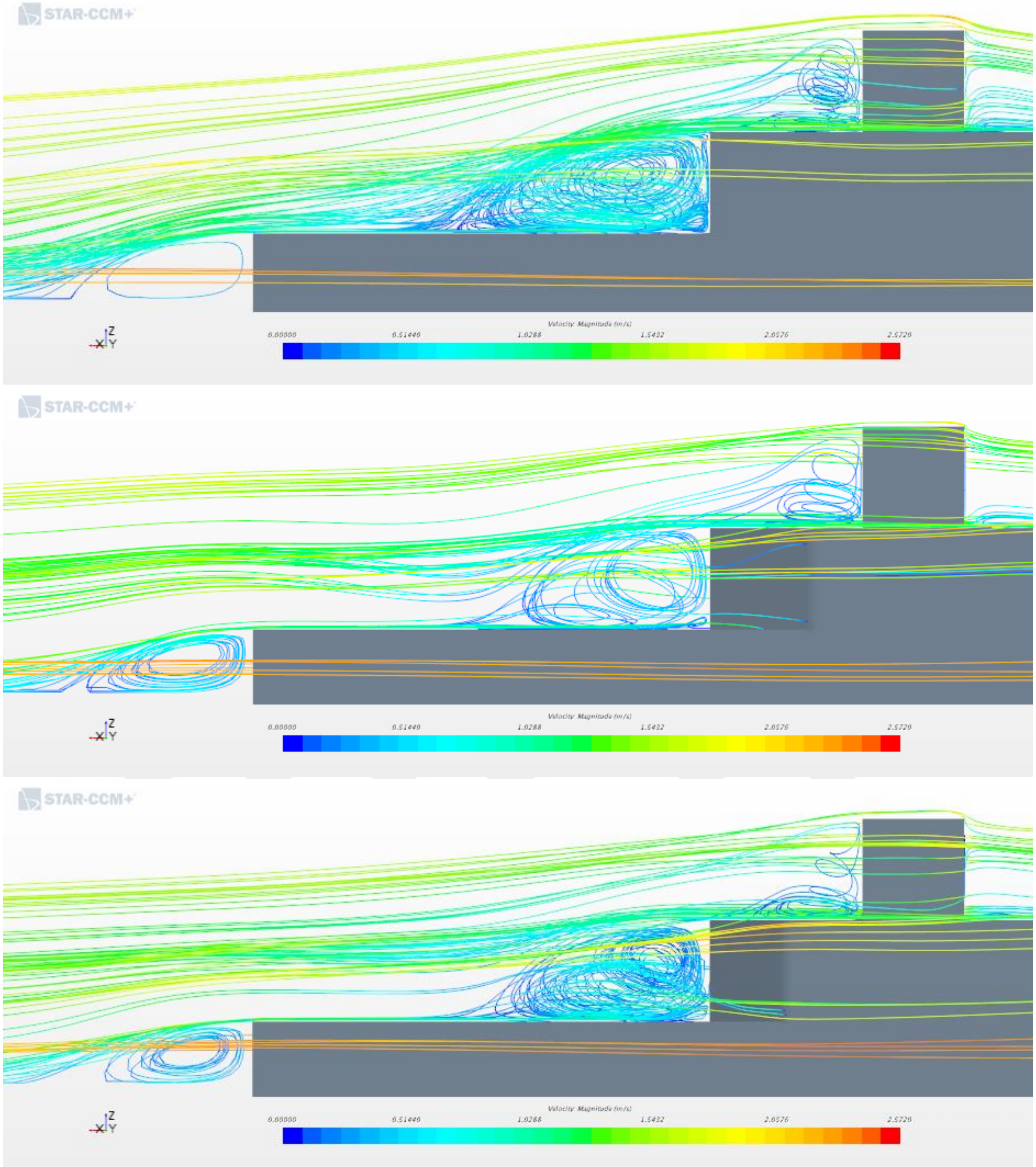


Figure 4.4 Velocity streamlines comparison at 30 knots headwind for original geometry (top), chamfer 1 (middle) and chamfer 2 (bottom)

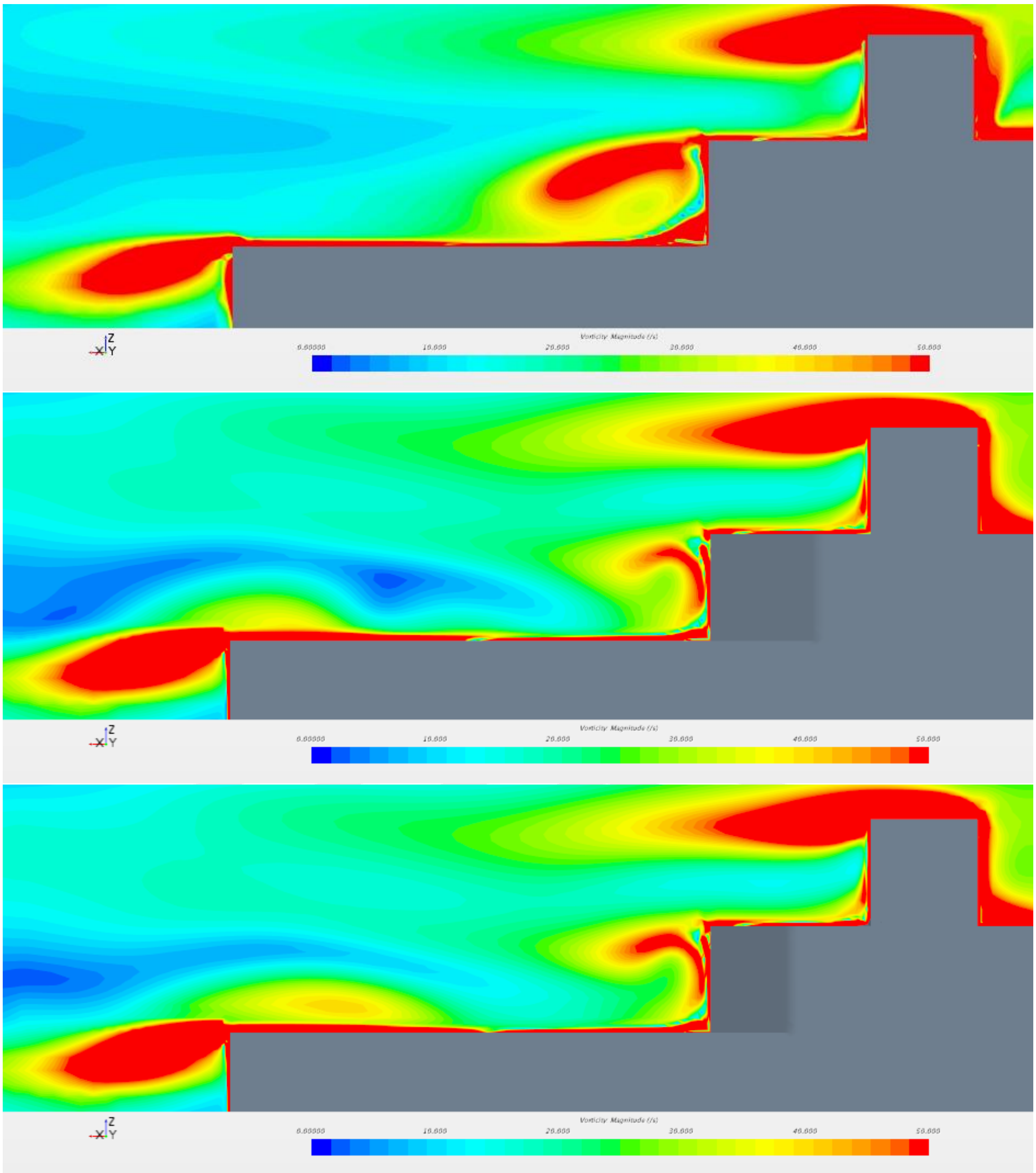


Figure 4.5 Vorticity comparison at 40 knots headwind for original geometry (top), chamfer 1 (middle) and chamfer 2 (bottom)

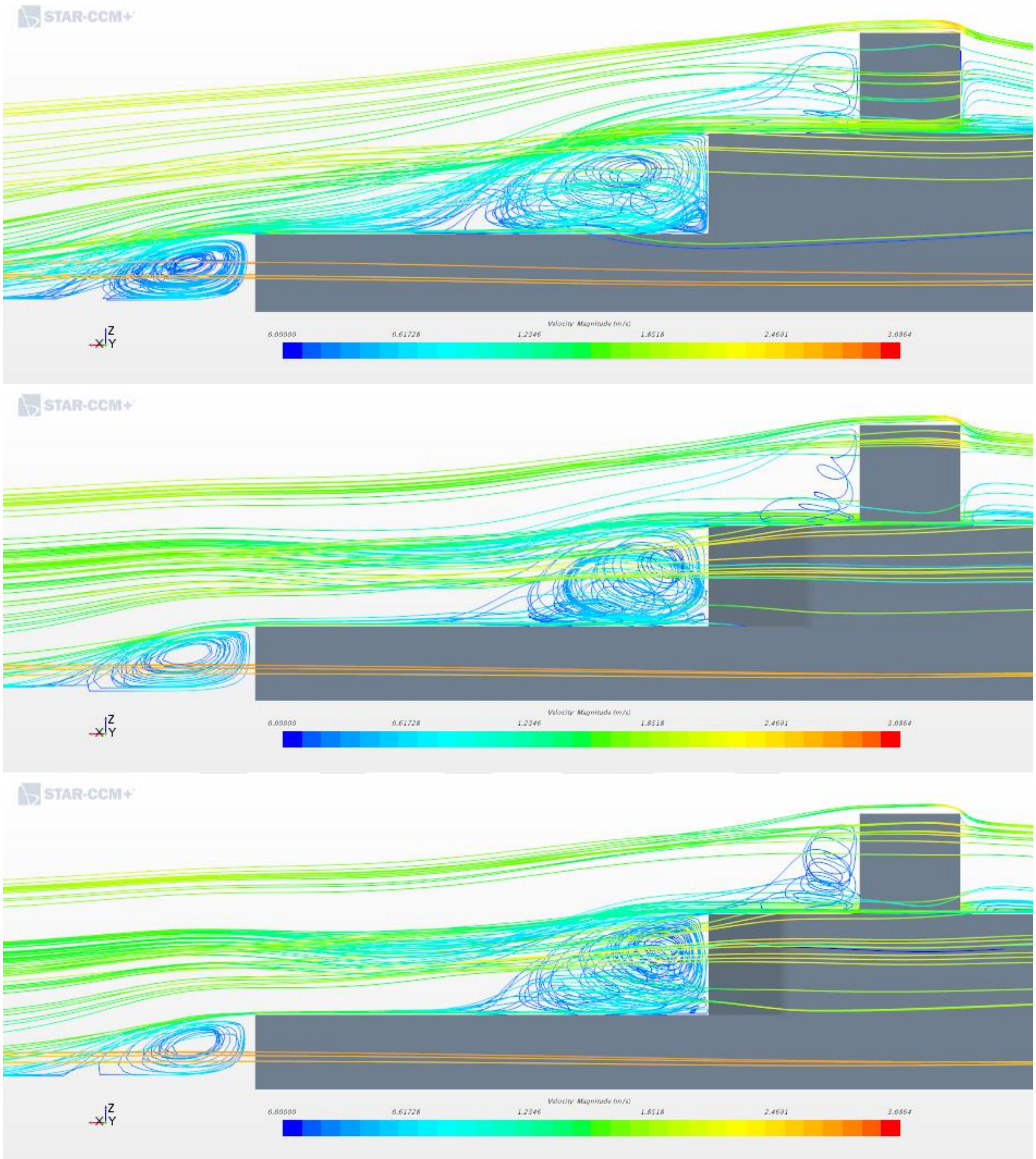


Figure 4.6 Velocity streamlines comparison at 40 knots headwind for original geometry (top), chamfer 1 (middle) and chamfer 2 (bottom)

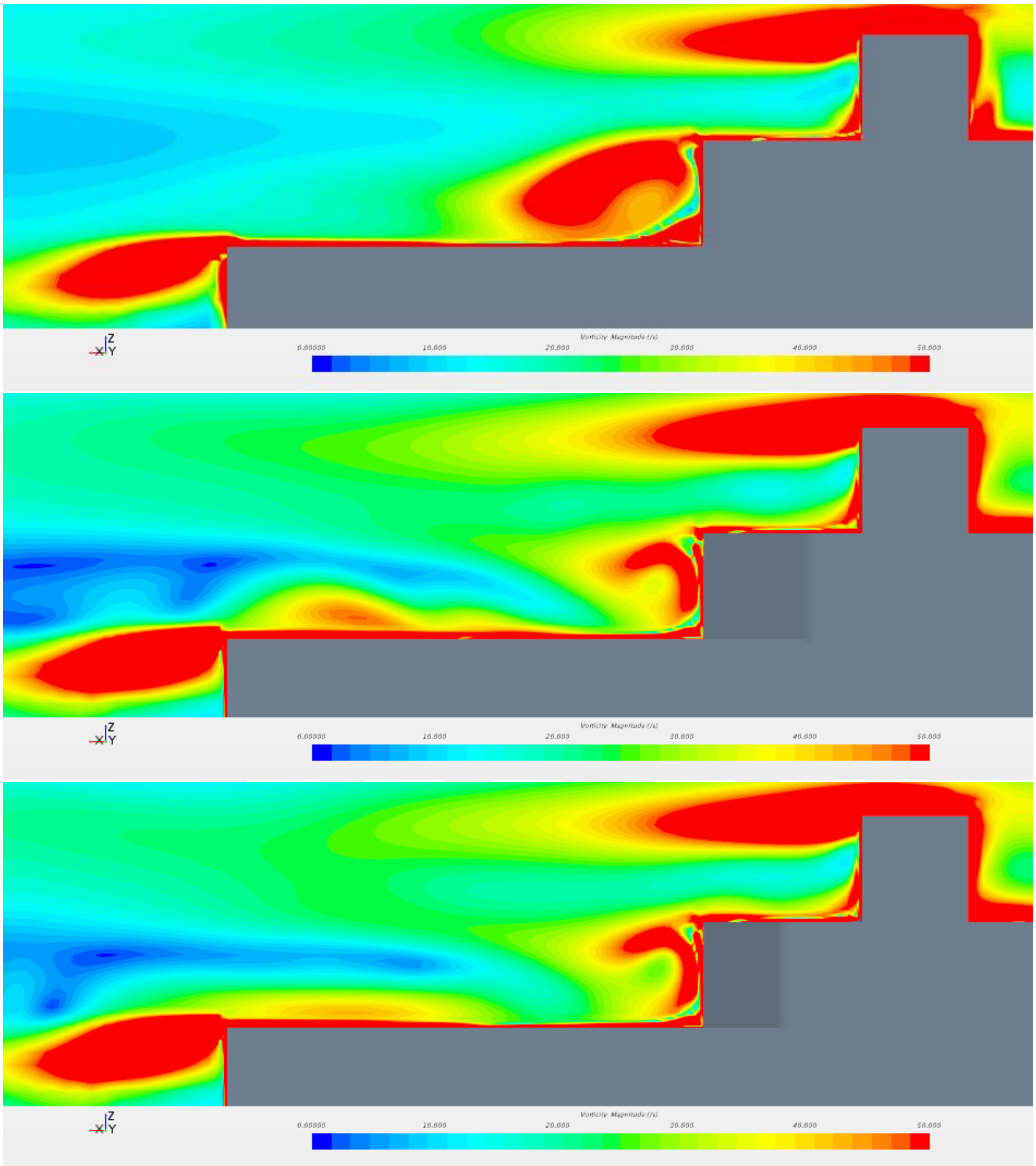


Figure 4.7 Vorticity comparison at 50 knots headwind for original geometry (top), chamfer 1 (middle) and chamfer 2 (bottom)

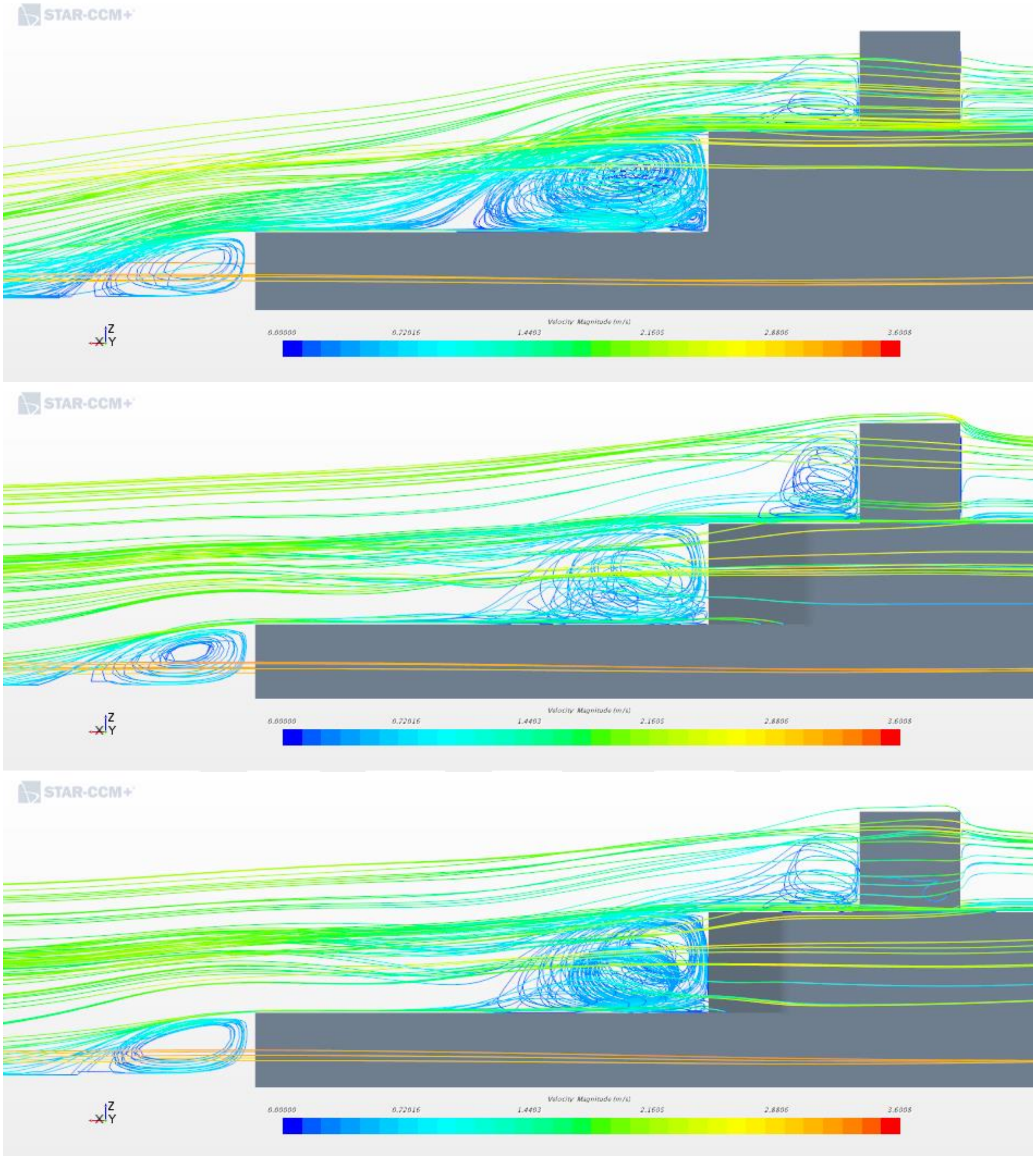


Figure 4.8 Velocity streamlines comparison at 50 knots headwind for original geometry (top), chamfer 1 (middle) and chamfer 2 (bottom)

4.3 Effect of Hangar Height

To based on literature review, it is seen that the hangar height is a significant parameter which has a direct effect on flight deck since it directs the airflow. To investigate the influence of hangar height, two different hangar height was set as hangar 1 and hangar 2. The simulations were performed and compared with original SFS2 geometry. The design parameters for constituted hangar heights can be seen in Table 4.2.

Table 4.2 SFS2 dimensions with different hangar heights

	Percentage	Height(feet)
Original Hangar Height	100%	20
Hangar 1	+10%	22
Hangar 2	-10%	18

Figure 4.9 Hangar 1 (a) and Hangar 2 (b) side view

It is seen that the vorticity character does not change with hangar height but it expands through hangar deck in figures 4.10,4.12 and 4.14. Higher hangar height makes the headwind velocity effects slow and slightly reduces the vorticity area in front of the hangar. With increasing headwind speed, the vorticity region expands dramatically in hangar 2.

The velocity streamlines in figures below show the increasing hangar height causes the lower speed in front of the hangar and decreasing hangar height causes a higher speed region over the deck which is undesired flow characteristic for flight operations.

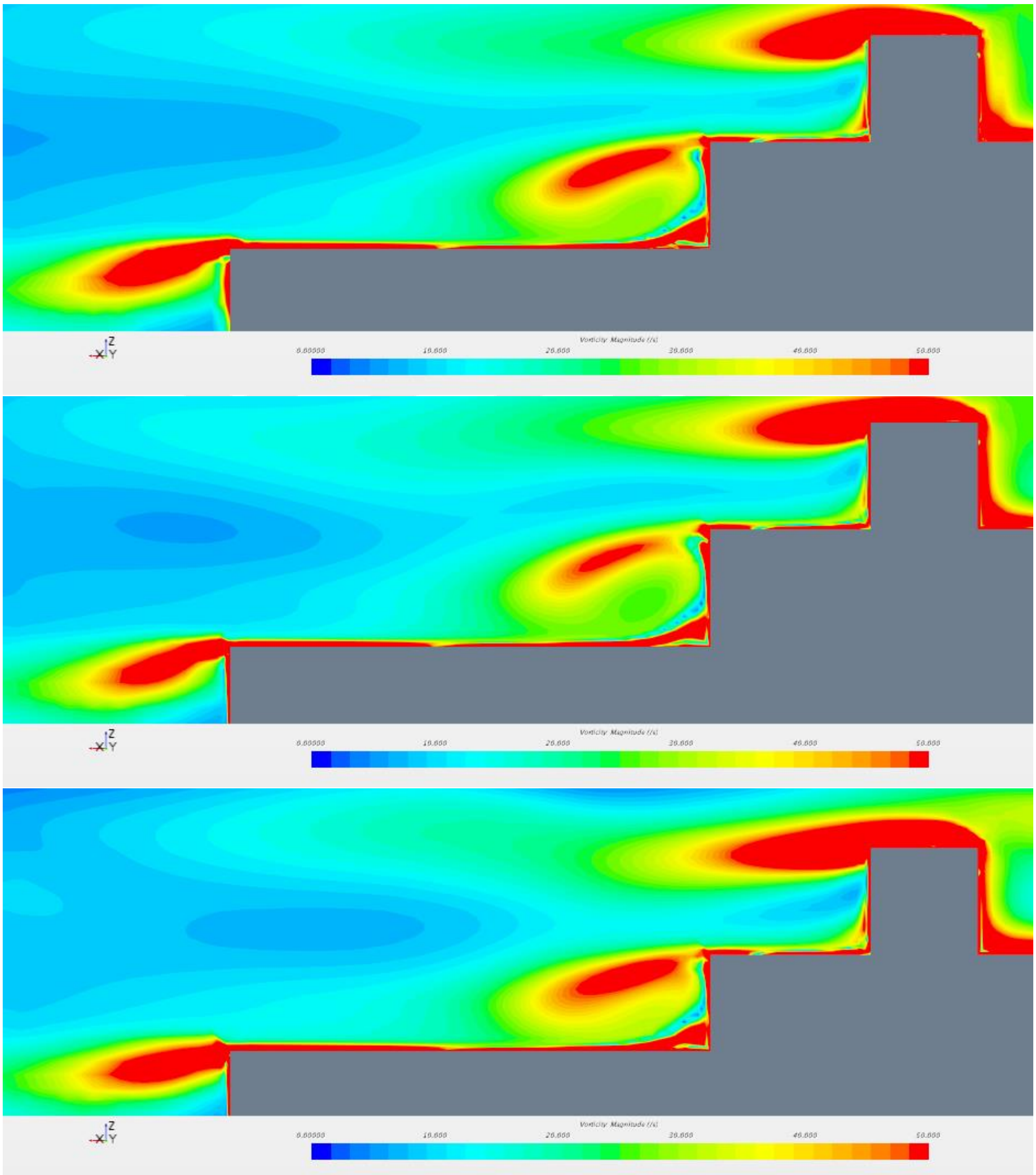


Figure 4.10 Vorticity comparison at 30 knots headwind for original geometry (top), hangar 1 (middle) and hangar 2 (bottom)

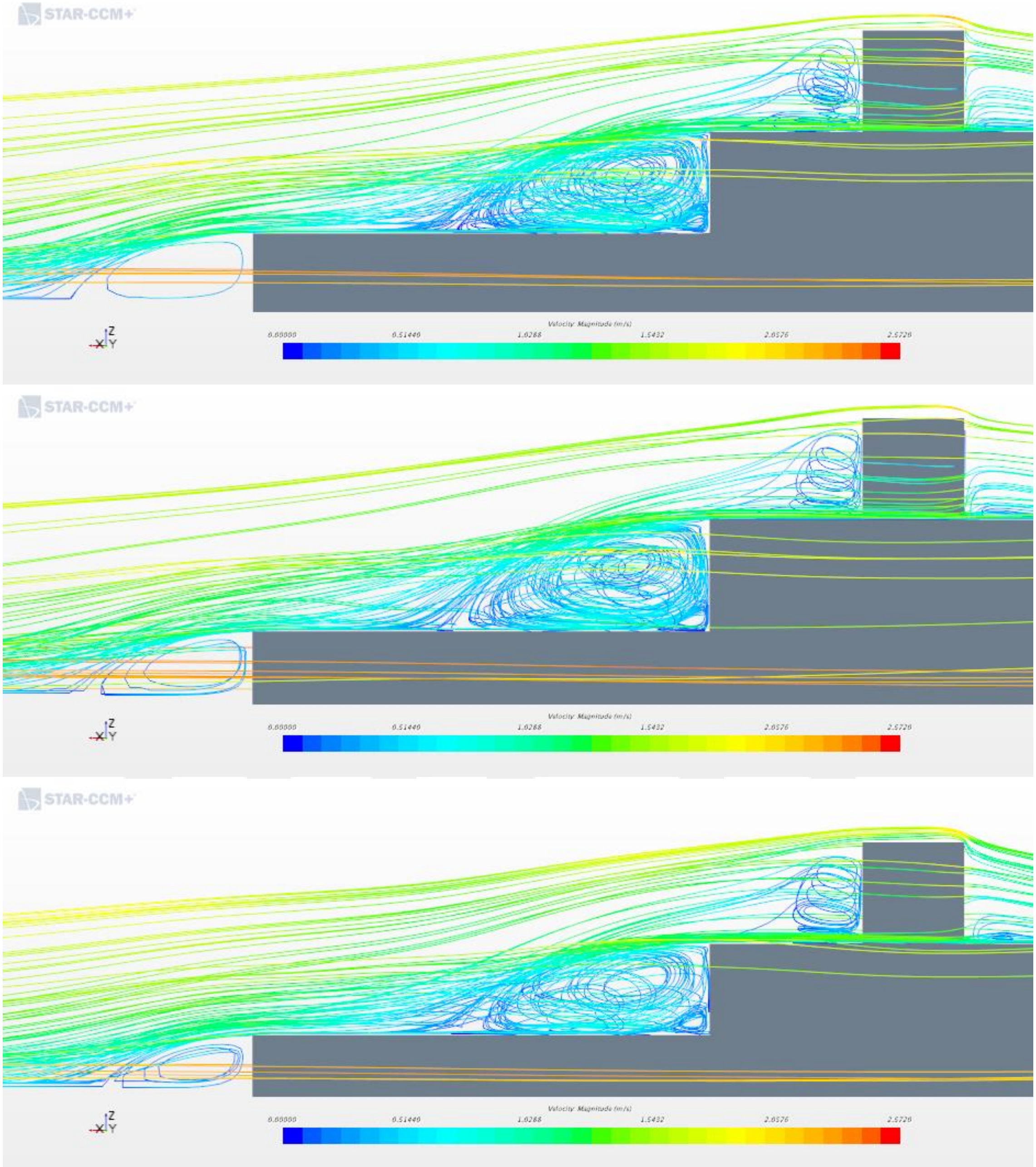


Figure 4.11 Velocity streamlines comparison at 30 knots headwind for original geometry (top), hangar 1 (middle) and hangar 2 (bottom)

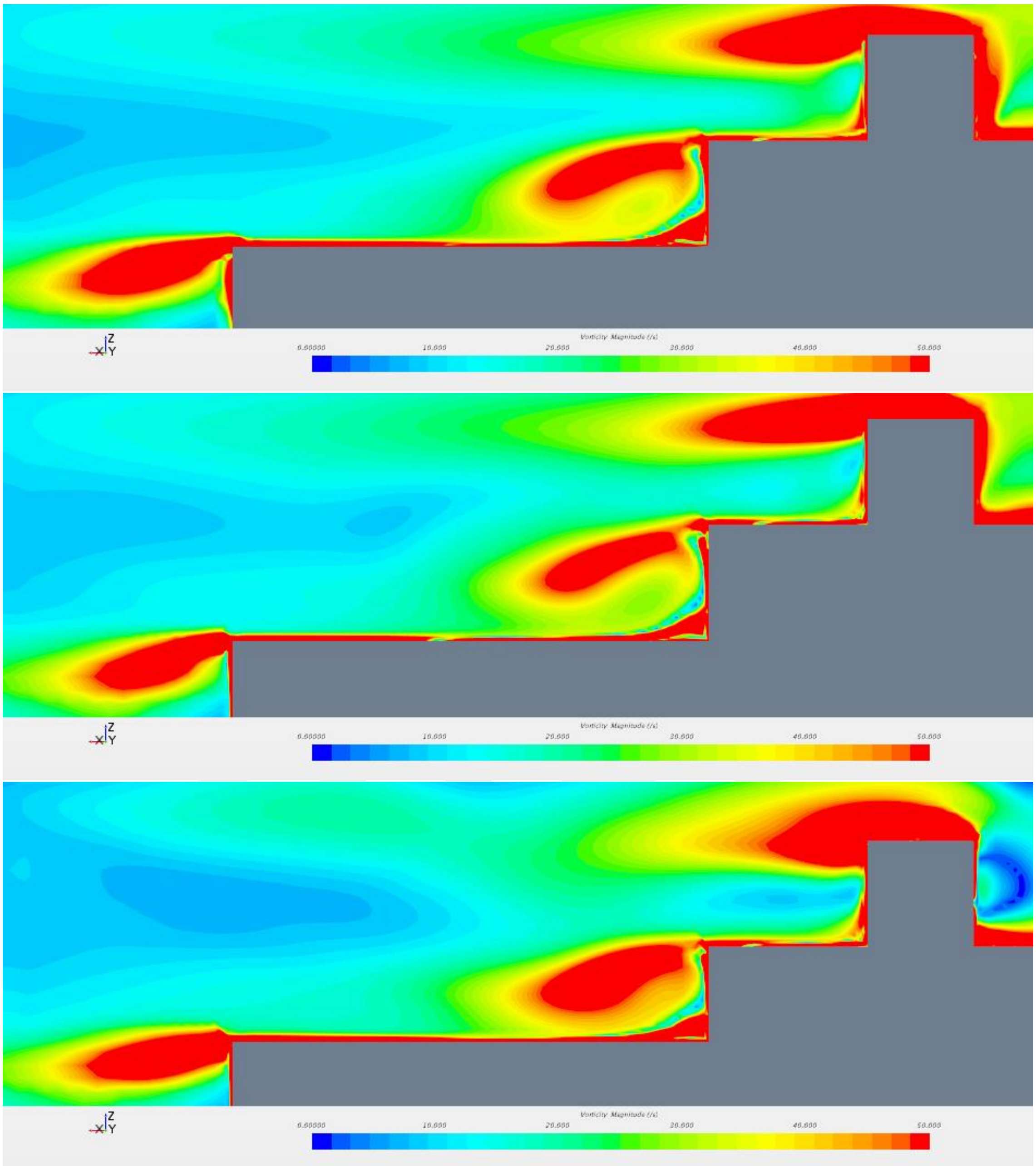


Figure 4.12 Vorticity comparison at 40 knots headwind for original geometry (top), hangar 1 (middle) and hangar 2 (bottom)

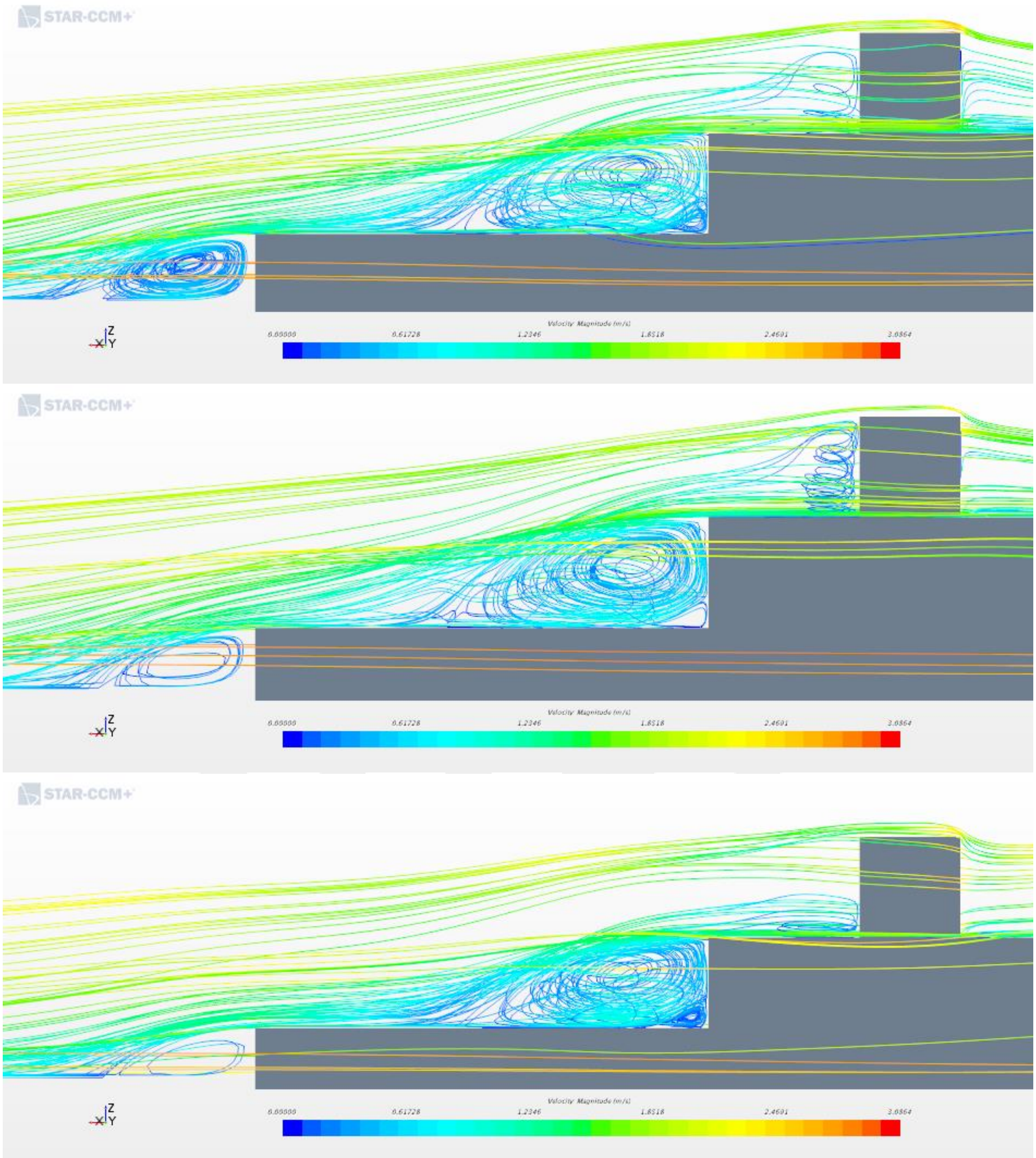


Figure 4.13 Velocity streamlines comparison at 40 knots headwind for original geometry (top), hangar 1 (middle) and hangar 2 (bottom)

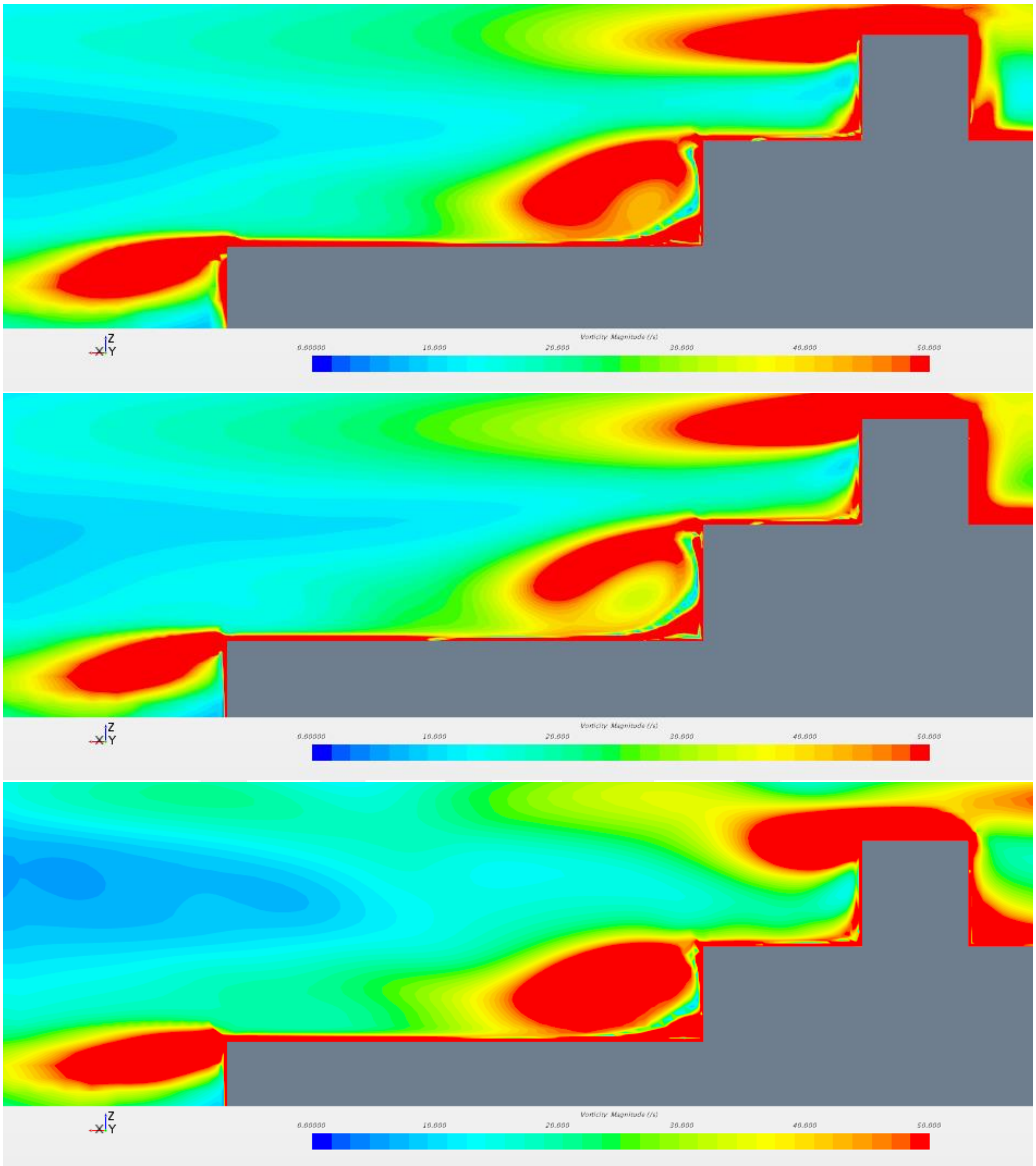


Figure 4.14 Vorticity comparison at 50 knots headwind for original geometry (top), hangar 1 (middle) and hangar 2 (bottom)

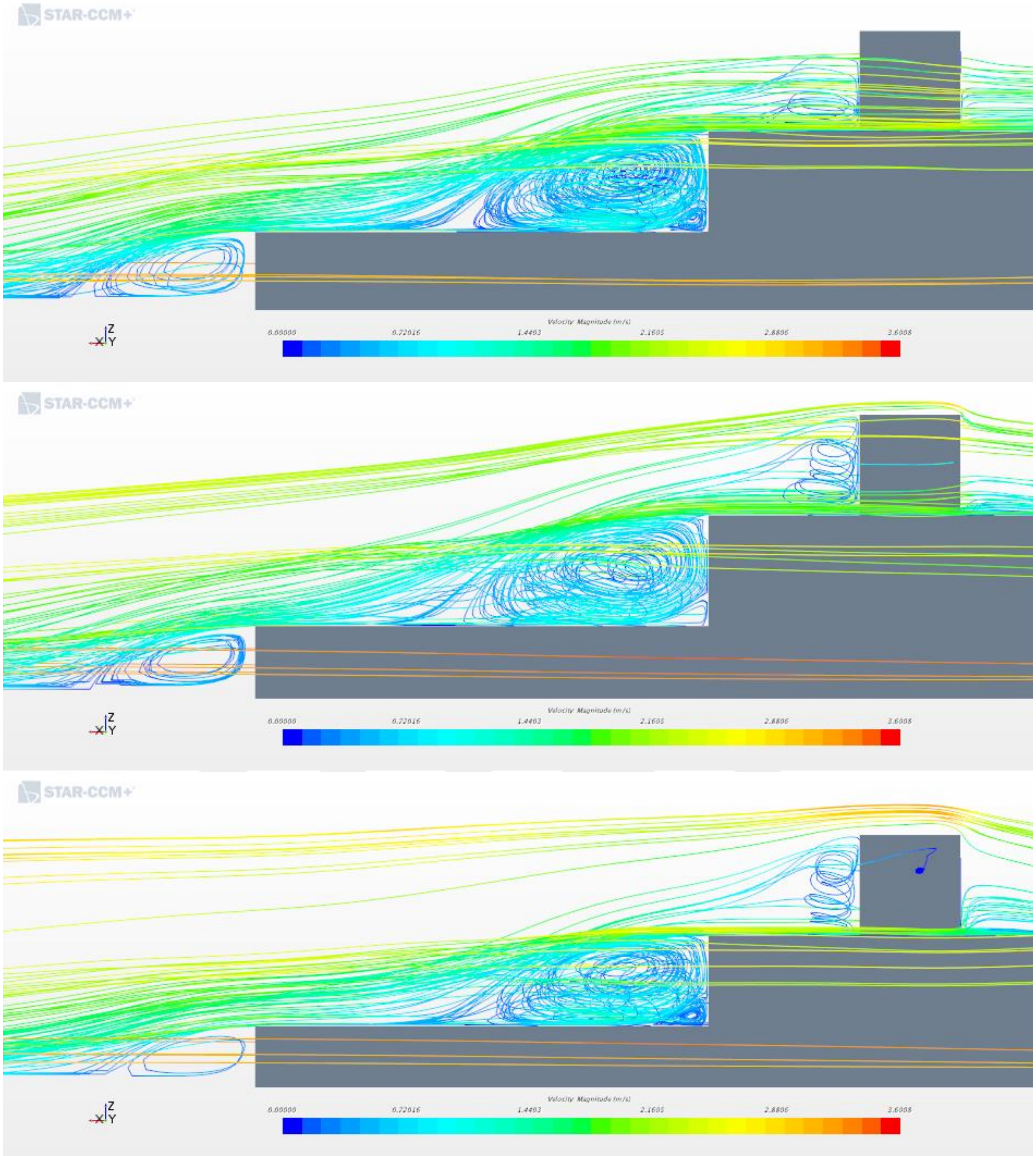


Figure 4.15 Velocity streamlines comparison at 50 knots headwind for original geometry (top), hangar 1 (middle) and hangar 2 (bottom)

Y velocity character is nearly the same for hangar 1 design with original geometry in figures 4.19 and 4.20 but hangar 2 geometry reduces Y velocity. With increasing velocity, both hangar 1 and hangar 2 design give closer velocity values with original geometry in figure 4.21. In figure 4.19, the chamfer 1 design has very low Y velocity values comparing by chamfer 2. By increased velocity, the negative velocity regions start to see in chamfer 2 design in Figures 4.20 and 4.21.

The highest X velocity gradient is seen for hangar 2 design in figures 4.19, 4.20 and 4.21. Also, hangar 1 design has very close velocity values with the original geometry. Both chamfer designs decrease the X values. The difference between them looks so smaller in the graphs but it can be said the velocity values are lower in chamfer 1 design at 30 and 40 knots.

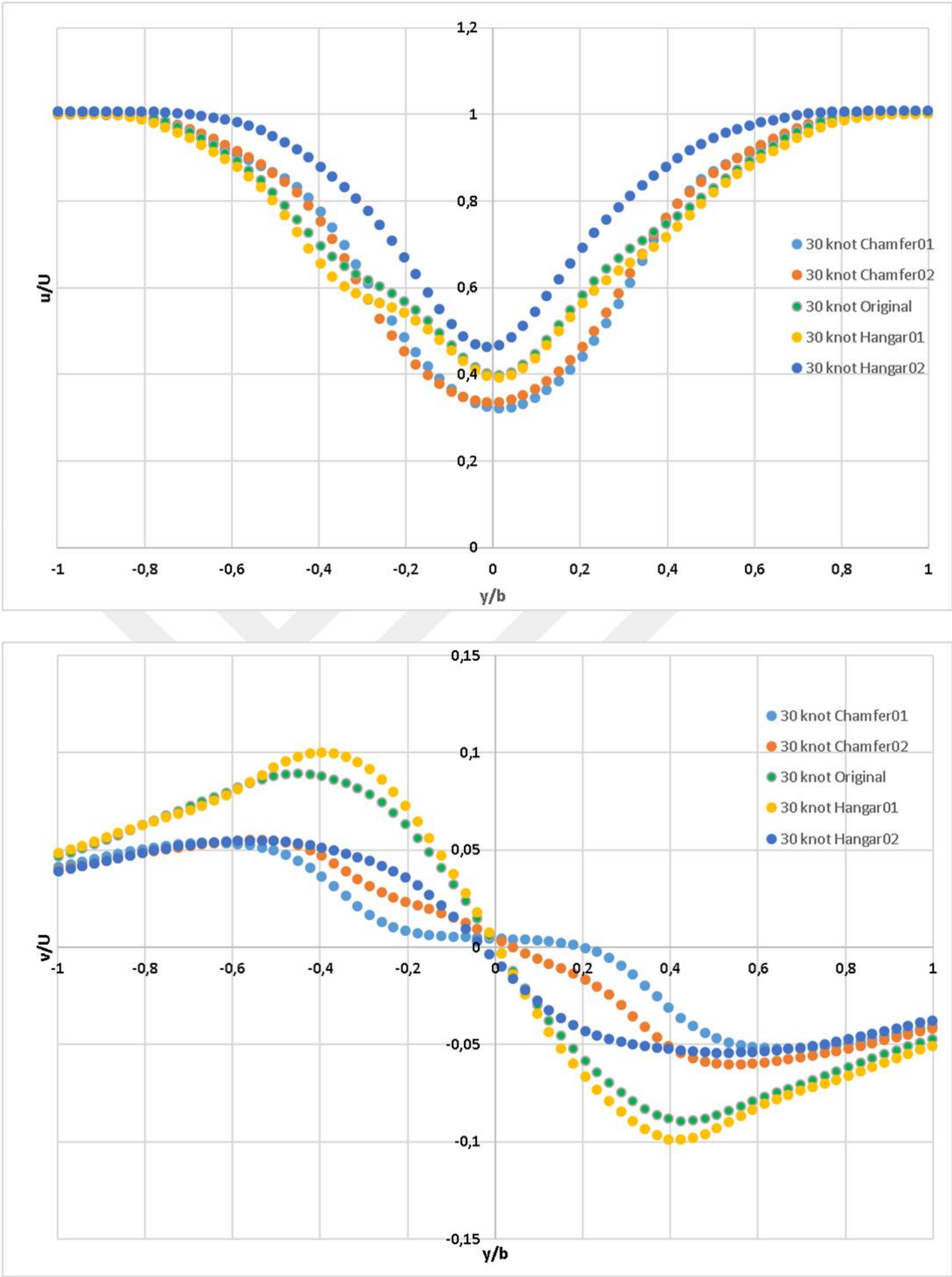
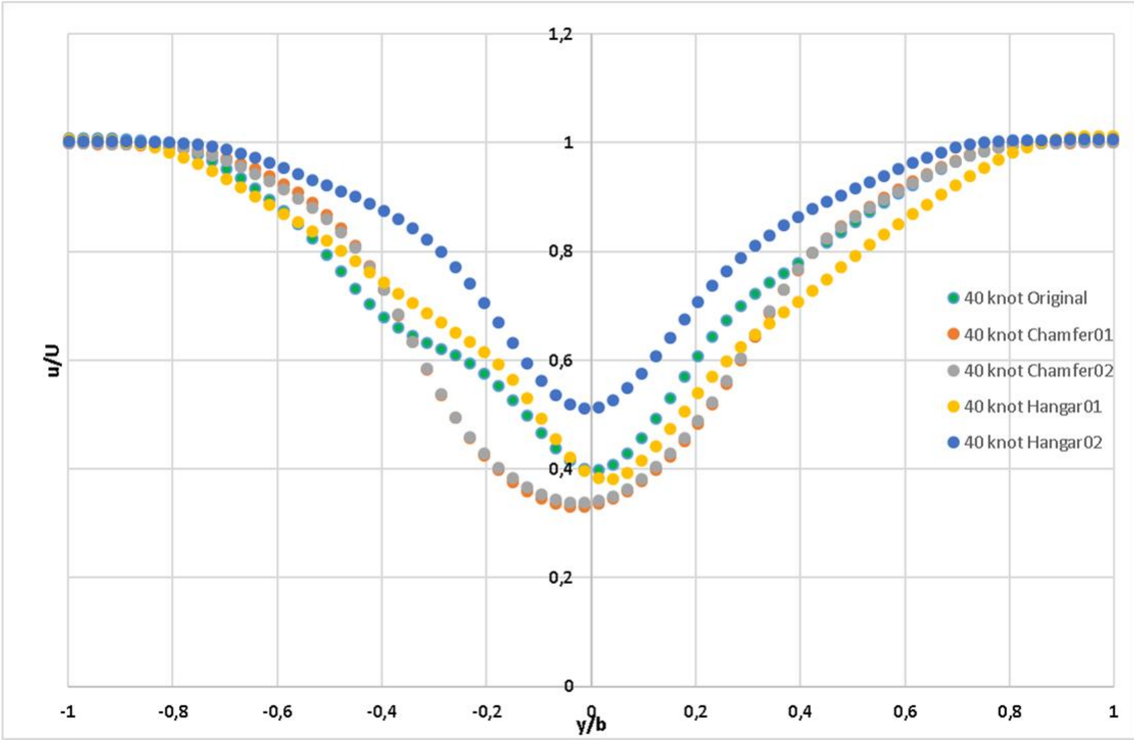


Figure 4.16 X velocity (top) and Y velocity (bottom) comparisons for hangar edges and hangar heights with original geometry at the 30-knot headwind



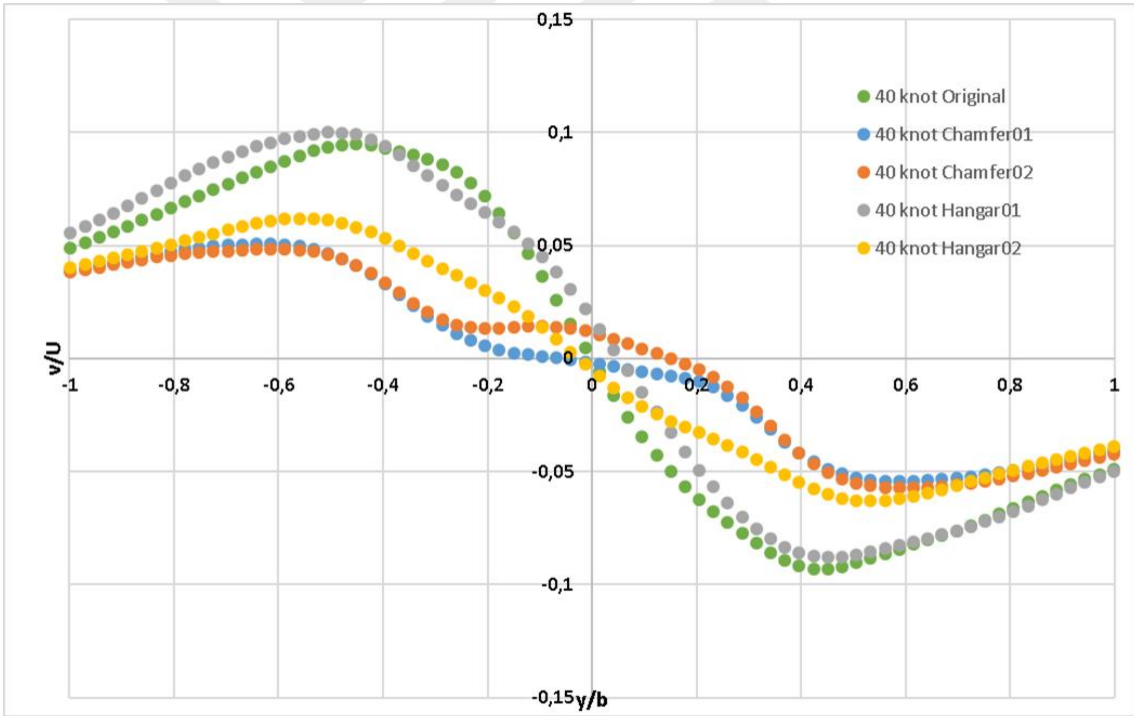


Figure 4.17 X velocity (top) and Y velocity (bottom) comparisons for hangar edges and hangar heights with original geometry at the 40-knot headwind

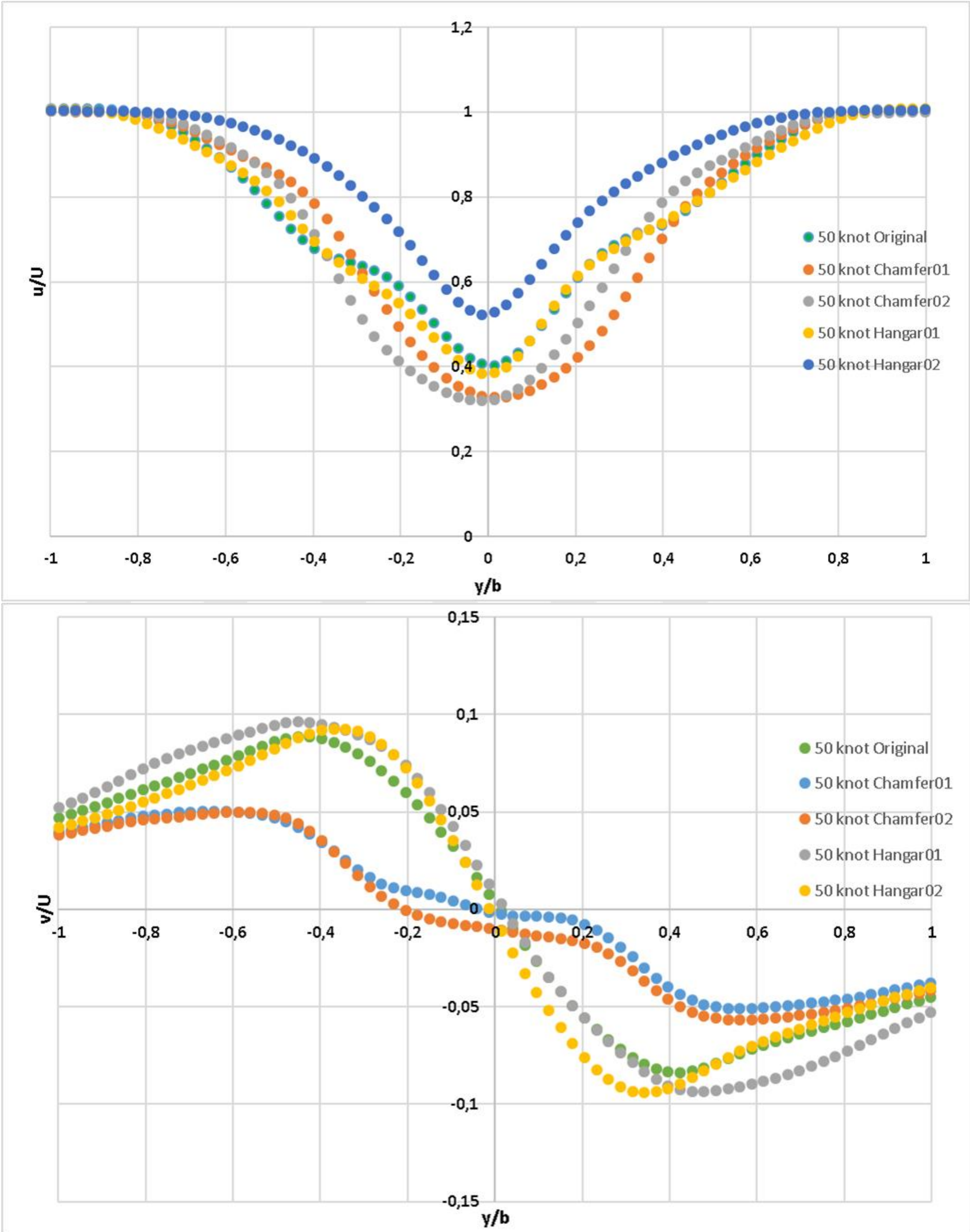


Figure 4.18 X velocity (top) and Y velocity (bottom) comparisons for hangar edges and hangar heights with original geometry at the 50-knot headwind

5
RESULTS AND DISCUSSION

The ship air wake directly affects pilot flight experience and determines fuel consumption of helicopters, also it can be dangerous if the turbulence level is higher than safe limits. To decrease turbulence and velocity around the flight deck, the current situation was analyzed as the first step. The velocity and vorticity change were mapped using commercial software Star CCM+. The previous studies were examined to determine significant parameters on ship air wake. It was seen that ship-helicopter studies were conducted as with regarding free surface or one phase. Moreover, the ship motions and helicopter rotor forces and momentums can be taken into account while the calculation process. In this study, the simulations were performed in one phase case (air). The environment effects (sea state) and ship motions (heave, pitch, and roll) were neglected to simplify problems due to limited time and computer processor limits. The previous studies made in this area were performed both mapping and flight simulator to investigate flight experiences. It is known that the turbulence limit is important to control helicopter since human consciousness is impressed dramatically by the change of velocity and turbulence. It makes hard to control helicopter such as helideck, which has a very limited area by comparing to large airport landing areas. If the ship movements due to wind and waves are thought, since they cannot control by a human it is focused on the change of ship design to solve this problem. The turbulence area can be solved by using Detached Eddy Simulations to catch small eddies better. It needs very small time steps when it is compared with the Reynolds Averaged Navier Stokes method. It needs more time for calculation and processor capacity. Depending on these reasons, the RANS method was used to solve the flow region of the ship air wake in this study.

By comparing turbulence models, it is seen that each model has own characteristics and different error values. Also, the RANS model is not sensitive as Detached Eddy Simulation as it can be seen in results. However, the RANS model ensures results faster which are critical in limited time and needs reasonably lower processor capacity. The Spalart-Allmaras model gives more accurate results when compared with the other turbulence models. It is also known that this model is preferred in aerodynamic researches. Moreover, the grid dependence tests were performed to understand mesh cell number effects on the accuracy of the results. By comparing chamfer 1 and chamfer 2 designs, the chamfer 1 design ensures the lowest velocity and vorticity over the flight deck. Increasing angle causes to increase the velocity values slightly. Hangar 1 geometry makes the velocity slower over the deck but does not decrease the vorticity area. Hangar 2 geometry increases both vorticity and velocity values over the flight deck. It means by decreasing hangar height, the instability over the deck also increases. Also, by increasing hangar height, the flow slows down but it does not amend to ensure low turbulence area.

To sum up, it is seen that the design changes in frigate superstructure effects directly the ship air wake which has a critical role in helicopter operations. By changing hangar height, it is seen the pressure and velocity changes are dramatically changed. By making chamfer design on hangar edges, the ship air wake turbulent characteristic area can be decreased but it is seen that the angle value has crucial importance at this point.

The ship motions can be considered as a future work for this study. With free surface effects, more sensible methods can be used to solve flow region such as LES and DES methods.

REFERENCES

[1] "Frigate - Wikipedia." [Online]. Available: https://en.wikipedia.org/wiki/Frigate. [Accessed: 31-Dec-2019].

[2] *Turkish Naval Forces.* .

[3] "List of active ships of the Turkish Naval Forces - Wikipedia." [Online]. Available: https://en.wikipedia.org/wiki/List_of_active_ships_of_the_Turkish_Naval_Forces. [Accessed: 31-Dec-2019].

[4] "Barbaros sınıfı fırkateyn - Vikipedi." [Online]. Available: https://tr.wikipedia.org/wiki/Barbaros_sınıfı_fırkateyn. [Accessed: 24-Feb-2020].

[5] North Atlantic Treaty Organization. Research and Technology Organization. Applied Vehicle Technology Panel. and North Atlantic Treaty Organization. Advisory Group for Aerospace Research and Development. Fluid Dynamics Panel., *Fluid dynamics problems of vehicles operating near or in the air-sea interface = Problèmes de dynamique des fluides des véhicules évoluant dans ouprès de l'interface air-mer.* North Atlantic Treaty Organization, Research and Technology Organization, 1999.

[6] J. S. Forrest, S. J. Hodge, I. Owen, and G. D. Padfield, "Towards fully simulated ship-helicopter operating limits: The importance of ship airwake fidelity," *Annu. Forum Proc. - AHS Int.*, vol. 1, pp. 339–351, 2008.

[7] J. Buchholz, J. Martin, A. F. Krebill, G. M. Dooley, and P. Carrica, "Structure of a Ship Airwake at Model and Full Scale," 2018.

[8] W. Blendermann, "Parameter identification of wind loads on ships," 1994.

[9] M. R. Haddara and G. Soares, "Wind loads on marine structures," 1999.

[10] M. M. S. P. Tattersall, C.M. Albone, "Prediction of Ship Air wakes over Flight Decks Using CFD," 1998.

[11] T. Van Nguyen, I. Watanabe, S. Miyake, N. Shimizu, Y. Ikeda, and T. Hyogo, "A Study on Reduction of Air Resistance Acting on a Large Container Ship Development of High Lift Rudder View project Development of Air Cavity Tank View project A Study on Reduction of Air Resistance acting on a Large Container Ship," 2016.

[12] T. Van Nguyen, N. Shimizu, A. Kinugawa, and Y. Tai, "Development of Gap-Flow-Protectors to Reduce Air Resistance acting on Deck Containers of a Ship Development of CFD Turbulence Models View project Development of High Lift Rudder View project Development of Practical Gap Covers to Reduce Air Resistance Acting on Deck Containers of a Ship," 2018.

[13] T. Van Nguyen, N. Shimizu, A. Kinugawa, Y. Tai, and Y. Ikeda, “Numerical Studies on Air Resistance Reduction Methods for a Large Container Ship with Fully Loaded Deck-Containers in Oblique Winds,” 2017.

[14] H. Majidian and F. Azarsina, “Numerical simulation of container ship in oblique winds to develop a wind resistance model based on statistical data,” *J. Int. Marit. Safety, Environ. Aff. Shipp.*, vol. 2, no. 2, pp. 67–88, Feb. 2019.

[15] Y. Kim, K.-S. Kim, S.-W. Jeong, S.-G. Jeong, S.-H. Van, and J. Kim, “Design and Performance Evaluation of Superstructure Modification for Air Drag Reduction of a Container Ship,” *J. Soc. Nav. Archit. Korea*, vol. 52, no. 1, pp. 8–18, Feb. 2015.

[16] I. M. V. Andersen, “Wind loads on post-panamax container ship,” *Ocean Eng.*, vol. 58, pp. 115–134, 2013.

[17] A. D. Wnek and C. Guedes Soares, “CFD assessment of the wind loads on an LNG carrier and floating platform models,” *Ocean Eng.*, vol. 97, pp. 30–36, Mar. 2015.

[18] S. Brizzolara and E. Rizzuto, “Wind Heeling Moments on Very Large Ships. Some Insights through CFD Results.”

[19] M. J. Yelland, B. I. Moat, R. W. Pascal, and D. I. Berry, “CFD Model Estimates of the Airflow Distortion over Research Ships and the Impact on Momentum Flux Measurements.”

[20] J. Wang, G. Jiang, and X. Wang, “Effect Analysis of the Hangar Rear Edge Curvature on the Ship Airwake Effect Analysis of the Hangar Rear Edge Curvature on the Ship Airwake,” 2018.

[21] C. H. Kääriä, “Determining the Impact of Hangar-Edge Modifications on,” no. May, 2010.

[22] J. S. Forrest, C. H. Kaaria, and I. Owen, “Evaluating ship superstructure aerodynamics for maritime helicopter operations through CFD and flight simulation,” *Aeronaut. J.*, vol. 120, no. 1232, pp. 1578–1603, Oct. 2016.

[23] W. Yuan, A. Wall, and R. Lee, “Combined numerical and experimental simulations of unsteady ship airwakes,” *Comput. Fluids*, vol. 172, pp. 29–53, Aug. 2018.

[24] C. H. Kääriä, J. S. Forrest, and I. Owen, “Using flight simulation to improve ship designs for helicopter operations,” *RINA, R. Inst. Nav. Archit. - Int. Conf. Comput. Appl. Shipbuild. 2011, Pap.*, vol. 2, no. January 2011, 2011.

[25] G. F. Syms, “Simulation of simplified-frigate airwakes using a lattice-Boltzmann method,” *J. Wind Eng. Ind. Aerodyn.*, vol. 96, no. 6–7, pp. 1197–1206, Jun. 2008.

[26] G. F. Syms, “Numerical Simulation of Frigate Airwakes,” in *International Journal of Computational Fluid Dynamics*, 2004, vol. 18, no. 2, pp. 199–207.

[27] J. S. Forrest and I. Owen, “An investigation of ship airwakes using Detached-Eddy Simulation,” *Comput. Fluids*, vol. 39, no. 4, pp. 656–673, Apr. 2010.

[28] D. Lee, N. Sezer-Uzol, J. F. Horn, and L. N. Long, “Simulation of Helicopter Shipboard Launch and Recovery with Time-Accurate Airwakes,” *J. Aircr.*, vol. 42, no. 2, pp. 448–461, Mar. 2005.

[29] J. S. Forrest, I. Owen, G. D. Padfield, and S. J. Hodge, “Ship-Helicopter Operating Limits Prediction Using Piloted Flight Simulation and Time-Accurate Airwakes,” *J. Aircr.*, vol. 49, no. 4, pp. 1020–1031, Jul. 2012.

[30] S. J. Zan, “On aerodynamic modelling and simulation of the dynamic interface,” *Proceedings of the Institution of Mechanical Engineers, Part G: Journal of Aerospace Engineering*, vol. 219, no. 5. pp. 393–410, Oct-2005.

[31] V. Seshadri, S. N. Singh, and P. R. Kulkarni, “Study of Problem of Exhaust Smoke Ingress into GT Intakes of a Naval Ship,” *J. Sh. Technol.*, vol. 2, pp. 22–35, 2006.

[32] P. R. Kulkarni, S. N. Singh, and V. Seshadri, “Parametric studies of exhaust smoke-superstructure interaction on a naval ship using CFD,” *Comput. Fluids*, vol. 36, no. 4, pp. 794–816, May 2007.

[33] N. A. Watson, 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 Eng.*, pp. 562–574, Jan. 2019.

[34] K. R. Reddy, R. To�oletto, and K. R. W. Jones, “Numerical simulation of ship airwake.”

[35] E. Alpman, L. N. Long, D. O. Bridges, and J. F. Horn, “Fully-Coupled Simulations of the Rotorcraft / Ship Dynamic Interface.”

[36] P. Scott, M. White, and L. Owen, “The effect of ship size on airwake aerodynamics and maritime helicopter operations,” *41st Eur. Rotorcr. Forum 2015, ERF 2015*, vol. 2, 2015.

[37] C. H. Kääriä, Y. Wang, M. D. White, and I. Owen, “An experimental technique for evaluating the aerodynamic impact of ship superstructures on helicopter operations,” *Ocean Eng.*, vol. 61, pp. 97–108, 2013.

[38] J. Geder, R. Ramamurti, and W. C. Sandberg, “Ship Airwake Correlation Analysis for the San Antonio Class Transport Dock Vessel,” 2008.

[39] C. Crozon, R. Steijl, and G. N. Barakos, “Numerical Study of Helicopter Rotors in a Ship Airwake,” *J. Aircr.*, vol. 51, no. 6, pp. 1813–1832, Aug. 2014.

[40] M. Rahimpour and P. Oshkai, “Experimental investigation of air fl ow over the helicopter platform of a polar icebreaker,” *Ocean Eng.*, vol. 121, pp. 98–111, 2016.

[41] C. H. Kääriä, Y. Wang, J. Curran, J. Forrest, and I. Owen, “AirDyn: An Airwake Dynamometer for measuring the impact of ship geometry on helicopter operations.”

[42] C. H. Kääriä, J. S. Forrest, and I. Owen, “Assessing the suitability of ship designs for helicopter operations using piloted flight simulation.”

[43] S. Shukla, S. S. Sinha, and S. N. Singh, “Ship-helo coupled airwake aerodynamics: A comprehensive review,” *Progress in Aerospace Sciences*, vol. 106. Elsevier Ltd, pp. 71–107, 01-Apr-2019.

[44] W. D. Janssen, B. Blocken, and H. J. van Wijhe, “CFD simulations of wind loads on a container ship: Validation and impact of geometrical simplifications,” *J. Wind Eng. Ind. Aerodyn.*, vol. 166, pp. 106–116, Jul. 2017.

[45] R. W. Coleman, W. G. Steele, and J. Wiley, “Experimentation And Uncertainity Analysis For Engineers Second Edition,” 1999.

[46] P. J. Roache, “Perspective: A Method for Uniform Reporting of Grid Refinement Studies,” 1994.

[47] A. J. E. M. L. Sirovich, H. P. Holmes, J. K. J. Keller, B. J. M. A. Mielke, and C. S. P. K. R. Sreenivasan, *Volume 158*. .

[48] R. G. and J. V. John D. Anderson Jr., Joris Degroote, G´ erard Degrez, Erik Dick, *Computational Fluid Dynamics*, Third Edition. Belgium, 2009.

[49] L. F. Richardson, “The Approximate Arithmetical Solution by Finite Differences of Physical Problems Involving Differential Equations, with an Application to the Stresses in a Masonry Dam,” *Philos. Trans. R. Soc. Lond. Ser. Contain. Pap. Math. Phys. Character*, vol. 210, 1910.

[50] F. Stern, R. V. Wilson, H. W. Coleman, and E. G. Paterson, “Comprehensive approach to verification and validation of CFD simulations—Part 1: Methodology and procedures,” *J. Fluids Eng. Trans. ASME*, vol. 123, no. 4, pp. 793–802, 2001.

[51] P. J. Roache, “Verification of codes and calculations,” *AIAA J.*, vol. 36, no. 5, pp. 696–702, 1998.

[52] I. B. C. Celik, U. Ghia, P. J. Roache, C. J. Freitas, H. Coleman, and P. E. Raad, “Procedure for Estimation and Reporting of Uncertainty Due to Discretization in CFD Applications,” *J. Fluid Eng.*, vol. 130, no. 078001–3, 2008.

[53] D. C. Wilcox, *Turbulence Modelling for CFD 3rd Edition*, 3rd ed. 2006.

[54] J. Forrest and I. Owen, “Determining the Impact of Hangar-Edge Modifications on Ship-Helicopter Operations using Offline and Piloted Helicopter Flight Simulation Rotorcraft Virtual Engineering View project Flow Misalignment and Tidal Stream Turbines View project,” 2010.

[55] F. Çelik, “Ship Resistance and Propulsion Lecture Notes.”

[56] V. Bertram, *Practical Ship Hydrodynamics*. Elsevier Ltd, 2012.

[57] J. SODJA Mentor and prof Rudolf PODGORNIK, “Turbulence models in CFD,” 2007.

[58] S. W. K. Y.S. Chen, “Computation of Turbulent Flows Using an Extended k-epsilon Turbulence Closure Model,” *NASA Contract. Rep.*

[59] S. P. S. J.K. Kok, “Efficient and Accurate Implementation of the k-omega Turbulence Model in the NLR Multi-block Navier-Stokes System,” *Natl.*

Aerosp. Lab. Rep., vol. 73, pp. 1–30.

[60] S. R. Allmaras, F. T. Johnson, and P. R. Spalart, “Modications and Clarications for the Implementation of the Spalart-Allmaras Turbulence Model * YourEncore; Boeing Commercial Airplanes retired * * Senior Technical Fellow, Boeing Commercial Airplanes,” 2012.

PUBLICATIONS FROM THE THESIS

Contact Information: sinem.oksuz.91@gmail.com

Conference Papers

1. S. Oksuz and A. Dogrul, “ON THE VERIFICATION AND VALIDATION OF FLOW AROUND SUPERSTRUCTURE OF A SURFACE COMBATANT,” in ICAME 2019, 2019, no. December, pp. 1–17.