

Bio-Mimicry and the use of Shark-Skin Inspired Technology

For the Reduction of Surface Resistance
of Submarines and Other Underwater Craft

Undergraduate Honours Thesis

Presented In Partial Fulfilment of the Requirements

for

4th Year Individual Project

**DELETED DUE TO
ANONYMITY CHECK**

DELETED DUE TO ANONYMITY CHECK

8th of May 2015

ABSTRACT

The skin patterns of fast swimming sharks present varying configurations of interlocking microscopic scales, for the permission of increased efficiency between the interaction of the sharks surface and the surrounding fluid. These patterns have been deemed to follow riblet structured configurations by multiple past researchers, with representative surfaces investigated in their individual emanation of specific shark skin properties, for the accumulative reduction of surface friction resistance. Previously publicised research has recorded a notable reduction of surface friction resistance in the region of 10%, through optimisation of the particular structures geometrical attributes, highlighting the desire for application within the commercial sector. These studies have considered varying aspects of any riblet structures influence on drag reduction, however, they present multiple evaluations between the justification of any reduction and the instigation of fundamental characteristics within the fluid flow. In order to clarify the balance of characteristics required for presentation of any reduction in surface friction resistance, this study explores the simulation of three riblet structured configurations by computational fluid dynamics analysis, whereby, the endorsed drag influences of each case are evaluated in respect to their resultant fluid flow properties. In conclusion of this process, a conquering geometrical configuration is then apparent, allowing further investigation of the particular structure towards application on a representative submarine hydroplane to commence. This secondary area of investigation is presented in attempt to relay the findings of simplistic riblet application towards realistic structures. Although the evidential restraints enforced on the final simulation prohibit any reduction in surface friction resistance to be taken from this analysis, this area of subsidiary investigation nonetheless contributes value towards the advancement of future shark skin study, where a perceived shift in focus is destined towards their investigation on ‘real life’ bodies.

ACKNOWLEDGEMENTS

DELETED
DUE TO
ANONYMITY
CHECK

Completion of this study has been aided by the guidance of my advisor, whose knowledge and time contributions have allowed me to not only produce the study at hand, but develop as a student and researcher along the way. Furthermore, I would like to thank Mr

DELETED DUE TO ANONYMITY CHECK

Engineering Research Centre, for all his time and effort spent assisting me with the computational fluid dynamics analysis associated with this study.

Lastly, I would like to send additional thanks to the DELETED DUE TO

ANONYMITY CHECK in providing me with the computational power and time, required in order to meet the demands of the inherent study.

TABLE OF CONTENTS

ABSTRACT

ACKNOWLEDGEMENTS

TABLE OF CONTENTS

ABBREVIATIONS & NOMENCLATURE

LIST OF FIGURES

LIST OF TABLES

1. INTRODUCTION

1.1 Project Background

1.2 Project Outline

2. PROJECT JUSTIFICATION

2.1 Benefits of Introducing Riblet Structures

2.2 Physical & Commercial Enhancements by Riblet Application

2.2.1 Current Enhancements by Shark Skin Inspired Surfaces

2.2.2 Proposed Fuel and Environmental Enhancements

2.2.3 Proposed Enhancements of Application to Submarines & Underwater Craft

3. PROJECT OBJECTIVES

3.1 Summarised Project Objectives

4. LITERATURE REVIEW

4.1 Introduction to Literature Review

4.2 Introduction to Fluid Flow Mechanisms

4.3 Review of Riblet Mechanisms on Drag Reduction

4.4 Effects of Differing Riblet Geometries

4.4.1 Dimensionless Riblet Characteristics

4.4.2	<i>Characteristics of Two-Dimensional and Three-Dimensional Riblet Structures</i>
4.4.3	<i>Examination into Location of Riblet Application on a Body</i>
4.5	Theoretical Models for Riblet Study
4.5.1	<i>Reynolds Averaged Navier Stokes Models, RANS</i>
4.5.2	<i>Large Eddy Simulation, LES</i>
4.5.3	<i>Direct Numerical Simulation, DNS</i>
4.6	Experimental Study of Riblet Structures
5.	ANALYTICAL METHODOLOGY
5.1	Summarisation of Methodological Process
5.2	Computation Fluid Dynamics Software
5.3	Validation & Verification Test Cases
5.3.1	<i>Two-Dimensional Cylinder</i>
5.3.2	<i>Two-Dimensional Airfoil</i>
5.3.3	<i>Driven Cavity Simulation</i>
5.3.4	<i>Backward Facing Step Simulations</i>
5.4	Shark Skin Inspired Riblet Applied Flat Plate Configurations
5.4.1	<i>Particulars of the Riblet Geometry</i>
5.4.2	<i>Geometrical Attributes of the Fluid Domain</i>
5.4.3	<i>Characteristics of the Applied Mesh</i>
5.4.4	<i>Configurations of the Computational Physics</i>
5.5	Hydroplane Application by Optimised Riblet Structure
5.5.1	<i>Particulars of the Hydroplane Geometry & Surrounding Fluid Domain</i>
5.5.2	<i>Particulars of the Riblet Geometry</i>
5.5.3	<i>Characteristics of the Applied Mesh</i>
5.5.4	<i>Configurations of the Computational Physics</i>
6.	RESULTS & DISCUSSION
6.1	Depiction of Validation & Verification Test Cases
6.1.1	<i>Two Dimensional Cylinder</i>

6.1.2	<i>Two Dimensional Airfoil</i>
6.1.3	<i>Driven Cavity Simulation</i>
6.1.4	<i>Backward Facing Step Simulations</i>
6.2	Exploration of Riblet Applied Flat Plate Test Cases
6.2.1	<i>Flow Characteristics of Each Model</i>
6.2.2	<i>Influence of Riblet Configuration on Drag Reduction</i>
6.3	Exploration of Optimised Riblet Application to Hydroplane Simulation
6.3.1	<i>Flow Characteristics of Hydroplane Simulations</i>
6.3.2	<i>Influence of Riblet Application on Drag Reduction</i>
7.	INTERPRETATION OF PROJECT RESULTS
7.1	Imposed Knowledge Contributions of Study
7.2	Evaluation for Industrial Application
7.3	Development for Future Study
8.	PERSONAL DEVELOPMENT & REFLECTION
9.	PROJECT CONCLUSIONS
10.	REFERENCES

ABBREVIATIONS & NOMENCLATURE

A,	Area of Domain Inlet Region
C_f,	Coefficient of Skin Friction
d,	Height of Inlet Region for Laminar Backward Facing Step
D,	Diameter of Applicable Geometry
h,	Physical Quantity of Riblet Structure Height
h⁺,	Non Dimensional Height of Riblet Structure
H_d,	Hydraulic Diameter
l_e,	Length of Inlet Channel for Laminar Backward Facing Step
L_x,	Geometrical Stipulation of Domain Width, Height and Length (1, 2 and 3)
ρ,	Density of Fluid
P,	Perimeter of Domain Inlet Region
Q,	Instantaneous Flux Volume
Re,	Characteristic Reynolds Number
s,	Physical Quantity of Riblet Structure Lateral Spacing
s⁺,	Non Dimensional Lateral Spacing of Riblet Structure
δ,	Fluid Domain Half Width
τ₀,	Theoretical Wall Shear Stress
t,	Physical Quantity of Individual Riblet Blade Thickness
t⁺,	Non Dimensional Thickness of Individual Riblet Blades
Δt,	Computational Time Step
μ,	Dynamic Viscosity of Fluid
U_i,	Centreline Velocity of Parabolic Laminar Profile
ν,	Kinematic Viscosity of Fluid
V,	Mean Fluid Velocity
V_τ,	Wall Shear Stress Velocity

LIST OF FIGURES

- Figure 1,** **Examples of Bio-mimicry at use within the Commercial World, Adapted from [1]**
- Figure 2,** **Examples of Shark Skin Dermal Denticle Skin Structure, Adapted from [5]**
- Figure 3,** **Turbulent Flow Visualisation of Vortices Interacting with Flat Plate (a, b) and Riblet Surfaces (c, d) in a Vertical Cross-Section, Adapted from [7]**
- Figure 4,** **Representation of Future Marine Fuel Price Increase, Adapted from [14]**
- Figure 5,** **Representation of Fluctuating Crude Oil Prices Within Period of 1861 to 2011, Adapted from [15]**
- Figure 6,** **Representation of Laminar, Transitional and Turbulent Fluid Flow, Adapted from [19]**
- Figure 7,** **Illustration of the Process of Span-Wise Lifting and the Resulting Formation of a Counter-Rotating Vortex Pair, Adapted from [23]**
- Figure 8,** **Effective origin of Individual Riblet, Adapted from [26]**
- Figure 9,** **Representation of Standardised Riblet Geometries, Adapted from [8]**
- Figure 10,** **Effects of Sawtooth Riblets, Adapted From [28]**
- Figure 11,** **Example of Staggered 3-D Riblet Structure, Comparison of 2-D and 3-D Riblets on Drag Reduction, Adapted from [8, 22]**
- Figure 12,** **Scale Pattern Variation on Fast-Swimming Sharks, Adapted from [22]**
- Figure 13,** **Introductory Aspects of Choosing Particular CFD Method, Adapted from [31,32]**
- Figure 14,** **Representative Experimental Procedures for Testing Drag Reduction of Riblet Structures Adapted From [37, 38]**
- Figure 15,** **Computational Configuration for Two-Dimensional Cylinder Test Case**
- Figure 16,** **Representation of Airfoil at 30° Angle of Attack to Advancing Fluid Flow, Adapted From [43]**
- Figure 17,** **Computational Configuration of Two-Dimensional Airfoil Test Case, Conveying Mesh Characteristics in Relation to Airfoil Wall**
- Figure 18,** **Representation of Vortex Formulation Within Lid Driven Cavity, Adapted From [45]**
- Figure 19,** **Representative Form of Backward Facing Step Geometry, Adapted From [46]**
- Figure 20,** **Geometrical Configurations of Riblet Applied Flat Plate Test Cases**
- Figure 21,** **Generated Unstructured Mesh of Model 2 Fluid Domain, Showcasing Domain Exterior and Front Face, with Closer Examination of Region Surrounding Individual Riblet Blades**

Figure 22,	Generated Structured Mesh of Model 2 Fluid Domain, Showcasing Attributable Edge Sizings
Figure 23,	Representation of Applied Boundary Conditions for Each Riblet Applied Flat Plate Test Case
Figure 24,	Geometrical Representation of Hydroplane Body & Corresponding Fluid Domain
Figure 25,	Velocity Magnitude Contour Plot of Airfoil Simulation at Zero Angle of Attack, Adapted From [43]
Figure 26,	Geometrical Representation of Riblet Applied Hydroplane Body
Figure 27,	Generated Unstructured Mesh for Smooth Surfaced Hydroplane Model
Figure 28,	Generated Unstructured Mesh for Riblet Applied Hydroplane Model
Figure 29,	Representation of Applied Boundary Conditions for Each Hydroplane Test Case
Figure 30,	Publicised Drag Coefficient of Smooth Cylinder at Varied Reynolds Number, Adapted From [52]
Figure 31,	Contour Plots of Static Pressure & Velocity Magnitude Inherent to Test Case Simulation
Figure 32,	Comparison of Horizontal Velocity Through Vertical Cavity Centre
Figure 33,	Comparison of Vertical Velocity Through Horizontal Cavity Centre
Figure 34,	Comparison of Static Pressure Within Cavity, Test Case Contour Plot Left, Bruneau Right
Figure 35,	Comparison of Stream Function Within Cavity, Test Case Contour Plot Left, Bruneau Right
Figure 36,	Comparison of Test Case Horizontal Fluid Velocity at $x=7$ and $x=15$ to Gartling [54]
Figure 37,	Comparison of Test Case Vertical Fluid Velocity at $x=7$ and $x=15$ to Gartling [54]
Figure 38,	Representative Velocity Magnitude of Laminar Test Case, Containing Non Dimensional Divisions
Figure 39,	Detachment & Reattachment Locations of Flow Throughout Step Geometry, Adapted From [54]
Figure 40,	Examination of Primary Reattachment Region of Turbulent Test Cases, With Top Representative of $Re = 93,000$ and Bottom of $Re = 210,000$
Figure 41,	Contour Plot of Velocity Magnitude Inherent to Model 2 Simulation
Figure 42,	Contour Plot of Tangential Velocity Inherent to Model 1 Simulation, Representative of Secondary Vortex Development
Figure 43,	Contour Plot of Tangential Velocity Inherent to Model 2 Simulation, Representative of Secondary Vortex Development
Figure 44,	Contour Plot of Tangential Velocity Inherent to Model 3 Simulation, Representative of Secondary Vortex Development

- Figure 45, Integral Wall Shear Stress Contour Plot Comparison Between Smooth and Riblet Structured Surfaces Inherent to Model 2 Simulation**
- Figure 46, Integral Wall Shear Stress Development in Relation to Spanwise Location of Fluid Domain, Inherent to Model 1 Simulation**
- Figure 47, Integral Wall Shear Stress Development in Relation to Spanwise Location of Fluid Domain, Inherent to Model 2 Simulation**
- Figure 48, Integral Wall Shear Stress Development in Relation to Spanwise Location of Fluid Domain, Inherent to Model 3 Simulation**
- Figure 49, Contour Plots of Velocity Magnitude Upon Intersection with Vertical and Horizontal Centres of Smooth Hydroplane**
- Figure 50, Contour Plots of Velocity Magnitude Upon Intersection with Vertical and Horizontal Centres of Riblet Applied Hydroplane**
- Figure 51, Contour Plots of Static Pressure Upon Intersection with Vertical and Horizontal Centres of Smooth Surfaced Hydroplane**
- Figure 52, Pressure Coefficient Development in Relation to Streamwise Location of Hydroplane Body, Comparison of Smooth and Riblet Applied, with Further Inspection of Smooth Hydroplane**
- Figure 53, Integral Wall Shear Stress Development in Relation to Spanwise Location of Hydroplane Body, Inherent to both Smooth and Riblet Applied Test Cases**
- Figure 54, Contour Plot of Integral Wall Shear Stress Development, Inherent to Smooth Surfaced Hydroplane Test Case**
- Figure 55, Contour Plot of Integral Wall Shear Stress Development, Inherent to Riblet Applied Hydroplane Test Case**
- Figure 56, Balance of Introduced Characteristics of Riblet Structured Surface in Provision of Drag Reduction**

LIST OF TABLES

Table 1,	Representation of Past, Present and Future NO_x and SO_x Emission Levels, Adapted from [16]
Table 2,	Representation of RANS Models Available for Application, Adapted from [34]
Table 3,	Representation of LES Models Available for Application, Adapted from [34]
Table 4,	Experimental Procedures for Testing Drag Reduction of Riblet Structures
Table 5,	Characteristics of Two-Dimensional Cylinder Mesh
Table 6,	Configurations of Applied Computational Physics for Two-Dimensional Cylinder Test Case
Table 7,	Characteristics of Two-Dimensional Airfoil Mesh
Table 8,	Configurations of Applied Computational Physics for Two-Dimensional Airfoil Test Case
Table 9,	Configurations of Applied Computational Physics for Lid Driven Cavity Test Case
Table 10,	Characteristics of Applicable Backward Facing Step Mesh(s)
Table 11,	Configurations of Applied Computational Physics for Backward Facing Step Test Cases
Table 12,	Non Dimensional Spacing Configurations of Riblet Applied Flat Plate Models
Table 13,	Specification of Endorsed Unstructured Mesh for Riblet Applied Flat Plate Test Cases
Table 14,	Specifications of Endorsed Structured Mesh for Riblet Applied Flat Plate Test Cases
Table 15,	Configurations of Applied Computational Physics for Riblet Applied Flat Plate Test Cases
Table 16,	Characteristics of the Endorsed Unstructured Mesh for Smooth Surfaced Hydroplane Model
Table 17,	Characteristics of the Endorsed Unstructured Mesh for Riblet Applied Hydroplane Model
Table 18,	Configurations of Applied Computational Physics for Each Hydroplane Test Case
Table 19,	Comparison of Drag Coefficient-Published to Test Case Results, at Reynolds Number 1, 20 & 100
Table 20,	Test Case Lift & Drag Analysis in Regards to Experimental Data, Adapted From [43]
Table 21,	Comparison of Reattachment Location - Published to Test Case Results
Table 22,	Examination of Individual Riblet Configuration on Friction Drag Influence
Table 23,	Evaluation of Lift and Drag Coefficients Inherent to Smooth and Riblet Applied Hydroplane Cases

INTRODCUTION

1.1 Project Background

The evolutionary process has created a plethora of examples of organisms whose traits can be exploited for commercial interest and application [Figure 1]. Historically, many marine creatures have been subject to in-depth study and have formed the inspiration for varied areas of research, from Leonardo Da Vinci's interest in flow dynamics to renaissance design of streamlined bodies, and with the need for new technologies at an all time high, this focus remains today [2]. Sharks may conjure notions of fearsome predators whose demeanour is to be left untouched, however, they possess many intriguing features of use in a wide range of specialities, from the advancing nature of medical technologies to swimsuit designs, and of key relevance within this study, to marine applications. In particular, it is the multifunctional properties of the skins of fast swimming sharks which are of interest within the marine sector, and provide the basis of this entire study.

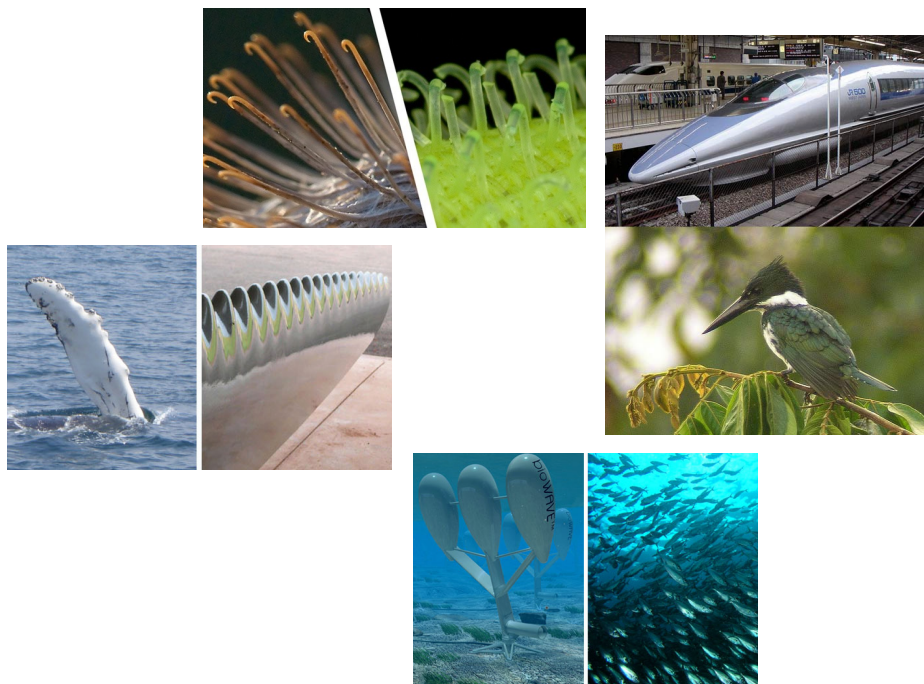


Figure 1, Examples of Bio-mimicry at use within the Commercial World, Adapted from [1]

Simply by nature, sharks are required to move through water with great efficiency in order to catch fast moving prey, obtain sufficient oxygen and maintain buoyancy. Their skin is essential in reducing friction drag and auto cleaning eco-parasites (found within seawater, such as barnacle larvae, algae and various marine bacterias) from its own surface [2, 3, 4]. These factors form major sources of energy inefficiency in the performance of marine vessels, consequently emphasising the strengths that shark skin inspired surfaces can have within the industry.

1.2 Project Outline

The main focus of this project will be in the investigation of shark skin inspired riblet structures in search of offering the technology as a passive drag reducing method for application to submarines and other underwater craft. In doing so, an insight into the justifying need for this research will be presented in terms of the widespread benefits of riblet application, and the distinct advantages which the technology possesses when directed towards implementation on submarines and underwater craft.

Further to this, a substantial reexamination of past studies which focus on shark skin inspired riblet technology is delivered through aid of an in-depth review of past literature, with focus on the following areas — fluid flow and riblet mechanisms for drag reduction — the effects of differing riblet geometries and locations on an applied body — the theoretical and experimental models applicable to riblet study. This review of past literature and experimentation establishes the guidance necessary for the initiation of computational analysis within this particular project.

Following articulation of the reviewed literature, the computational modelling of the desired riblet geometries is then demonstrated, with a selection of validation and verification

models first analysed as to provide some initial insight into the computational fluid dynamics configurations required for obtaining sufficiently accurate results. The modelling of said structures is detailed, with much of the focus directed towards the geometrical, mesh and theoretical setup of each individual test case.

The results of the computational analysis will then be discussed in terms of the integral wall shear stress present over the riblet applied surfaces when compared to that over their smooth counterparts. Furthermore, the induced characteristics of the resultant fluid flow contained within each test domain will be placed under scrutiny, allowing the generated results to be continually examined against past research. This then permits conclusions to be drawn, and contributions to be made towards a better appreciation of the role shark skin inspired riblet surfaces can have in reducing surface friction resistance. In consequence, it is hoped that this particular project can provide a basis for further study into surfaces of ever increasing drag reducing effect and application, while allowing for future transition towards focus in development of practical construction and implementation techniques for riblet surfaces to commence.

PROJECT JUSTIFICATION

The importance of this study has been based upon the application of shark skin inspired riblet technology to submarines and other underwater craft. As such, this section of the thesis is presented in view of showcasing the justifiable need for further study into the technology and its abilities to enhance various aspects of the embodied structure. Furthermore, it aims to deliver a concise foundation as to the underlying physical effects of a sharks natural skin pattern. In turn allowing the resultant benefits of a widespread introduction of shark skin inspired surfaces to the commercial sector to be articulated. The subsequent outcomes of this study and the corresponding knowledge contributions have been concluded on completion of the entire project, as to permit further development for use in other areas of commercial viability.

2.1 Benefits of Introducing Riblet Structures

Although a perfectly smooth surface may be thought of as the most effective aid to surface friction resistance reduction of a body, shark skin is in fact constructed of tiny scales, named dermal denticles [Figure 2].



Figure 2, Examples of Shark Skin Dermal Denticle Skin Structure, Adapted from [5]

These scales act like microscopic teeth along the surfaces of fast swimming sharks, causing their renowned coarse texture and allowing them to move more efficiently through water, the theory of which can be explained by first outlining the general rules of flow over a moving body [6]. Over smooth surfaces, fast moving water begins to break into turbulent vortices, in-part due to the water flowing immediate to the body moving slower than that at further distances. This then causes the water to become turbulent by the adjacent layers of flow intermixing. The primitive theories of laminar and turbulent flow will be discussed in greater detail within the literature review section of thesis [Section 4.2] where a continued focus is placed on fluid drag mechanisms. The grooves created by a sharks dermal denticle skin structure form varied riblet geometries which have been found to reduce the surface friction resistance of the shark by allowing the following processes to take place [4, 6]:

- i. Reinforcement of the flow direction though narrowed channels
- ii. An increase in the speed of the slower water (the same volumes of water remain, however, with the addition of narrower channels, this leads to an increased velocity) therefore reducing the speed difference between layers of flow
- iii. Creation of divisions in the sheet of water flowing over the surface provide a transition from laminar to turbulent flow within the smaller vortices [Figure 3]

Further to this, the capabilities of shark skin in the prevention of marine organisms adhering to the surface are as a result of the following factors [6]:

- i. The accelerated flow of surface water reduces the contact time with fouling organisms

- ii. A reduction in the shark skin surface area due to the roughened nano-texture of the riblet structure provides less space for fouling attachment
- iii. Constant movement of the tiny dermal denticles along the body with changes in pressure, reduce the risk of fouling organisms adhering to the body

From this, the desire to replicate this fascinating natural technology can be identified through application of just some of its abilities to the commercial sector.

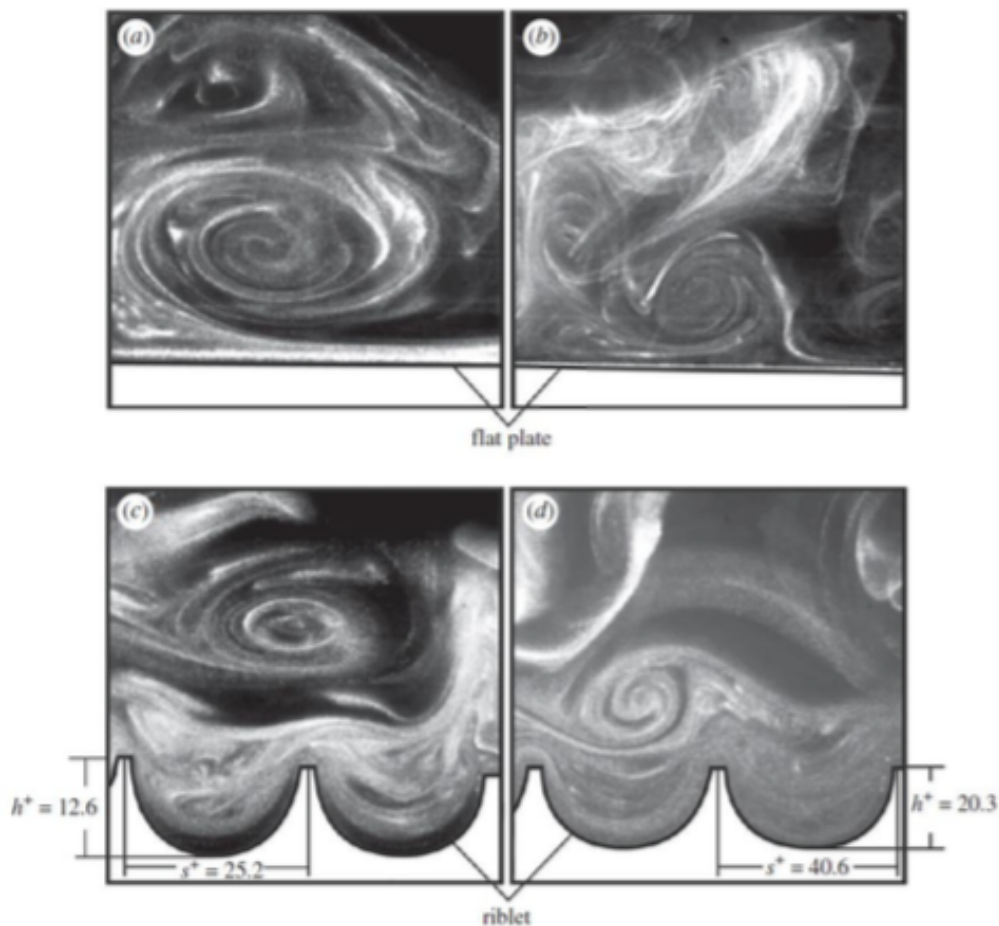


Figure 3, Turbulent Flow Visualisation of Vortices Interacting with Flat Plate (a, b) and Riblet Surfaces (c, d) in a Vertical Cross-Section, Adapted from [7]

2.2 Physical & Commercial Enhancements of Riblet Application

With the industry's focus shifting towards renewable, environmentally efficient and commercially viable solutions, a resultant demand for experimental engineering, exercising an abundance of natural inspirations, has become evident. As such, although much research has been carried out in regard to shark skin inspired surfaces, it remains an underestimated and undervalued sector of biological engineering, with this study relaying further focus on the relation between the results attainable and the practical need for implementation within the marine sector. The following sections of this justification provide reasoning towards adherence of this primary focus and explore the many commercial, environmental and performance enhancing effects which shark skin inspired surfaces can provide.

2.2.1 Current Enhancements by Shark Skin Inspired Surfaces

Surface coatings which emulate various features of sharkskin have been found to provide a maximum 10% drag reduction across various areas of past physical and experimental research. For example, the Speedo engineered 'sharkskin suit' first worn by athletes at the 2000 Olympics in Sydney, allowed swimmers to break thirteen of the fifteen world records available at the event, showcasing the abilities of the technology on a global scale [8, 9]. The results of the event have since been evaluated, with focus placed on the level of performance enhancement provided by the shark skin inspired swimwear, resulting in their prohibition from future events [10]. This prohibition emphasises the abilities possessed by shark skin technology, highlighting the desire to mirror these results within the marine sector, and ultimately sanctioning significant escalations in the performance characteristics of an applied vessel. Simultaneously, a large scale reduction in not only voyage costs but those of engine implementation and maintenance could be seen. For instance, studies by the University of Florida, in partnership with the US Navy, have

shown that a coating of tiny diamond like scales, which flex to impede growth, have given an 85% reduction in spore settlement. When this type of result is discussed in relation to the large scale costs (\$600million) the US Navy spend each year in powering its vessels, of which almost 9% (\$50million) is a direct result of biofouling, it can be seen that if this technology could be implemented on a large scale, it would lead to a cost saving of almost 8%, a significant figure at such high initial costs [11]. Furthermore, studies carried out in relation to the appliance of a riblet structured surface to aircraft have yielded similarly encouraging results [12]. Here 70% of the aircraft is pertinent to riblet application, with the remaining 30% consumed by areas of essential visibility and high abrasion. In following this configuration the study predicts a garnered drag reduction of 3%, translating to a subsequent reduction in fuel costs by a similar figure. When combined with current airline fuel usage, such as figures released by the American Bureau of Transportation Statistics for 2014 [13], where \$46.8 billion of jet fuel was utilised, it can be noted that the drag reduction rates found would render fuel savings of \$1.4 billion. As such, these figures along with the varied examples provided here, exploit the role of shark skin inspired riblet technology can have within the commercial sector.

2.2.2 Proposed Fuel and Environmental Enhancements

The projected rise in fuel costs over the coming decades [Figure 4] has emphasised the current reality in developing sustainable products which can aid the reduction of fuel consumption through various methods. The projection contained here has been obtained from studies carried out by Lloyds Register in coalition with DNV [14], and comes as a contradiction to the current reality, where dwindling oil prices have been common on the global market, resulting in correspondingly reduced fuel costs. However, as past reviews of global fuel and oil

prices have shown, there exist various fluctuations over the past decades, all the while, a continued trend towards a persistent increase in fuel costs remains [Figure 5].

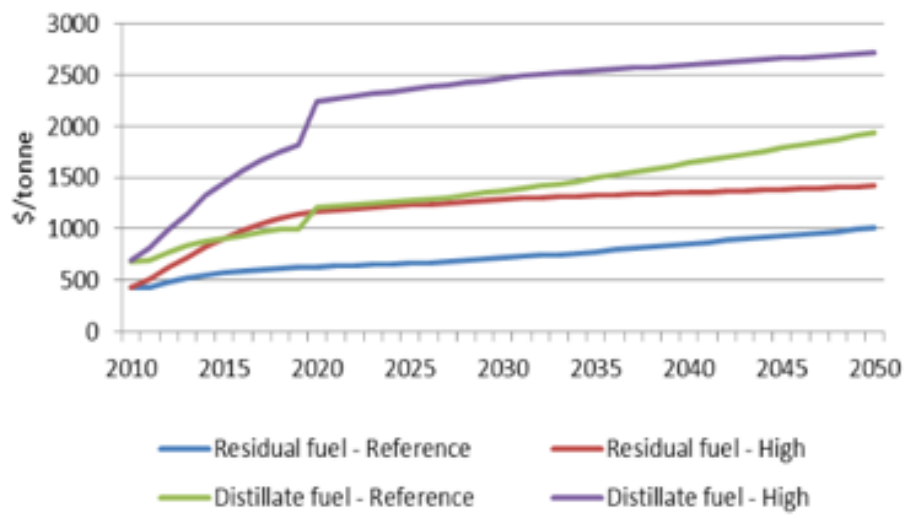


Figure 4, Representation of Future Marine Fuel Price Increase, Adapted from [14]

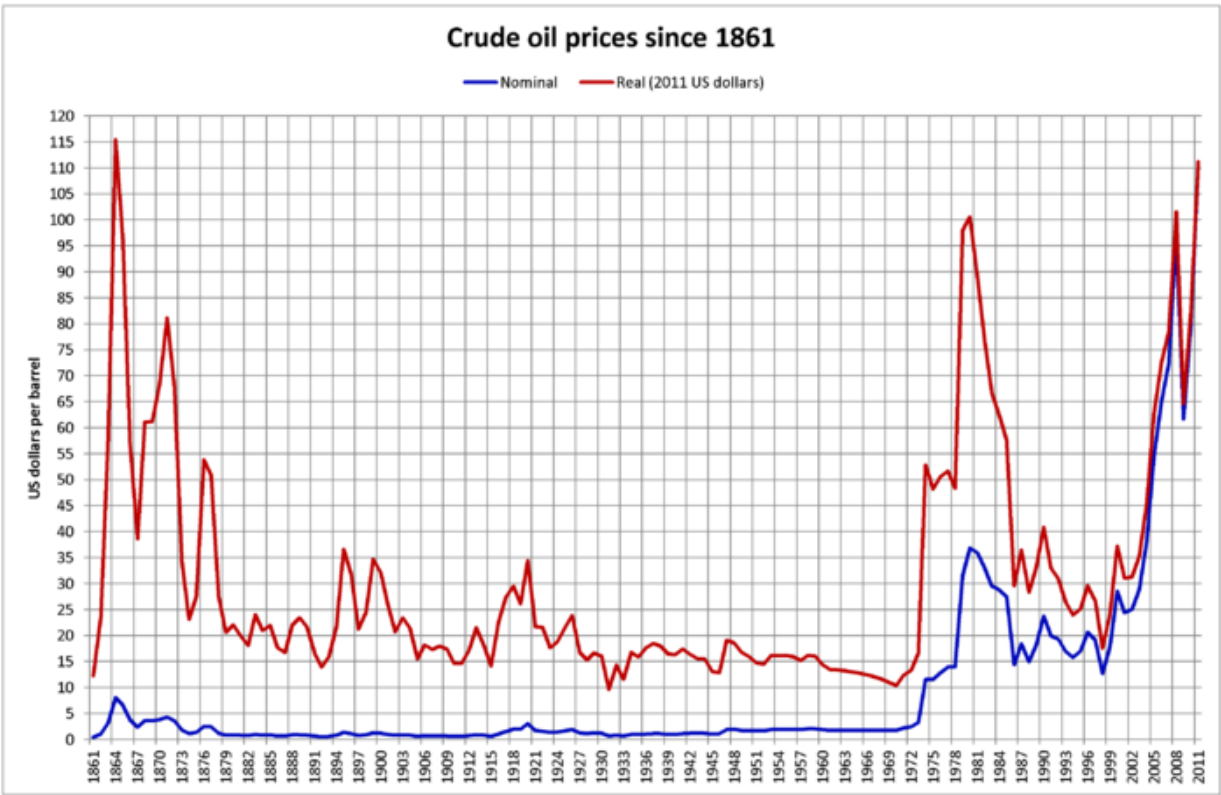


Figure 5, Representation of Fluctuating Crude Oil Prices Within Period of 1861 to 2011, Adapted from [15]

As such, the implementation of shark skin inspired riblet technology within the marine sector could allow for the widespread reduction of fuel costs through associations with skin friction resistance reduction, as is detailed throughout the above section [Section 2.2.1], where drag reducing rates have been shown to reach levels of 10%. In addition to concentration on fuel costs, the industry's current progressions towards an increasingly strict control of marine emissions [Table 1] could also be benefitted from the introduction of this technology.

Tier	Date	NOx Limit, g/kWh			Date	Sulphur Limit in Fuel, % m/m	
		n < 130	130 < n < 2000	n > 2000		SOx ECA	Global
Tier I	2000	17.0	$45 \cdot n^{-0.2}$	9.8	2000	1.5%	4.5%
Tier II	2011	14.4	$44 \cdot n^{-0.23}$	7.7	2010		
Tier III	2016	3.4	$9 \cdot n^{-0.2}$	1.96	2012	1.0%	3.5%
					2015		
					2020	0.1%	0.5%

Table 1, Representation of Past, Present and Future NOx and Sox Emission Levels, Adapted from [16]

This reduction in marine emissions would be met in conjunction with the lowered fuel consumptions realised due to the reduced skin friction resistance associated with riblet surfaces, as each of these factors contribute greatly to one another. This then provides a preliminary insight into the effects a lowered surface friction resistance can have, conveying the interdependent nature of surface friction resistance, fuel consumption, fuel cost and marine emissions. However, it is in the practical implementation of such a technology to the marine sector that can allow for drag reducing results to be obtained, emphasising the fundamental need for further study into the diverse nature and application of shark skin inspired technology.

2.2.3 Proposed Enhancements of Application to Submarines & Underwater Craft

The varied physical, commercial and environmental influences of riblet applied surfaces detailed throughout the above sections remain pertinent when relating to submarines and underwater craft. Many of these factors prove to have further significance in this particular sector where friction drag constitutes approximately 65% of total drag imposed on submarines, in comparison to approximately 50% on surface ships [17]. Consequently, a reduction in surface friction resistance, as a result of riblet application, will have a greater significance on the corresponding diminution of fuel consumption, fuel cost and marine emissions. Furthermore, when considering some of the distinct influences shark skin inspired surfaces can have on the surrounding flow, as in [Section 2.1], the turbulent aftermath of the vessel at hand can be reduced. This can be invoked by the fact that the surrounding flow of water is disturbed in two distinct ways:

- i. A component of the fluid is dragged along with the vessel, diminishing its energy while reducing its velocity and performance characteristics
- ii. A rushing phenomenon of fluid occurs within the vessels aftermath, where fluid compensates for the vacant space left by the vessel itself

Then the reduction in large scale vortices present over a riblet applied surface, in conjunction with the reinforcement of fluid flow through narrowed channels, can sanction a reduction in both these characteristics. The resultant reduction in the churning motion within the vessels wake permits a reduced level of noise and vibration transmission from the vessel to the sea state, which is of critical importance when discussing naval and military submarines, allowing for an escalation in the stealth and security of the vessel, its personnel and its operation. These factors can be further applied to underwater craft, such as AUV's, reiterating the fundamental necessity for further in-depth study into the role of shark skin inspired technology within the marine sector.

PROJECT OBJECTIVES

3.1 Summarised Project Objectives

- I. Investigation into varied areas of past physical and experimental research to generate detailed knowledge of shark skin inspired surfaces and their unique influence on friction resistance reduction. As a result, optimisation of applicable riblet geometries and the subsequent theoretical methods for analysis will be undertaken, establishing the data required for the succeeding objectives to be accomplished.
- II. Commencement of validation and verification process will be conducted to initiate the computational fluid dynamics (CFD) models applicable for study of riblet structures. Furthermore, the delivery of sufficiently similar results, in conjunction with past studies, will provide further evidence that the analysis conducted has been correctly setup.
- III. Geometry creation of riblet applied flat plates, and the corresponding CFD testing of said structures will be conducted in view of showcasing an ‘optimum’ geometry, in relation to the individual variation of integral wall shear stress present over each geometrical setup when compared to a smooth plate.
- IV. The above mentioned ‘optimum’ riblet geometry will then be applied to an airfoil as to mimic a similar geometrical setup as a submarine hydroplane (either stern control plane of bow control plane). This provides a means of examining the practical implementation of riblet surfaces to submarines and underwater craft, where the riblet applied hydroplane will again be examined in conjunction with its smooth counterpart.

- V. A detailed comparison of the outcomes of this analysis alongside past research will be conducted in order to provide clear evidence of the projects findings, and allow for the scope of future study to be outlined.

LITERATURE REVIEW

4.1 Introduction to Literature Review

Within this chapter, a review of past research into the use of shark skin technologies has been summarised, with particular focus paid to studies concerning the use of riblet structures and their effects on drag reduction. Within the first section, fluid flow mechanics are introduced, providing justification for the examination of riblet mechanisms for drag reduction which follows, where comparison of past results will be provided and attention concentrated on the riblet mechanisms which yield drag reducing effects. From this, previously studied riblet shapes and sizes will be analysed, allowing their dimensionless characteristics and subsequent effects to be optimised. Thereafter, a review of the differentiating results of studies concerning the use of two-dimensional and three-dimensional riblets will be discussed, alongside an examination of the location of riblet application to a body. This will then allow a further section to review the theoretical models associated with this area of study, with conclusions on the most appropriate method(s) throughout. In turn, a brief summarisation of the experimental methods previously utilised in the analysis of riblets has also been carried out, allowing for future comparison between theoretical and experimental study.

4.2 Introduction to Fluid Flow Mechanisms

In order to discuss the relations between riblet structures and drag reduction, fluid drag must first be introduced and subsequently resolved into two distinct components — pressure drag and skin friction drag (often referred to as viscous drag). Pressure drag is the component of drag associated with the energy required for movement of fluid from in front of an object to behind

said object [5, 18], where it can be relieved through the introduction of streamlined geometries, such as those contained within the designs of many surface ships and submarines. Then skin friction drag is the component of drag associated with the interaction of fluid flow layers and the surface of a body [5, 18], as such, this is the crucial component of drag discussed in relation to shark skin surfaces. Establishing the distinctly differing stages of fluid flow then provides some insight as to the functions of these drag components, where the effects of laminar and turbulent flow will have strong relation to the skin friction drag inflicted on a body, such that turbulence will lead to increased levels of skin friction drag. This is due to the chaotic nature of turbulent flow, in comparison to the smooth uniform paths followed by the fluid molecules in laminar flow, as is detailed through the below figure [Figure 6].

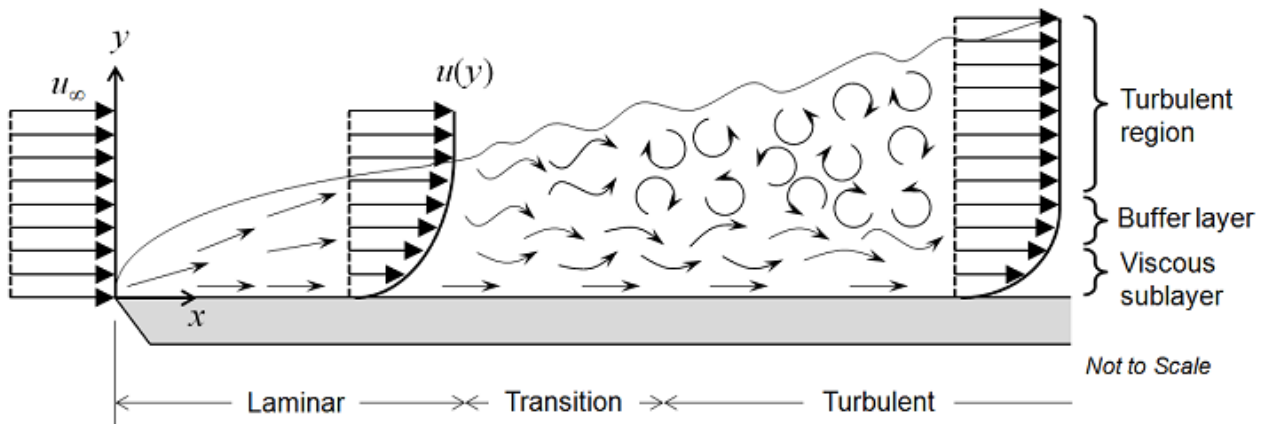


Figure 6, Representation of Laminar, Transitional and Turbulent Fluid Flow, Adapted from [19]

The period of transition between laminar and turbulent regimes is then depicted by the characteristic Reynolds number, whereby, transition occurs within a range of Reynolds numbers of 2300 to 4000 for internal fluid flows and at Reynolds numbers greater than 500,000 for external fluid flows [20]. The dimensionless Reynolds characteristic utilised here is composed of a ratio of inertial forces to viscous forces, as is shown through the formula below [Eq. 1], where ρ

is the fluid density, μ is the dynamic viscosity of the fluid, V is the mean fluid flow velocity and D in this instance reflects the characteristic length of the body, such that for internal flows D represents the hydraulic diameter.

$$Re = \frac{\rho V D}{\mu} \quad \text{Eq. 1}$$

Then, where these Reynolds numbers are reached and turbulence occurs, the fluid molecules have both rotational and translational movement characteristics within the viscous sublayer, representing turbulent vortices. Turbulent vortices have been found to interact not only with the surface of the influential body, but with other vortices within their vicinity also, forcing vortices out of the viscous sublayer and into the outer regions of boundary layer flow. This forced interaction then causes velocity vectors to form in the cross stream direction, of similar magnitudes to those present in the streamwise direction of flow, increasing the momentum transfer, and forcing a higher drag over the body [18, 20]. Then establishing a means of reducing the momentum transfer of flow will correspondingly provide a method of drag reduction.

4.3 Review of Riblet Mechanisms on Drag Reduction

With a brief insight into the mechanisms of riblet drag reduction previously outlined within the justification section of this report [Section 2.1], this subtopic aims to deliver an in-depth analysis of the mechanisms present within a riblet structure for skin friction resistance reduction of an applied body. Although the physical mechanisms of drag reduction by riblets has been investigated in detail for many years by researchers such as, Savill (1989), Walsh (1990) and Coustols & Savill (1992), there remain many unknowns and controversies surrounding their study. It has been demonstrated that riblets can provide a delay in the transition from laminar to

turbulent flow within the viscous sub layer over a body [21]. Additionally, riblets have been shown to divide the vortices created over the surface of a body, lifting the vortices towards the tips of the riblet structure rather than allowing them to rest on the body itself (within the valleys) [6, 8, 21, 22]. The resultant flow interacts only with the riblet tips, forcing a higher shear stress over these particular areas, and subsequently lowering the shear stress inflicted over the majority of the body. In conclusion, this provides a reduced shear stress over the riblet covered portion of the body [6, 22]. This in turn supports the above claim that riblets delay the transition to turbulence, as the momentum thickness growth rate over a smooth surface will generally be greater than that over a riblet covered surface [21, 22]. Further to this, the inclusion of riblets on a surface ensures the redirection of flow over the body, decreasing the cross stream translation and in turn reducing the activity between velocity layers, providing a directional focus on streamwise flow [7].

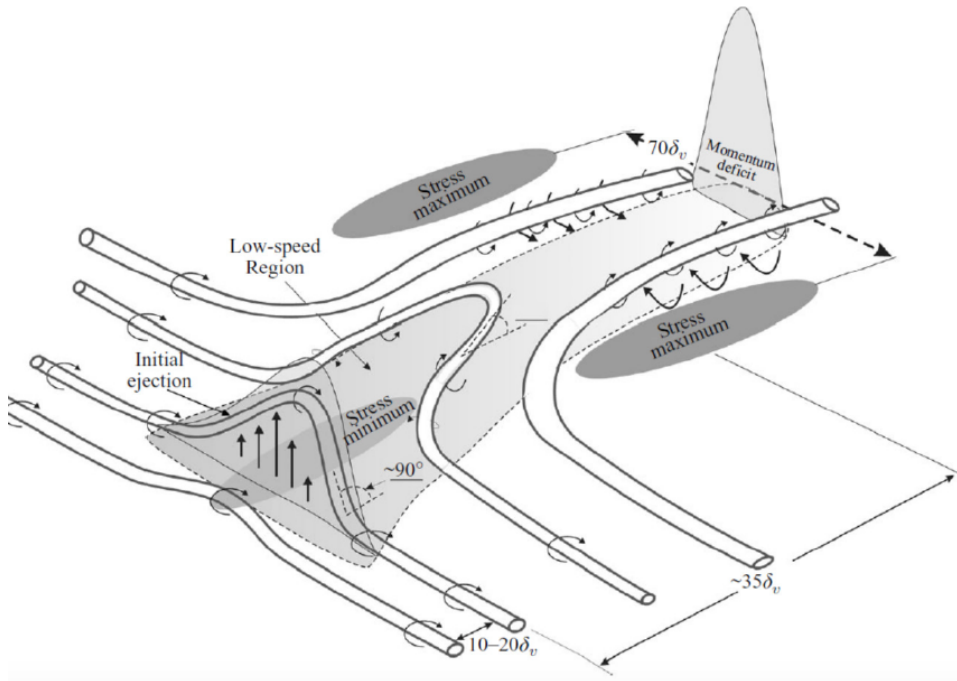


Figure 7, Illustration of the Process of Span-Wise Lifting and the Resulting Formation of a Counter-Rotating Vortex Pair, Adapted from [23]

Further to limitations of cross stream flow over a body, the generation of secondary vortices within the riblet valleys themselves [Figure 7], has been studied by a number of researchers in the field, such as Bacher & Smith (1985). These studies have suggested that the secondary vortices generated provide interference within the span-wise flow at the surface of a body [24]. Further study has concluded that the addition of secondary vortices over the riblets is subject to the separation of flow, providing a loss of vortex strength and resultant decrease in drag by damping associations [24].

4.4 Effects of Differing Riblet Geometries

4.4.1 *Dimensionless Riblet Characteristics*

In addition to the above drag reducing influences riblet structures can provide to a body, the specific role of the riblet's geometry has been shown to have wildly varying effects on the results obtained. Optimisation of a riblets specific geometrical attributes must provide the previously discussed effects — extrusion of the vortices along the surface to the riblet tips, whilst simultaneously directing the flow in the streamwise direction and accounting for an overall increase in surface area — for maximum levels of surface friction residence reduction. In order to analyse varying riblet geometries, a series of non-dimensional parameters are utilised for lateral spacing s^+ , height h^+ and thickness t^+ [Figure 9]. These parameters are defined below [Eq. 2, 3, 4], where s , h and t represent the physical quantities of lateral spacing, height and thickness, ν is the kinematic viscosity of the fluid and V_τ represents the wall shear stress velocity, calculated through kinetic energy relations with the wall shear stress [Eq. 5], [8, 20, 22].

$$s^+ = \frac{sV_\tau}{\nu} \quad h^+ = \frac{hV_\tau}{\nu} \quad t^+ = \frac{tV_\tau}{\nu} \quad \text{Eq. 2, 3, 4}$$

$$V_\tau = \left[\frac{\tau_o}{\rho} \right]^{1/2} \quad \text{Eq. 5}$$

Further studies have resulted in the inception of interdependent relationships between the dimensionless characteristics above and the coefficient of skin friction, C_f , through introduction of the Fanning friction factor formula [Eq. 6], providing the following associations [Eq. 7, 8, 9], [18, 25].

$$C_f = \frac{2\tau_o}{\rho V^2} \quad \text{Eq. 6}$$

$$s^+ = \sqrt{\frac{C_f}{2}} \frac{sV}{\nu} \quad h^+ = \sqrt{\frac{C_f}{2}} \frac{hV}{\nu} \quad t^+ = \sqrt{\frac{C_f}{2}} \frac{tV}{\nu} \quad \text{Eq. 7, 8, 9}$$

These then allow optimisation of predetermined riblet structures to be carried out with a number of conclusions drawn from previous study. The protrusion height, h_d , is one such factor which will vary depending on the spacings mentioned. This relates to the distance between the origin of the effective velocity profile to the tip of the riblet structure [26], [Figure 8]. The most effective location of this parameter, from Bechert & Bartnwerfer (1989), has allowed for a distribution in wall shear stress to be accommodated, and although no drag reducing estimations could be obtained from this particular study, further study has garnered more encouraging results.

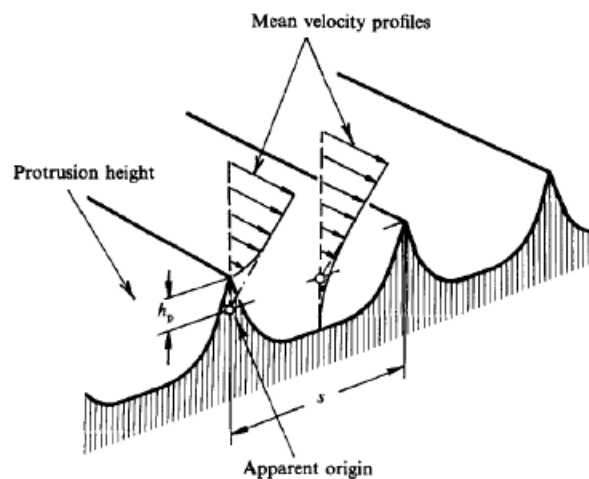


Figure 8, Effective origin of Individual Riblet, Adapted from [26]

4.4.2 Characteristics of Two-Dimensional and Three-Dimensional Riblet Structures

In addition to the above examination of a riblet structures geometrical attributes, the study of both two-dimensional and three-dimensional riblets has furthered the influence of selecting an optimal riblet spacing. Two-dimensional riblets, such as sawtooth (V), scalloped (U) and blade (L) designs provide a continuous structure in the streamwise direction [6, 8], [Figure 9].

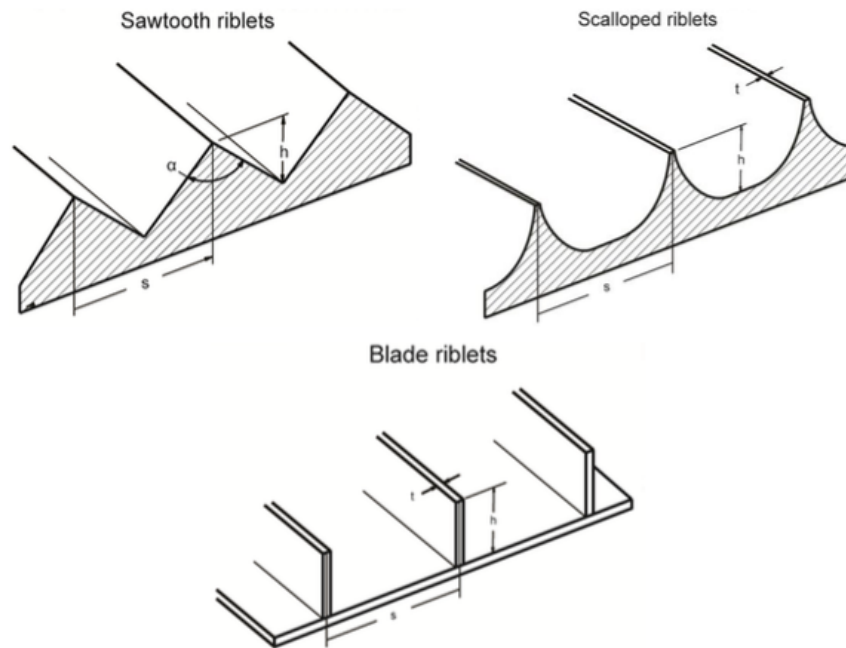


Figure 9, Representation of Standardised Riblet Geometries, Adapted from [8]

Past reviews into the drag reducing effects of these riblet structures, with spacings in the region of $s^+ = 10-15$, within wind tunnel experiments have been completed by the likes of Choi (2000) and Bushnell (2003) [27]. This spacing ratio has been predetermined to allow several riblets to be in contact with any one vortex, therefore, emphasising the riblets ability in lifting the vortex out with the riblet valley. These studies provided drag reduction rates of approximately 7-8% and demonstrated that the aspect ratio (h^+/s^+) plays a major role in achieving this [Figure 10]. Further studies have shown that the sharp edges contained within the blade (L) shaped riblet structure permit it to contribute the greatest drag reducing effect, with results in the region of

9-10% [22, 27]. Although much focus has been placed on the above analysis, very few studies have considered the geometries of the riblet valleys. Studies relating to this area by Vukoslavcevic et al. have been shown to measure low velocity flows within the riblet valleys, with the wall shear stress decreasing in the valleys, while simultaneously increasing on the riblet peaks [28].

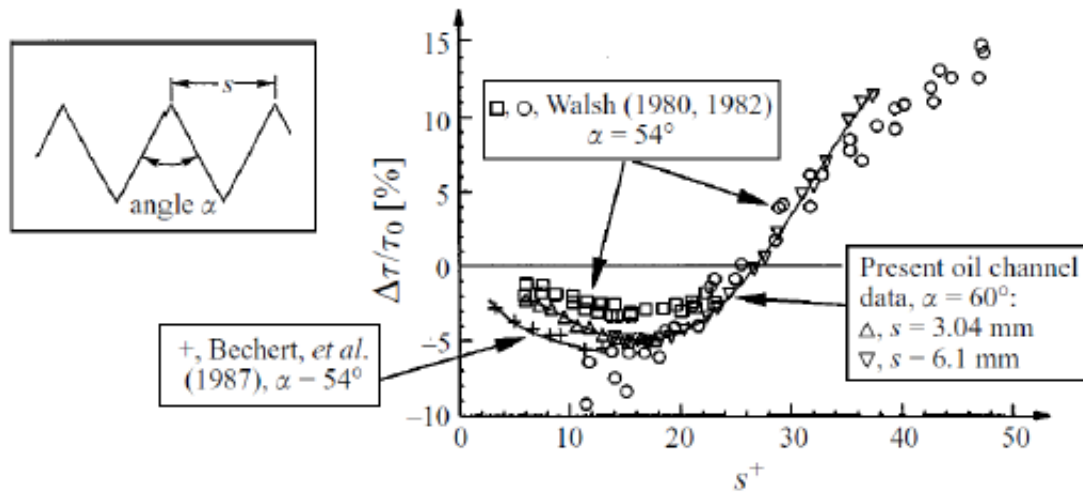


Figure 10, Effects of Sawtooth Riblets, Adapted From [8]

In addition to the commonly experimented two-dimensional riblet structures, three-dimensional patterns including, segmented blade, offset segmented and offset three-dimensional blade, remain understudied [22, 29]. Few attempts, such as those by Bechert et al. (2000) have examined textures and patterns which more closely resemble specific sharkskin properties, generally featuring a series of interlocking segments applied to a surface [Figure 11]. Experiments carried out with wind tunnels have claimed that as with two-dimensional riblet surfaces, a notable drag reduction level can be obtained, however, in contrast they have also advised against their use as practical drag reducing methods [6]. In part, this is due to the drag reduction levels noted being of similar size to those experienced through two-dimensional techniques, therefore, as three-dimensional methods obviously explore a more complicated setup, their increased demand on computational cost and a difficulty in practical application has lead to a redundancy in their use.

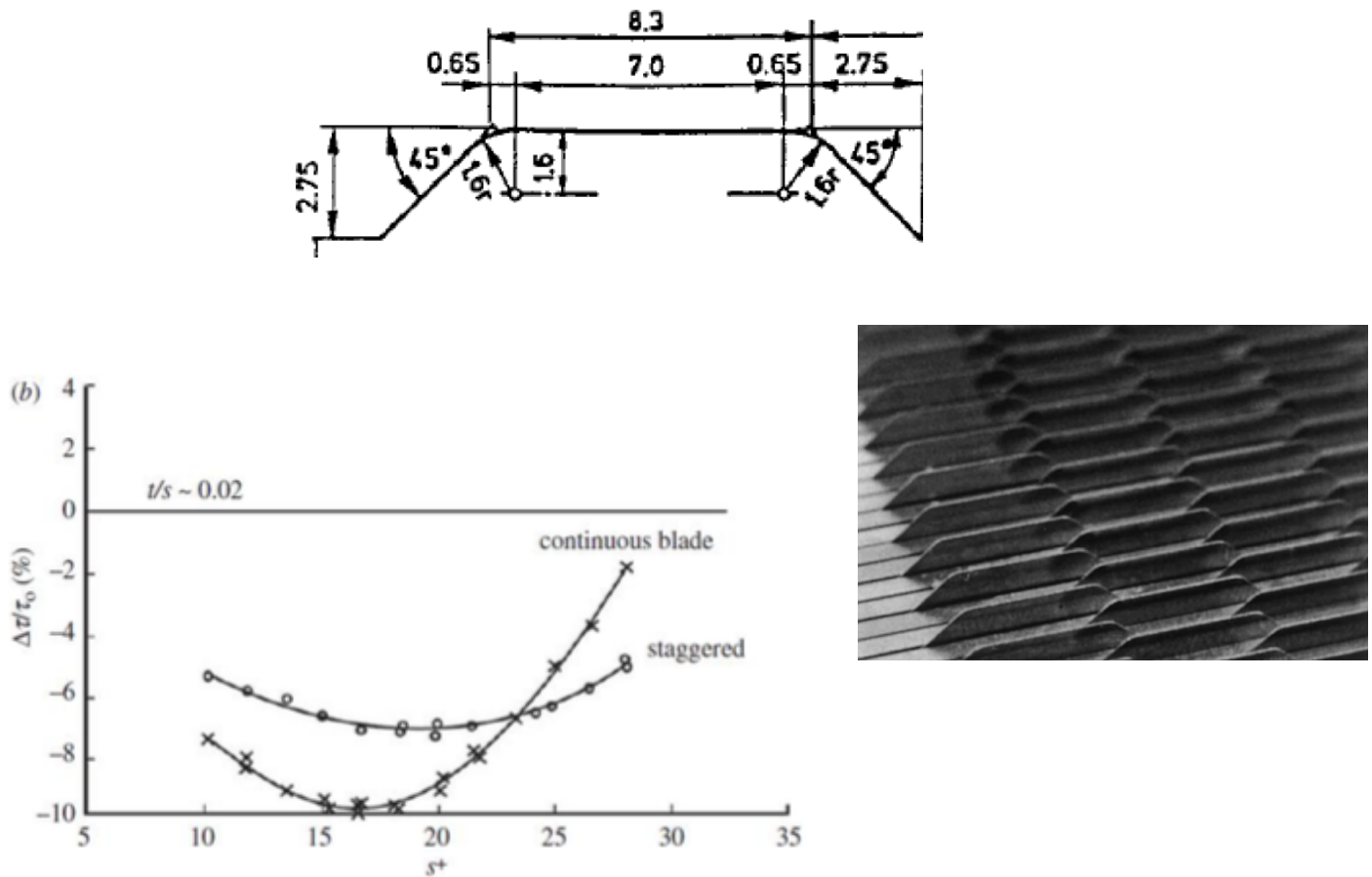


Figure 11, Example of Staggered 3-D Riblet Structure, Comparison of 2-D and 3-D Riblets on Drag Reduction, Adapted from [8, 22]

4.4.3 Examination into Location of Riblet Application on a Body

In addition to the geometrical traits of riblet structures, the most effective location(s) of riblet appliance to the surface of a body must come under scrutiny. The need for this supplementary focus can be further explained when discussed in relation to the specific properties of shark skin. The dermal denticle pattern found on the skin of sharks varies in size and shape, not only between shark species but along the surface of any one shark [22], [Figure 12]. It is for this reason that only the skin patterns found on fast swimming sharks are of concern within this area of study, as although other species may also possess this feature, their skin does not adhere to the drag reduction effects of key interest here. In addition to this, the varied patterns

and locations of riblet covered surface along the body of a fast swimming shark provide insight into the evolutionary optimisation of the sharks natural features. Consequently, a similar logic must be applied when considering the application of riblet surfaces to marine vessels, as the mass covering of such vessels will result in adverse effects on surface resistance and performance. As such, it has been common to study riblet structures in appliance to flat plate geometries [30], to accommodate the necessary spacings and setups for inducing drag reduction. However, practical marine bodies consist a number of streamlined geometries, therefore, study into the application of riblet structures to geometries present on realistic marine vessels, such as the hydroplanes of submarine controls and the vast sections on the sides surface ships which are effectively flat, will allow insight into the commercial viability and practicality of the implementation of riblets to the marine sector to be expressed.

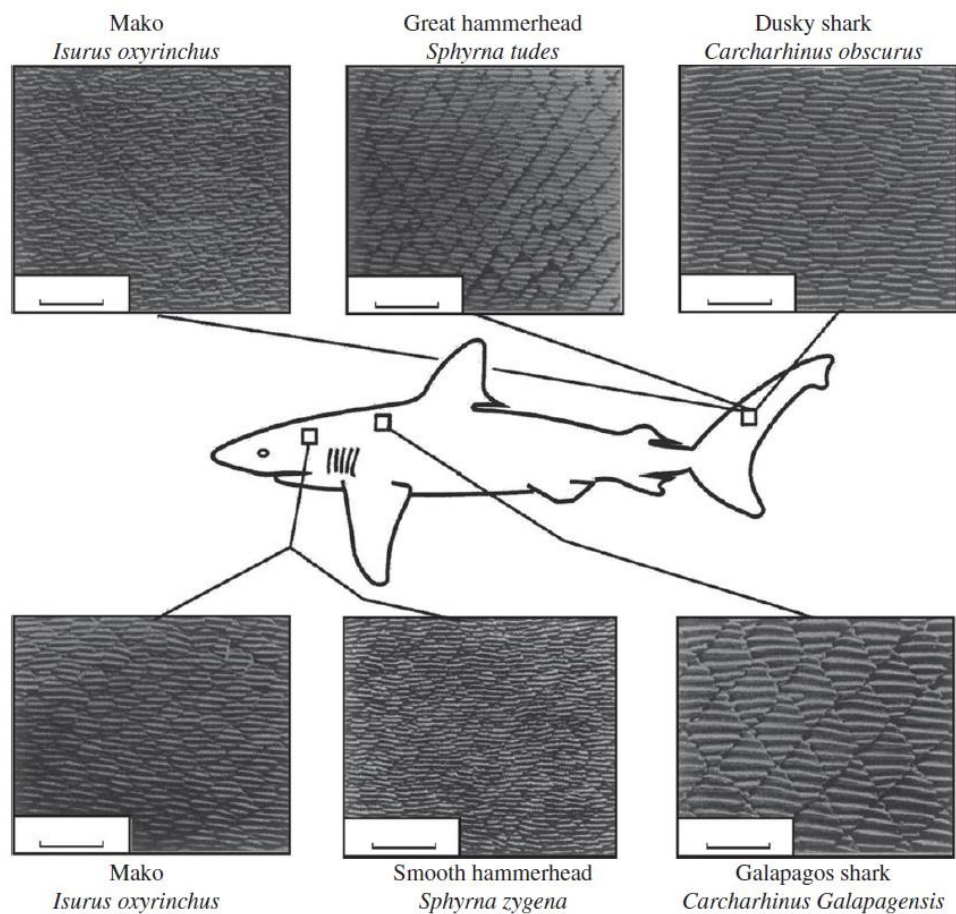


Figure 12, Scale Pattern Variation on Fast-Swimming Sharks, Adapted from [22]

4.5 Theoretical Models for Riblet Study

Computational fluid dynamics (CFD) is primarily exercised for theoretical analysis of flow over riblet structures. Due to the findings of sections 4.2 and 4.3 the study of riblet structures is conducted almost exclusively through use of turbulence models. These can generally be divided into three groups — Reynolds Averaged Navier Stokes (RANS), Large Eddy Simulation (LES) and Direct Numerical Simulation (DNS) — each containing a number of distinct attributes accredited to the attainment of sufficient results, however, it is mainly due to the limitations of a particular model for use on analysing flow over micro-structured surfaces that another has to be selected [Figure 13].

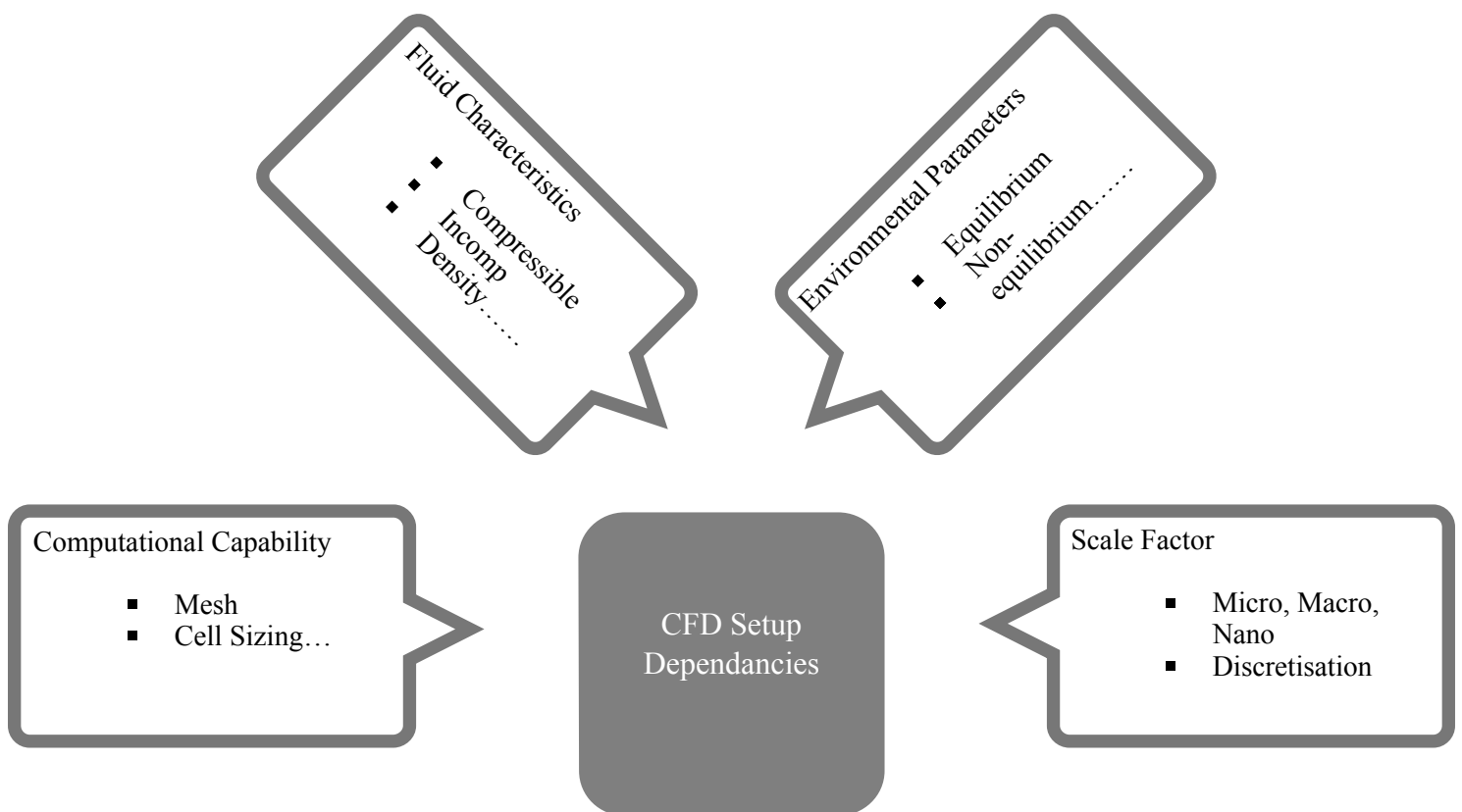


Figure 13, Introductory Aspects of Choosing Particular CFD Method, Adapted from [31,32]

4.5.1 Reynolds Averaged Navier Stokes Models, RANS

The RANS technique takes the average of the Navier Stokes equations of motion over time. As such, their use within the industry for visualisation of flow has been widely tested, and although they cannot provide any time dependent information, they can be utilised for simplistic applications. As a result, a number of previous studies examining the influences of riblets on fluid flow have utilised these equations, commonly stemming from the k- ϵ turbulence model [4, 32, 33]. There have been varied amendments to this model with Djenidi, Launder & Li (1991) all focusing on the lowered Reynolds number k- ϵ turbulence models. These studies focused on the use of curved grid lines running parallel to the riblet surfaces, however, the results were based on riblet geometries on average 2-3 times larger than experimentally optimised sizes, and as such have been criticised in the appropriateness of their use. This increased riblet size is due to the scaling of Reynolds number and the length of body used in providing adequate mixing time of flow [31, 33]. Although much of the past study into flow over riblet structures has utilised the k- ϵ turbulence model, varied RANS models exist as detailed below, with few being of interest in the study of micro-structured surfaces [Table 2].

RANS Based Turbulence Models						
Linear Eddy Viscosity Models					NonLinear Eddy Viscosity Models	Reynolds Stress Model, RSM
Algebraic Models		One Equation Models		Two Equation Models		Explicit Non Linear v2-f Models
Cebeci-Smith Model	Baldwin-Lomax Model	Prandtl's One Equation Model	Spalart-Almaris Model	k-epsilon Models	k-omega Models	
Standard Reasilable RNG Wilcoxs SST						

Table 2, Representation of RANS Models Available for Application, Adapted from [34]

4.5.2 Large Eddy Simulation, LES

The LES approach separates the fluid velocity through the region into a series of large and small scale components [32, 35]. The main reason in desiring this effect is mainly due to small velocity motions having a reduced significance in the overall transport of scalar properties, such as mass and energy. As such, the small scale velocities also provide a reduced accuracy when compared to larger scale velocities. This then leads to an omission of the smallest scales, reducing the effectiveness of this approach in modelling flow over riblets. As such, there are very few studies of riblet structures which have acquired the LES model [32, 35]. A number of LES models can be applied for analysis of any one fluid flow situation, as provided below [Table 3], where assessment of the specific attributes of each model will allow for optimum accuracy in the simulation results attained.

Large Eddy Simulation, LES, Models
Smagorinsky-Lilly Model
Dynamic Subgrid-Scale Model
RNG-LES Model
Wall Adapting Local Eddy Viscosity, WALE, Model
Kinetic Energy Subgrid-Scale Model
Near-Wall Treatment for LES Model

Table 3, Representation of LES Models Available for Application, Adapted from [34]

4.5.3 *Direct Numerical Simulation, DNS*

Lastly, the most progressive approach is the DNS, which solves the Navier Stokes equations of fluid flow without the averaging of a RANS model. As a result, the DNS method will produce a series of non-empirical solutions to a flow simulation. A DNS is then capable of providing a robust and highly accurate simulation, with its main limitation of use stemming from its large computational cost [31, 32]. Therefore, the widespread use of a DNS modelling approach is not generally witnessed as such large scale computational requirements can neither be endured nor justified in the majority of studies. This large computational cost is greatly emphasised when dealing with riblet structures due to the scaling of Reynolds number to flow velocity and applied body length, which can result in large Reynolds requirements, leading to an ever increasing demand on the available computational power. Studies by Corrsin (1961) have related the required number of grid points for a DNS to accommodate a fully developed turbulent flow across the riblet surface. This lead to an increase of $Re^{9/4}$ per time step, and as a complete simulation will require a large number of time steps (in the region of $Re^{3/4}$) current computational power (with the exception of large super computers) cannot cope with a full scale DNS [31, 32]. However, further simplification of the model, such as flow over a riblet applied flat plate, can allow for DNS to take place. As further detail into the differing methods of a DNS (finite and spectral methods) are not applicable to this study such information is not contained within this review. Unlike both RANS simulations and LES a DNS provides a means solving the Navier Stokes equations of fluid flow without application of any specific turbulence models, as such no such models can be provided here.

4.6 Experimental Study of Riblet Structures

Although theoretical methods will be utilised in order to conduct the analysis contained within this study, a brief outline of just some of the experimental techniques created for measurement of flow over riblet structures has been provided in the following table [Table 4], accompanied by graphical representation of some typical setups [Figure 14]. The results generated from experimentation of these techniques has varied in terms of drag reducing effect. However, results for optimised experimental procedures have allowed for similar drag reducing effects to be noted when compared to theoretical methods, reaching levels of 9-10% in some cases [31, 36].

Hot-Wire Anemometry, HWA
Laser Doppler Velocimetry, LDV
Particle Image Velocimetry, PIV
Variable Integral Time Averaging, VITA
Visualisation Techniques

Table 4, Experimental Procedures for Testing Drag Reduction of Riblet Structures

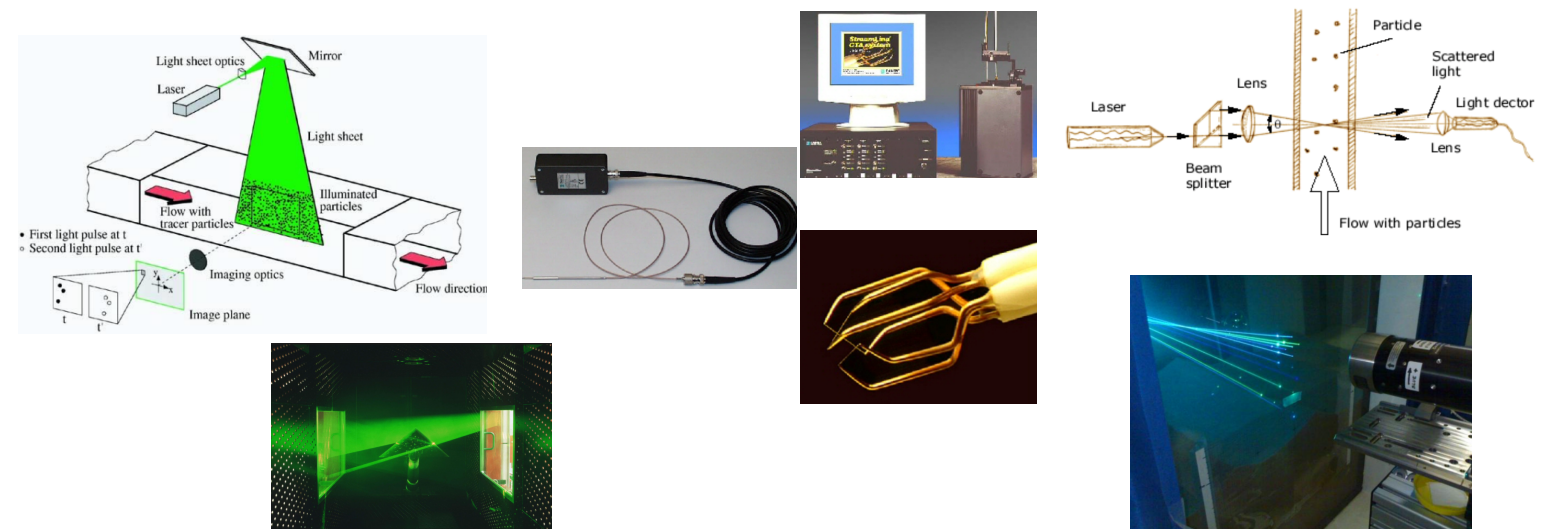


Figure 14, Representative Experimental Procedures for Testing Drag Reduction of Riblet Structures,
Adapted From [37, 38]

ANALYTICAL METHODOLOGY

5.1 Summarisation of Methodological Process

This section of the thesis initialises the analytical objectives of the project, inaugurated by a rudimentary appraisal of the computational fluid dynamics (CFD) software utilised throughout the study. Following this, a varied range of validation and verification test cases are presented, where focus is placed on the geometrical attributes, meshing characteristics and computational models of each, in conjunction with previous publications. In conclusion of the validation and verification process, the methodological procedure deployed for analysis of the shark skin inspired riblet surfaces will be presented, whereby, the riblet structured flat plate configurations are first depicted in view of showcasing the geometry and mesh generation processes exploited, alongside the models utilised for computational analysis. Although the results garnered by the CFD analysis of the flat plate test cases were required in order to determine an optimum drag reducing configuration, prior to implementation on a hydroplane structure, the procedures performed in carrying out analysis of a riblet applied hydroplane are presented within this section, as to consolidate all project results into the following chapter of this thesis [**Chapter. 6**].

5.2 Computation Fluid Dynamics Software

The Ansys Workbench 15.0 [39] advanced engineering simulation technology suite was employed in execution of the required analysis for this project. This suite offers an integrated environment, where its flexible framework permits a diverse spectrum of multi physics analyses to be accomplished with the simplicity of a drag and drop interface [39]. As a consequence, it provides a unified CFD package, encompassing geometry creation and meshing components,

together with fluid modelling configurations and post processing of fluid flow results. This amalgamated system nullifies the need for competence in multiple softwares, which is of particular advantage to new or inexperienced CFD users, allowing focus to be shifted away from study of the software and directed towards attaining the desired results. In addition, the individual packages contained within the suite are of particularly modular design, offering users a vast array of tools and configurations for accommodation of a broad range of model setups, and although the packages apply a vastly automated procedure, a high degree of user defined control can be implemented. These characteristics, among others, have had influence on the utilisation of the Ansys Workbench 15.0 suite within this project, as it has been found to provide a balance between achieving an adequate competency level within any software prior to progression with the necessary steps in attaining advantageous results, and curtailing the computational time required for development of each particular test case, stemming in part from the availability of CFD software within the university department.

5.3 Validation & Verification Test Cases

Validation and verification models are typically an obligatory constituent of any CFD study. The cases selected are deemed suitable in their provision of two main objectives:

- i. Conveyance of the users' ability to accurately and efficiently apply correct configurations in the attainment of sufficiently comparable results to previously published or recored material.
- ii. Initiation of the investigation into the models and CFD dependancies which may be applicable to the imminent study. As such, this allows the user to begin the study with

some introductory knowledge, providing an enlightened starting position and subsequent advantageous effect on the time required in determining valuable results.

Furthermore, although not applicable to this particular study, rigorous validation and verification testing is fundamental in the consideration of any new CFD code, providing insight into the nature of the codes accuracy, while determining the possibilities of its use and widespread application. In order to satisfy the above objectives of the validation and verification process, four distinct test cases were constructed and analysed within this study, namely, flow over a two-dimensional cylinder and two-dimensional airfoil, driven cavity simulation and flow over multiple backward facing step configurations. The cases presented here are universally two-dimensional in their flow characteristics, a factor that has been considered in detail due to the diminished computational demands of two-dimensional flow exploration when compared to three-dimensional problems. Therefore, although the main analysis of this project will utilise a three-dimensional configuration, the elected validation and verification cases nevertheless satisfy the objectives of the process. The subsequent segments of this thesis section provide detail into the geometrical, meshing and computational models implemented on each test case, where comparison and evaluation of their individual results can be found within section 6.1.

5.3.1 Two-Dimensional Cylinder

The two-dimensional circular cylinder is widely adopted for validation and verification purpose due to its simplistic geometrical setup, which has authorised a vast range of varying simulation configurations to be analysed and published [40]. On commencement of this particular case, the geometrical attributes of the cylinder, together with the corresponding flow conditions, are defined by selection of specific Reynolds numbers, as can be identified by [Eq. 1]. Hereby, Reynolds numbers of 1, 20 and 100 have been implemented in the interest of exploiting

the numerous publications associated with this setup. The resultant geometrical, mesh and boundary configurations of the analysed model are shown below [Figure 15].

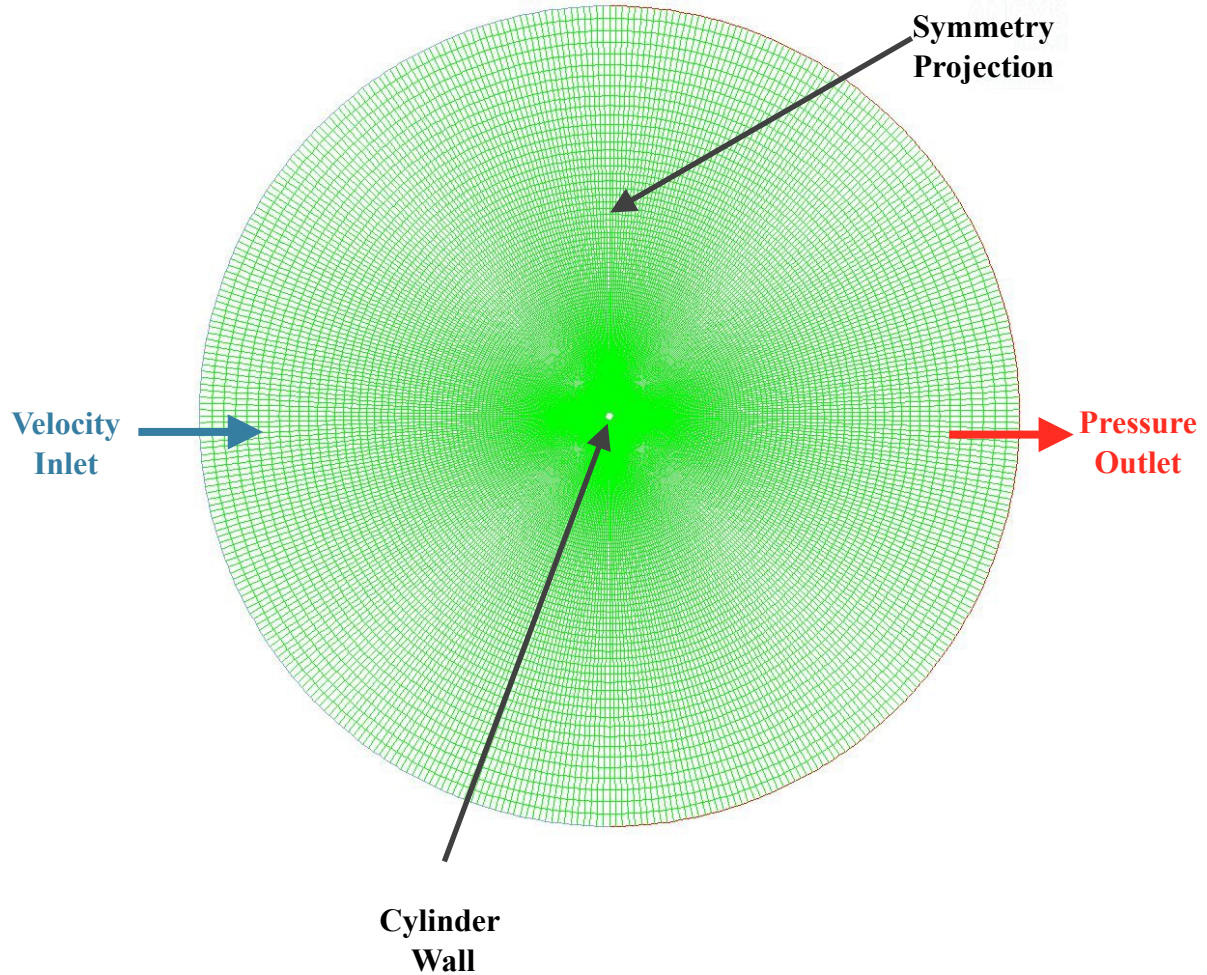


Figure 15, Computational Configuration for Two-Dimensional Cylinder Test Case

In attainment of this configuration, only one major geometrical stipulation had to be followed, with the outer fluid domain boundary created as to be 100 times larger than the cylinder wall diameter. Upon meshing the fluid domain, a mapped face method was adopted with edge sizings and bias options applied towards the cylinder wall, refining the mesh sufficiently enough as to resolve the instigated fluid flow characteristics of the case [Table 5].

Edge	Number of Divisions	Bias Factor
Fluid Domain Boundary	400	No Bias Applied
Symmetry Projection	200	500

Table 5, Characteristics of Two-Dimensional Cylinder Mesh

In conclusion of the meshing procedure, the fluent solver, solution models and solution methods were configured in accordance with the below table [Table 6]. Furthermore, as to permit adequate convergence of the solution [41], all residuals were run to limits of 10^{-6} , with results attained by means of the reported drag coefficients of each individual Reynolds setup.

Fluent Solver	
	Steady Pressure-Based, Absolute Velocity Formulation
Solution Models	
	Viscous-Laminar
Solution Methods	
	SIMPLE Scheme, Least Squares Cells Based Gradient, Standard Pressure, Second Order Upwind Momentum

Table 6, Configurations of Applied Computational Physics for Two-Dimensional Cylinder Test Case

5.3.2 Two-Dimensional Airfoil

Analysing the flow over a two-dimensional airfoil has become a staple of the validation and verification process within many projects, where the effects imparted by the airfoil on the surrounding fluid flow are investigated by application of varied angles of attack [Figure 16]. It was then the intention for this particular test case to simulate flow over a NACA 0012 airfoil profile at a 6 degree angle of attack, with the induced lift and drag coefficients of the case

later scrutinised against published results [Section 6.4.5]. This particular airfoil profile has been selected for testing due to it's simplistic geometrical attributes in comparison to the abundance of airfoil profiles available for study, mainly stemming from the omission of any camber along it's streamwise span [42, 43]. As such, it's geometrical physique permits relations to be developed to the physical framework of a simplistic submarine hydroplane, a factor discussed in greater detail within section 5.5.

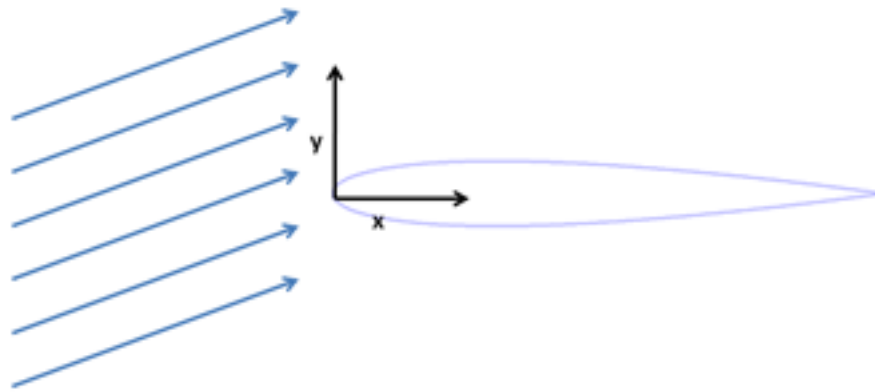


Figure 16, Representation of Airfoil at 30° Angle of Attack to Advancing Fluid Flow, Adapted From [43]

Due to the widespread study of NACA airfoil profiles, the geometry was sourced from a highly dependable NACA database [44], sanctioning coherence between the results attained and published records. The fluid domain was then constrained as to be at a distance of 12.5 times the airfoil chord length away from the profile origin, permitting sufficient development of the generated fluid flow characteristics. The dominant attributes of the fluid domain mesh have been condensed into the below table [Table 7], where a mapped face method was adopted in conjunction with edge bias options, as to refine the mesh in the direction of the airfoil wall [Figure 17].

Edge	Number of Divisions	Bias Factor
Inlet Boundary	200	No Bias Applied
External & Internal Straights	150	50

Table 7, Characteristics of Two-Dimensional Airfoil Mesh

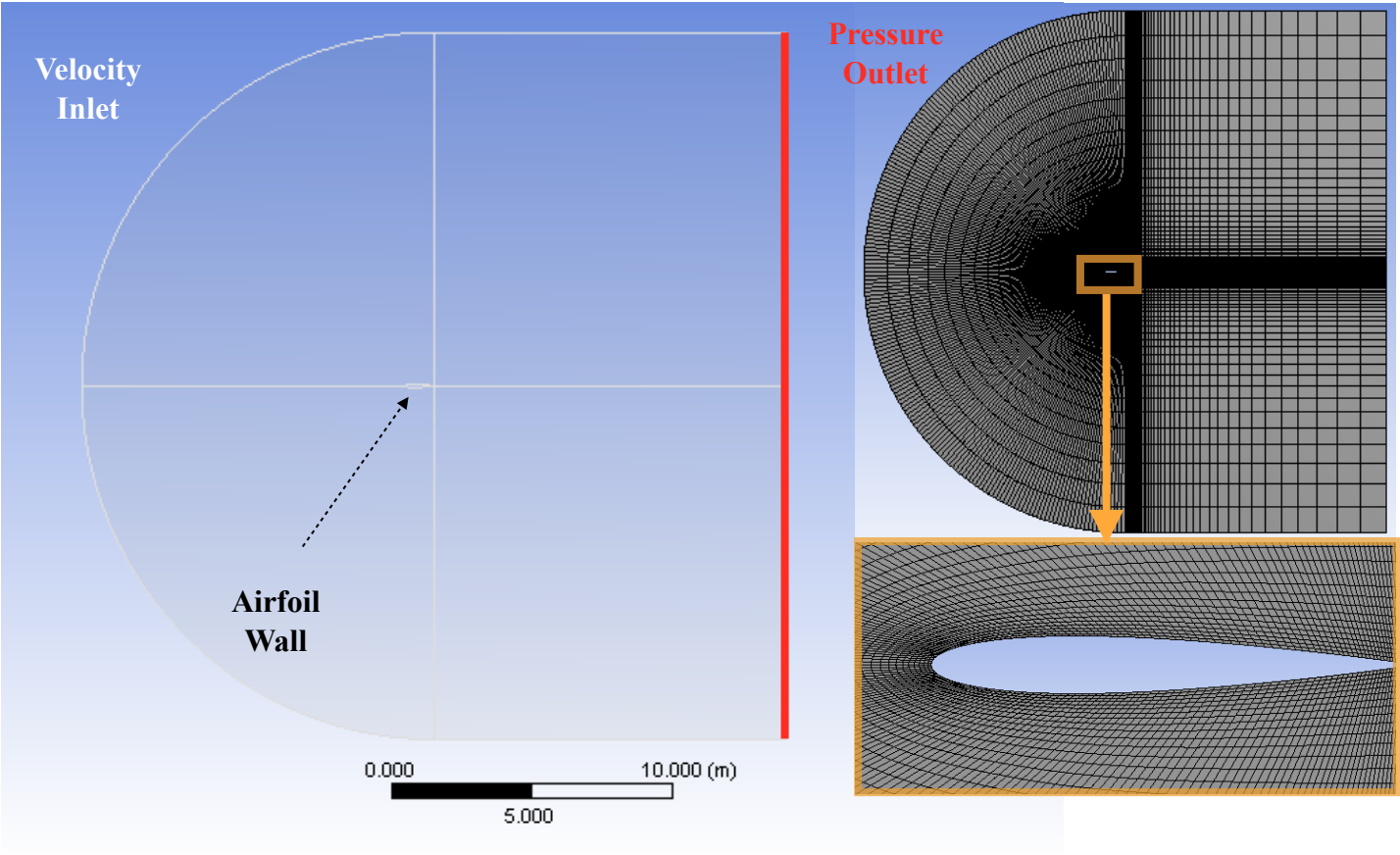


Figure 17, Computational Configuration of Two-Dimensional Airfoil Test Case, Conveying Mesh
Characteristics in Relation to Airfoil Wall

As with the two-dimensional cylinder, the computational configuration adopted has been defined in terms of fluent solver, solution models and solution methods, and outlined through [Table 8], with the case permitted to run until residuals decreased to limits of 10^{-6} .

Fluent Solver

Steady Density-Based, Absolute Velocity Formulation

Solution Models

Viscous-Inviscid

Solution Methods

Implicit Formulation, Roe-FDS Flux,
Least Square Cells Based Gradient, Second Order Upwind Flow

Table 8, Configurations of Applied Computational Physics for Two-Dimensional Airfoil Test Case

5.3.3 *Driven Cavity Simulation*

The simulation of flow within a lid-driven cavity model is considered to be one of the most predominant fluid flow problems studied within the CFD field. This interest in a model that can otherwise be classified as relatively basic, stems from a manifestation of multiple counter rotating recirculating regions of flow within the corners of the square cavity, induced by the positive x velocity of the moving top wall, referred to as the lid [45], [Figure 18].

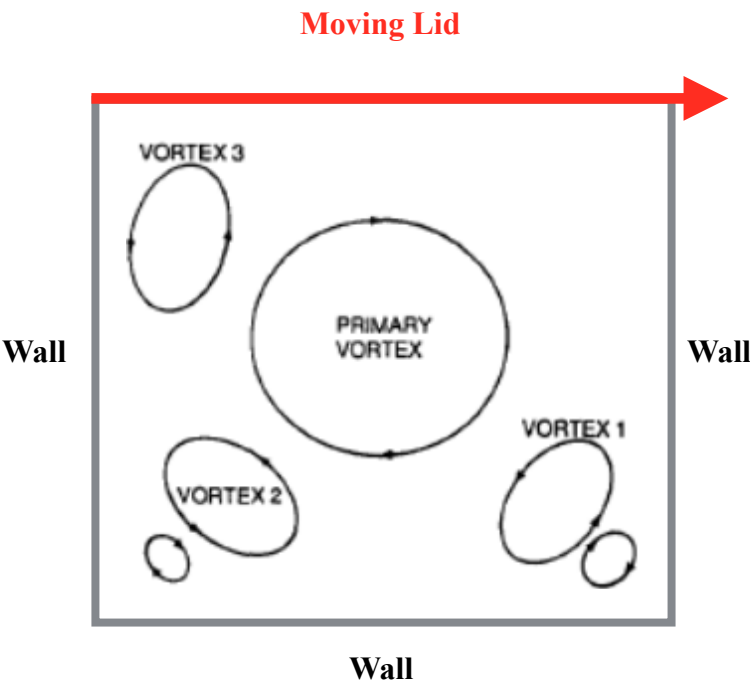


Figure 18, Representation of Vortex Formulation Within Lid Driven Cavity, Adapted From [45]

The regions of flow contained within the cavity model are dependant on the applied Reynolds characteristics, where from [Eq. 1], D corresponds to the length of any one wall, and V can be substituted for the velocity of the lid, allowing a Reynolds number of 1000 to be executed for this particular case. Due to the exceptionally simplistic nature of the geometry, a mapped face method was applied with 350 divisions imparted on the four boundary walls, providing a sufficiently similar setup to comparable studies for later analysis of the produced results [Section 6.1.3]. Then, as with the above test cases, the simulation was run until residuals decreased to levels of 10^{-6} , in accordance with the below computational constraints [Table 9].

Fluent Solver	
	Steady Pressure-Based, Absolute Velocity Formulation
Solution Models	
	Viscous-Laminar
Solution Methods	
	SIMPLE Scheme, Least Squares Cells Based Gradient, Standard Pressure, Second Order Upwind Momentum

Table 9, Configurations of Applied Computational Physics for Lid Driven Cavity Test Case

5.3.4 Backward Facing Step Simulations

Studies into the effects of flow over a backward facing step have been conducted for many years, resulting in its popularity for use in benchmark flow simulations. As a consequence, the scope of published setups and results is vast, with contributions relating to both computational and experimental analysis techniques. The primary interest in examining flow over this geometry is rooted in its formulation of separation regions of flow relative to the

enclosing walls of the computational structure, and the subsequent reattachment regions associated with variations in Reynolds number [46], [Figure 19].

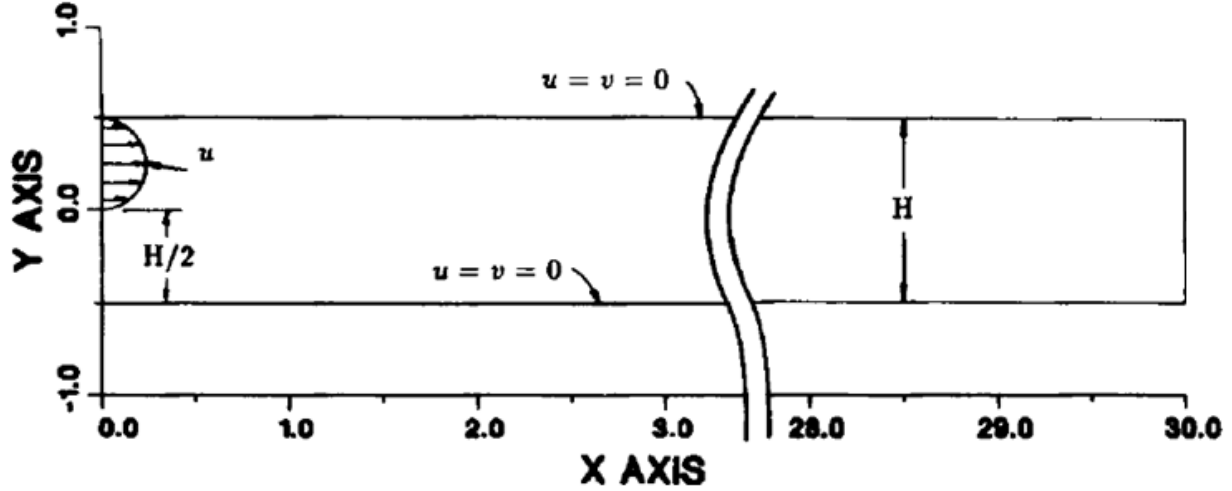


Figure 19, Representative Form of Backward Facing Step Geometry, Adapted From [46]

In order to maintain a configuration with close relations to published studies, laminar and turbulent simulations were run based on Reynolds numbers of 800 (laminar), 93000 and 210000 (turbulent) [46]. The Reynolds characteristics are again formulated in association with [Eq. 1], where D can be substituted for the total height of the flow channel, H . Differentiating the laminar and turbulent geometrical setups is the extended inlet channel required to replicate the parabolic velocity profile associated with a purely laminar flow. As such, the inlet region was extended through application of [Eq. 10], where d refers to the height of the inlet region ($H/2$), ensuring the flow consists a fully developed laminar profile upon journeying the step transition [18].

$$l_e = 0.06dR_e \quad \text{Eq. 10}$$

Then, through application of a mapped face meshing procedure the following characteristic mesh sizings were administered [Table 10], noting that the inlet region size controls were scaled down for the turbulent geometries in accordance with their reduced length, and that bias controls were

issued in support of step refinement. Furthermore, in exploration of reliable results, the following computational dependencies were utilised [Table 11], where a turbulence intensity of 5% was issued for the hydraulic diameter at inlet, and simulations prescribed to run until residuals diminished to levels of 10^{-6} .

Edge	Number of Divisions	Bias Factor
Inlet Region Length	800	20
Flow Channel Length	1600	10
Channel Height	80	No Bias Applied

Table 10, Characteristics of Applicable Backward Facing Step Mesh(s)

Fluent Solver	Steady Density-Based, Absolute Velocity Formulation	
Solution Models	Laminar Case Viscous-Laminar Turbulent Case(s) Viscous-RNG k-ϵ, Standard Wall Function	
Solution Methods	Laminar Case SIMPLE Scheme, Least Squares Cells Based Gradient, Second Order Pressure, Second Order Upwind Momentum Turbulent Case(s) SIMPLE Scheme, Least Squares Cells Based Gradient, Second Order Pressure, Second Order Upwind Momentum, First Order Upwind Kinetic Energy & Turbulent Dissipation	

Table 11, Configurations of Applied Computational Physics for Backward Facing Step Test Cases

5.4 Shark Skin Inspired Riblet Applied Flat Plate Configurations

This section of the thesis concentrates attention towards the geometrical, mesh and physics characteristics pertinent to the fundamental analysis of flow over riblet applied flat plate geometries. Examination into the flow characteristics induced by these flat plate test cases is crucial in delivering shark skin inspired riblet surfaces as a reputable drag reduction method, and as such, the configurations applied within this section can be proven to relay significant effect on the results attained. Furthermore, the results garnered here have direct consequence to the application of an optimum drag reducing riblet geometry for hydroplane use, as is detailed through section 5.5. In result of an amalgamation of these factors, this section of the thesis can be regarded as the most prominent in relation to the primary aims of the project, whereby, the following subsections articulate the necessary procedures in realising the desired results.

5.4.1 *Particulars of the Riblet Geometry*

In conclusion of the literature review process conducted in initiation of this project, the effects of blade (L shaped) riblet configurations have been deemed to support the highest levels of drag reduction, consequently, this simplistic geometry has been adopted for use within this study. Furthermore, through integration of specific non-dimensional riblet characteristics, the effects translated to the surrounding fluid flow have been proven to vary significantly. As such, in order to utilise the wealth of past research available in this area of study, three variations of the blade riblet geometry have been produced here, with the non-dimensional lateral spacing, s^+ , taking priority, and constant ratios of height/spacing and thickness/spacing implemented in determination of the final riblet configurations ($h/s = 0.5$, $t/s = 0.02$). The lateral spacings selected for application within this project [Table 12] have been determined as to relate closely with published studies, allowing realistic comparison of results to be accomplished [30].

Model Reference	Lateral Spacing, s^+	Height, h^+	Thickness, t^+
Model 1	15	7.5	0.3
Model 2	25	12.5	0.5
Model 3	40	20	0.8

Table 12, Non Dimensional Spacing Configurations of Riblet Applied Flat Plate Models

Further detail as to the geometrical configurations of the riblet plate(s) is contained within the following subsection [Section 5.4.2], as this must be considered in relation to the fluid domain.

5.4.2 Geometrical Attributes of the Fluid Domain

As was the case with the aforementioned validation and verification test cases, the geometrical properties of the fluid domain are constrained in relation to the characteristic Reynolds number applied to the flow situation. Then as to allow dependable comparisons of the recorded results to be made in relation to published studies, a Reynolds number of 4200 was utilised for all three model configurations. In addition to permitting coalition of results, this Reynolds number is characteristic of turbulent flow within a fluid channel, allowing the effects of both laminar and transitional flow to be neglected, and focus applied to the escalation of turbulence within the fluid domain. The development of turbulence within the fluid domain was aided by specification of domain sizes, whereby, a ratio of the domain width (δ) to height (2δ) to length (3δ) was applied [47]. With establishment of an applicable Reynolds characteristic, domain geometry and non-dimensional lateral spacings, the physical attributes of the riblet structure were identified by means of the following equations [Eq. 11, 12, 13, 14, 15], noting that the characteristics present here have been adapted from [30], with all expressed nomenclature available within the corresponding section at the beginning of this thesis.

$$Re = \frac{3Q}{4\nu L_x} = \frac{U_l \delta}{\nu} \quad \text{Eq. 11}$$

$$H_d = \frac{4A}{P} \quad \text{Eq. 12}$$

$$\tau_o = 0.03955 \nu^{\frac{1}{4}} \rho V^{\frac{7}{4}} H_d^{-\frac{1}{4}} \quad \text{Eq. 13}$$

$$V_\tau = \left(\frac{\tau_o}{\rho} \right)^{0.5} \quad \text{Eq. 14}$$

$$s = \frac{s^+ \nu}{V_\tau} \quad \text{Eq. 15}$$

In conclusion of the above theoretical calculation procedure, the geometrical configurations of the three riblet applied flat plate test cases, along with the encompassing fluid domains are derived. This resultant setup can be viewed through the below figure [Figure 20], where the riblet structured surface is subtracted from the bottom of the fluid domain in opposition of the smooth surface remaining on the top. This setup was issued as to allow direct comparison of the fluid flow characteristics induced by the two surfaces, where particular inquisition was placed on the variations of wall shear stress in determining a reduction of surface friction resistance.

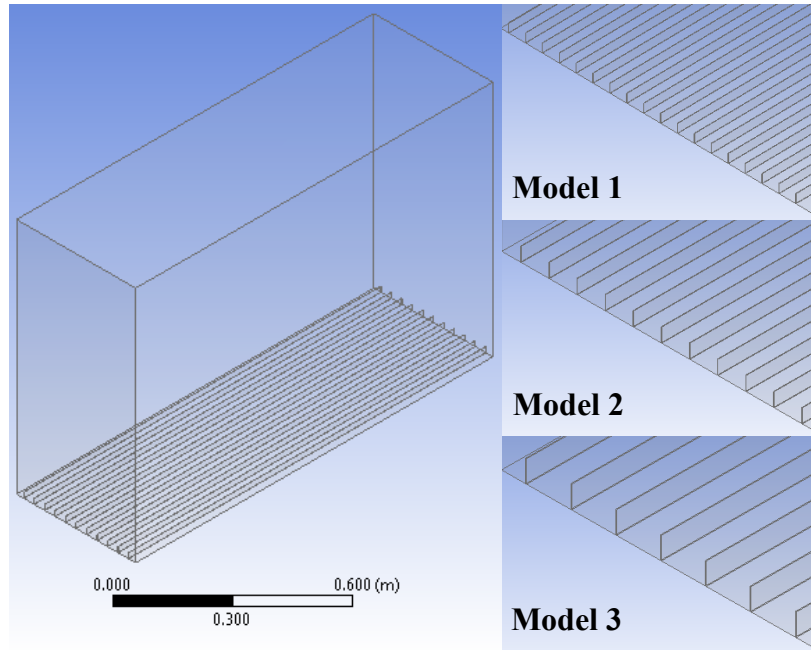


Figure 20, Geometrical Configurations of Riblet Applied Flat Plate Test Cases

5.4.3 *Characteristics of the Applied Mesh*

The principal stipulation applied in generation of the mesh of each fluid domain focused on the requirement of sufficient refinement over the top and bottom surfaces, with particular focus on the region surrounding each shark skin inspired riblet surface in prescription of ample accuracy of the encompassing fluid regime characteristics. However, this had to be met in conjunction with the limiting nature of the academic Fluent solver license utilised for CFD analysis of the models, where a maximum of 512,000 meshed cells are permitted for any serial simulation. Therefore, in adherence to these conditions both unstructured and structured mesh types were trialled, allowing the compromise of mesh refinement properties in relation to the number of generated mesh cells to be evaluated. In initiation of the meshing process an unstructured mesh was developed for each model through application of specific edge sizings to the front face of the geometry, with all bias factors applied for near wall refinement [Table 13]. This face was then swept through each corresponding fluid domain as to provide coherence in the mesh properties when compared at various planes normal to the streamwise direction of flow. A sweep parameter of sixteen evenly spaced divisions along the streamwise direction of the fluid domain was utilised in conjunction with previous studies, whereby, altering the parameter for increased mesh refinement has not been found to significantly improve the solutions garnered [30].

Edge	Number of Divisions	Bias Factor
Fluid Domain Height	200	150
Individual Riblet Height	25	10
Individual Riblet Thickness	5	No Bias Applied
Spacing Between Riblets	30	15

Table 13, Specification of Endorsed Unstructured Mesh for Riblet Applied Flat Plate Test Cases

The resultant mesh formed in conclusion of the above procedures was found to be inaccurate in its resolution within the riblet structured surface boundary layer, stemming from the requirement of diminutive y^+ when considering flow over such structures, where it is common to evaluate the flow at a y^+ of unity [48]. This inexactness of the mesh becomes distinct when conveyed in the following figure [Figure 21], leading to the requirement for revision of the mesh generation process.

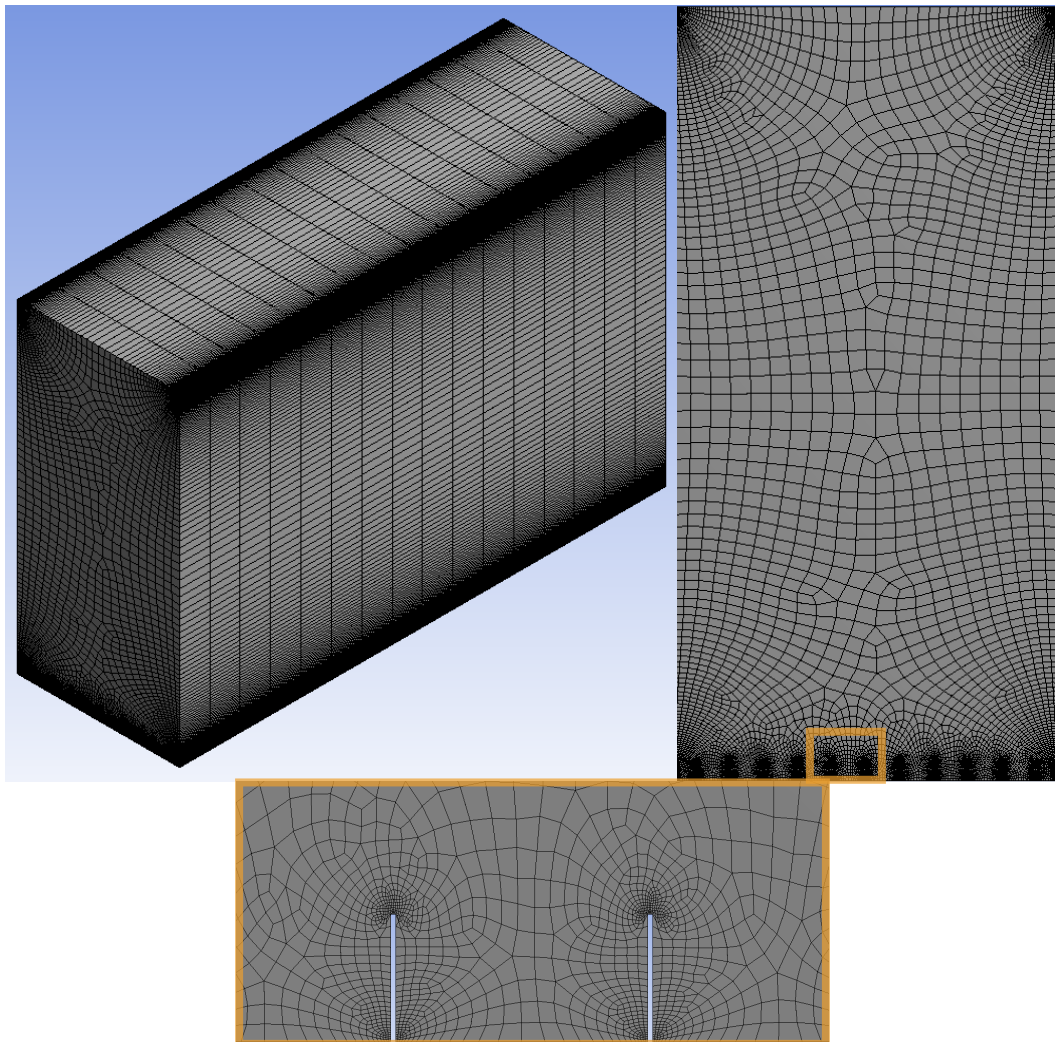


Figure 21, Generated Unstructured Mesh of Model 2 Fluid Domain, Showcasing Domain Exterior and Front Face, with Closer Examination of Region Surrounding Individual Riblet Blades

In response to the above unstructured mesh, it was concluded that a structured configuration would provide greater definition in the region surrounding not only the riblet structured surface but also in relation to the smooth surface of the top wall. As such, the meshing process was reinstated with the inclusion of a mapped face method in appliance to the front face of the geometry. Although initially produced to the above specifications of [Table 13] this culminated a mesh with an exceedingly vast amount of cells, beyond the scope of the utilised software. As such, the individual spacings were altered as to retain mesh refinement towards both the riblet structured and smooth surfaces, while coarsening the mesh within the majority of the fluid domain. Further to this, the individual edge sizings of each model were altered as to capture the geometrical attributes of the particular case, for example, the riblet dimensions of Model 1 are characteristically less prominent within the fluid flow when compared to those of Model 3, with the specified sizings detailed through the below table [Table 14].

Edge	Number of Divisions	Bias Factor
Fluid Domain Height	50	50
Individual Riblet Height	25-30	10
Individual Riblet Thickness	3-5	No Bias Applied
Spacing Between Riblets	25-30	15

Table 14, Specifications of Endorsed Structured Mesh for Riblet Applied Flat Plate Test Cases

In conclusion of this process the eventual mesh configurations were deemed to be suitably accurate for the capture of characteristic properties contained within the related fluid flow, with each consisting a purely hexagonal mesh in the range of 309600-501600 cells (Model 1 - Model 3), as is exhibited through representation by the attributed mesh of Model 2 [Figure 22].

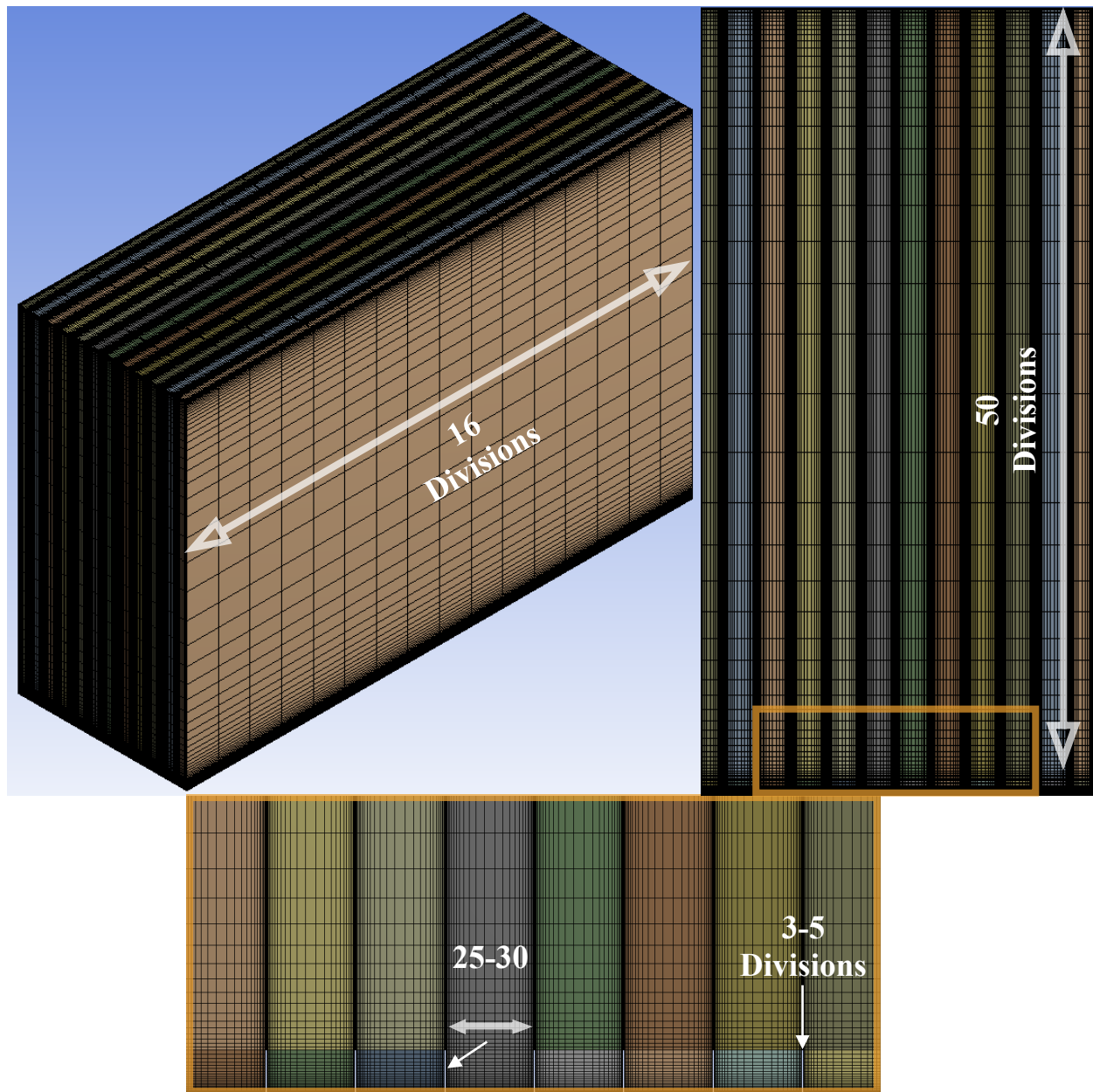


Figure 22, Generated Structured Mesh of Model 2 Fluid Domain, Showcasing Attributable Edge Sizings

5.4.4 Configurations of the Computational Physics

In realisation of results of adequate accuracy, a number of simulations were permitted through variations in their computational configuration. The details of these simulations have been consolidated into the succeeding subsections, where rationalisation of the setups utilised is contained.

5.4.4.1 *Fluent Solver*

It can be concluded by means of sections 4.2 & 4.3 that the induced turbulence of flow contained within the fluid domain prescribes to a random and chaotic nature of fluid molecules, culminating in the creation of secondary vortices along the spanwise direction. In view of witnessing the formulation of these vortices a transient simulation was required as to allow the solutions to be recorded at discontinuous moments in ‘real’ time, in contrast to the continued formulation of the solution attained through a steady computational solver. Therefore, in following a transient simulation a computational time step was issued in adherence with previous studies, outlined by [Eq. 16], with the number of iterations solved per time step standardised to twenty.

$$\frac{\Delta t U_l}{\delta} = 0.05 \quad \text{Eq. 16}$$

5.4.4.2 *Solution Models*

The selection of any turbulence model has significant consequence on the results attainable from the corresponding simulation, clarifying the application of varied configurations throughout this study. Upon commencement of the computational simulation process, the k- ϵ turbulence model was initially selected in reference to previous studies of flows over shark skin inspired surfaces. However, the final results of this study have been taken through application of large eddy simulations (LES), due to the perceived accuracy in the velocity plots of this model in comparison to those presented by the k- ϵ turbulence model. Furthermore, although both models provided similar outcomes in collation of the integral wall shear stress present over both the smooth and riblet applied surfaces, the attributes of the fluid flow contained within the LES were deemed to have greater characteristic relation to previous studies.

5.4.4.3 Boundary Conditions

The applied boundary conditions were formulated in exploitation of their contributions towards the development of turbulent flow within the fluid domain. As such, translationally periodic boundary conditions were opted, removing the effects of any wall conditions and simulating the fluid flow into a repeating nature, where the fluid particles exiting one face of the looped periodic boundary reenter through the relative second face. In adherence to this condition, the inlet and outlet, as well as the left and right faces of the fluid domain were modelled in such as way, resulting in the stipulation of a mass flow rate through the domain in correlation with the applied Reynolds characteristic of section 5.4.2. This then permitted focus to be directed towards the flow characteristics instigated by the top and bottom faces of the domain, which were modelled as stationary walls with specified no slip conditions. The resultant configuration of the applied boundary conditions is expressed through the below figure [Figure 23].

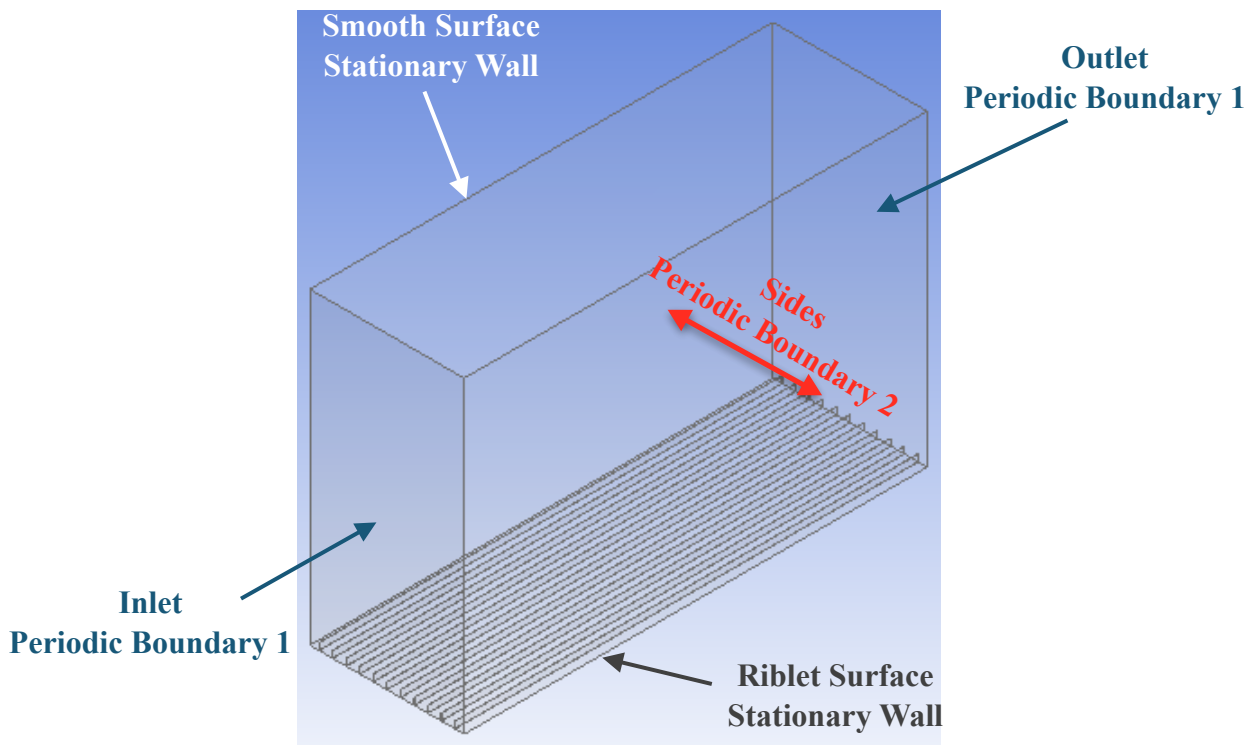


Figure 23, Representation of Applied Boundary Conditions for Each Riblet Applied Flat Plate Test Case

5.4.4.4 Solution Methods & Applicable Monitors

In the aim of deploying minimal computational power while minimising the time of each model simulation, the defaulted solution methods associated with a LES were applied. This arrangement was met with the exception of transient formulation, where the bounded second order implicit option was issued for increased control of the respective time step. Further to these conditions, surface monitors were created for both the smooth and riblet structured walls of the domain, attendant to the integral wall shear stress in occurrence at each time step of the flow simulation. These monitors then allow for direct comparisons of the integral wall shear stress between the two walls, with the subsequent drag variations determining the effectiveness of the particular riblet model. In accordance with the previously detailed validation and verification test cases, the simulations were permitted to run in accordance with an absolute residual criterion of 10^{-6} , with initialisation of the solution indicated by zeroed conditions of flow, namely, zero values of x, y and z velocity. In summary of the above articulations of computational setup, the concluding configuration of each case is outlined below [Table 15].

Fluent Solver

Transient Pressure-Based, Absolute Velocity Formulation

Solution Models

Viscous-LES-Smaorinsky-Lilly

Solution Methods

SIMPLE Scheme, Least Squares Cells Based Gradient,
Standard Pressure, Bounded Central Differencing Momentum,
Bounded Second Order Implicit Transient Formulation

Table 15, Configurations of Applied Computational Physics for Riblet Applied Flat Plate Test Cases

5.5 Hydroplane Application by Optimised Riblet Structure

Expression of the procedures applied for inclusion of a predetermined riblet structure for application towards a hydroplane geometry are provided within this thesis section. As such, the results produced in analysis of the riblet applied flat plate test cases introduced through section 5.4 have been evaluated in terms of their individual influences on wall shear stress, with the subsequent impact of each on skin friction drag detailed by section 6.2. The outcome of this evaluation has put forth an optimum riblet configuration for application here, initiating this section of articulation. The succeeding subsections have been formulated in coalition with the previous section [Section 5.4] in exploration of the geometrical, mesh and physics characteristics in appliance, with the view of depicting said riblet structure as a desirable drag reducing method, noting that all results have been confined within section 6.3.

5.5.1 *Particulars of the Hydroplane Geometry & Surrounding Fluid Domain*

In aim of generating a hydroplane geometry of sufficient resemblance to realistic structures, while minimising the complexity of the overall simulation, the NACA 0012 airfoil profile was selected for CFD investigation. As has been previously mentioned within section 5.3.2 this provides the most simplistic of airfoil profiles, permitting analysis to be completed in accordance with both the computational and time constraints inherent to this study. Alterations were not applied to the two dimensional geometry profile itself, as to retain relations to published results and allow coherence in the solutions attained here to those of the validation and verification test case. However, in order to model the airfoil as a hydroplane with attached riblet structured surface, in turn simulating its resultant effects on the surrounding fluid flow, an obligation to three dimensional analysis was enforced. The geometrical configuration of the three dimensional hydroplane was modelled in accordance with published data regarding the

geometrical constraints of such bodies, where it was found that the ratio of hydroplane chord to width varies indiscriminately with the particular case [49, 50]. In consequence, the geometry utilised for the two dimensional validation and verification test case was remodelled in a three dimensional form, with the main precondition stemming from the ratio of profile chord to width (1:2). In further alliance with previous research, the geometrical constraints of the fluid domain were issued as to permit adequate development of the induced fluid flow characteristics, where increased significance is placed on allowing contours of both velocity and pressure to be plotted in their entirety (too small a fluid domain surrounding the hydroplane geometry will intersect the contour plots even if the applied boundary conditions permit accurate results at near wall level). In conclusion, the following geometrical configurations of both airfoil and fluid domain were deemed to satisfy all necessary conditions [Figure 24].

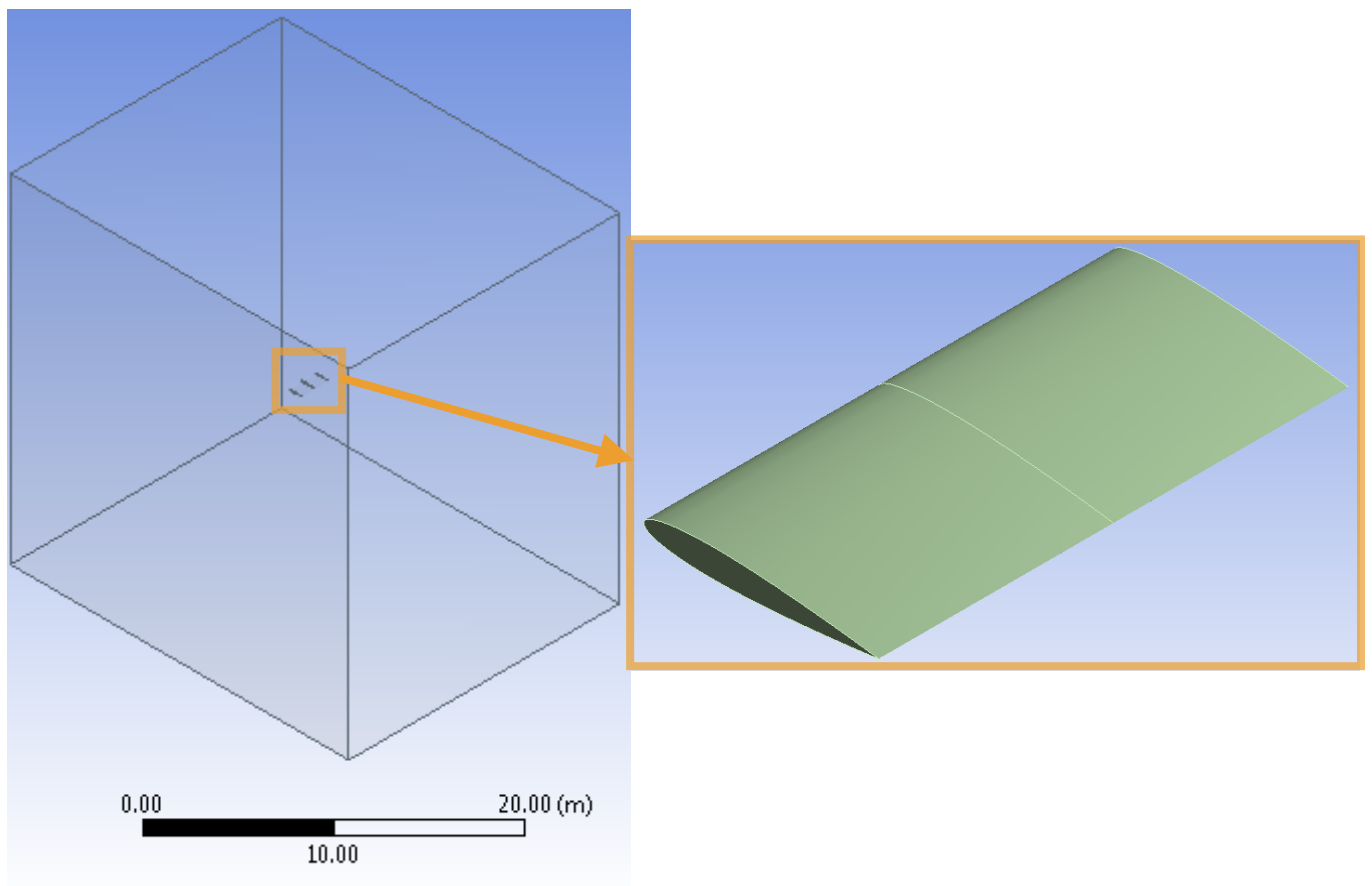


Figure 24, Geometrical Representation of Hydroplane Body & Corresponding Fluid Domain

5.5.2 *Particulars of the Riblet Geometry*

Evaluation of the riblet applied flat plate test cases culminates in the introduction of a particular geometrical riblet configuration for advantageous integral wall shear stress reduction, as is detailed exclusively through section 6.2. As such, the characteristics of the applied riblet structure in reference here have been modelled as to adhere to this optimum configuration, where Model 2 of section 5.4.1 can be found to transmit the greatest drag reducing effect. In compliance with this configuration, the theoretical procedure outline by [Eq. 11-15] is further applied in aim of deriving the physical attributes of the riblet structure, where reliance is placed on conforming to the characteristic lateral spacing ($s^+ = 25$ for Model 2). The riblet structured surface was initially modelled through this process by stipulation of a Reynolds number of one million, as to simulate an externally flowing fluid of characteristically turbulent regime. However, due to the increased physical size of the applicable geometry and the subsequent fluid domain required for this particular simulation when compared to that of the domain utilised for each flat plate test case, the blade riblet structure was produced to microscopic proportions. A direct consequence of the resultant geometry was in the exceedingly vast number of cells required in order to mesh the surrounding region of the fluid domain, far in advancement of the capabilities of the processing units applied for computational simulation. In the interest of reducing the meshing demands, scaling of the Reynolds characteristic was enforced, whereby, a Reynolds number of 4200 was established in conjunction with the flat plate test cases. With significant reduction of Reynolds number, an increase in the physical dimensions of any riblet configuration through reapplication of the theoretical process outlined by [Eq 11-15] is permitted, consequently reducing the number of meshed cells generated. In addition to the individual dimensions of the riblet structure, the position of application to the hydroplane geometry was also considered. In conclusion, the leading edge of the riblet structure in relation to the corresponding

fluid flow was constrained to the position of maximum thickness of the enveloped hydroplane, as to capture the effects of a variant fluid flow velocity evident to this position through previous studies when simulated at zero angle of attack **[Figure 25]**. The corresponding trailing edge of the riblet structure was then deemed to end slightly prior to the trailing edge of the hydroplane, in analysis into the effects of the trailing edge on drag reduction.

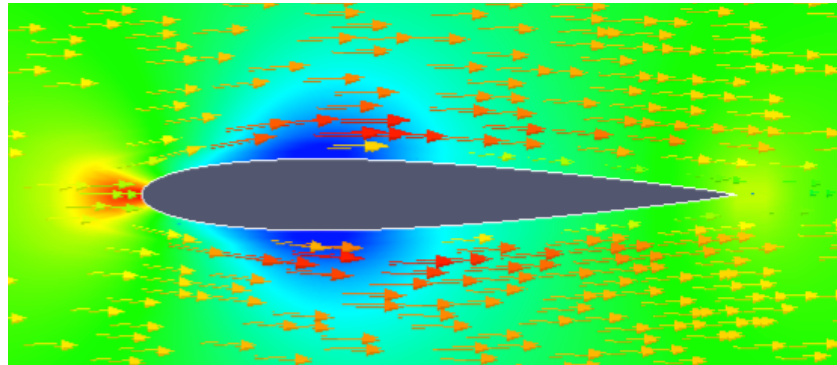


Figure 25, Velocity Magnitude Contour Plot of Airfoil Simulation at Zero Angle of Attack, Adapted From [43]

Summarisation of the above contributions of geometrical constraint, result in the simulated riblet applied hydroplane geometry, as is conveyed by the ensuing figure **[Figure 26]**.

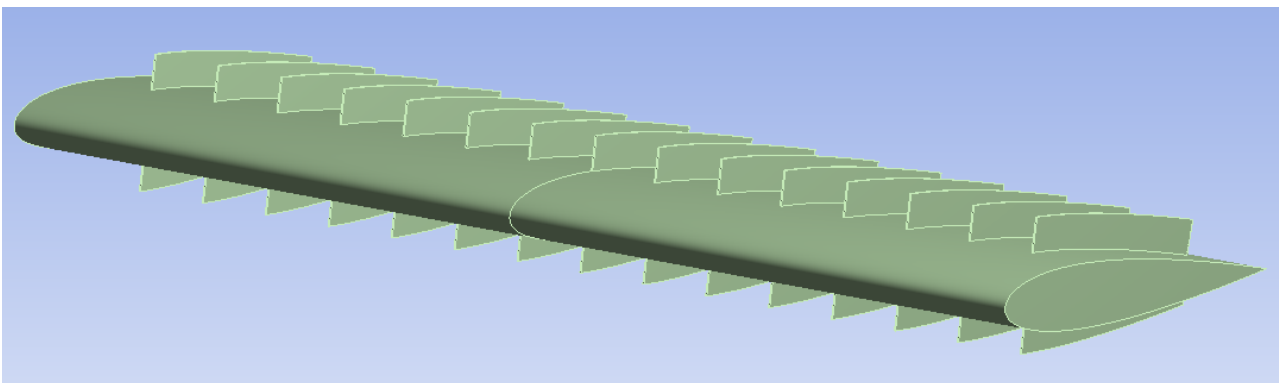


Figure 26, Geometrical Representation of Riblet Applied Hydroplane Body

5.5.3 *Characteristics of the Applied Mesh*

As has become evident throughout this study, there is a demand for the imposition of sufficient mesh refinement properties in relation to both the geometry of evaluation and the characteristics of the simulated fluid flow. The meshing procedures obeyed throughout this section have therefore been applied in establishment of adequate mesh refinement characteristics within the fluid domains, with respect to the hydroplane geometries. Initially the smooth surfaced hydroplane test case was meshed through application of a cut cell assembly method, whereby, automatic inflation layers are generated in the region surrounding the geometry at hand. This provision of automatic layers within the fluid domain permits advantageous effect on not only the results attainable in permission of characteristic fluid flow development, but also towards the number of generated cells within the meshed volume. However, the accuracy of this method in regard to the narrowing clearance present between the upper and lower walls of the hydroplane trailing edge, culminated in the disagreeable generation of cells within this region of nonexistent body (stemming from the subtraction of the hydroplane geometry from the fluid domain — cells should only be created within the domain and not the subtracted hydroplane itself). Further meshing applications were trialled with prioritisation of the mesh region surrounding the hydroplane geometry, as to consider the individual levels of accuracy of each method. In result, an unstructured mesh was developed for the model through application of mapped face procedures in conjunction with specific edge sizings to the hydroplane structure. The volume mesh produced in result of this process was deemed to satisfy the necessary conditions applicable to three dimensional analysis of airfoil structures, with the exception of the number of cells produced in regard to the utilised software limitations. As has been previous stated, the number of meshed cells authorised for transmission of the mesh to the educational Fluent solver license is restrained to 512,000 for any serial simulation. In consequence, comprehensive investigation into

the number of cells generated within the fluid domain was conducted, in aim aiding the computational time and subsequent cost of simulations containing such vast volume meshes. A parallel simulation was configured in resolution of this issue, permitting a finer mesh with substantially increased cell count to be analysed by the fluent software, with the hindrance of an increased computational time outweighed by the possibility of increased accuracy in the developed fluid flow characteristics. The concluding properties of the applied edge sizing features are outlined by the below table, with the succeeding figure conveying the resultant mesh for the smooth surfaced hydroplane case, containing approximately 609,000 meshed cells [Table 16, Figure 27].

Edge	Number of Divisions	Bias Factor
Leading/Trailing Edge	100	No Bias Applied
Curved Side Edges	100	No Bias Applied

Table 16, Characteristics of the Endorsed Unstructured Mesh for Smooth Surfaced Hydroplane Model

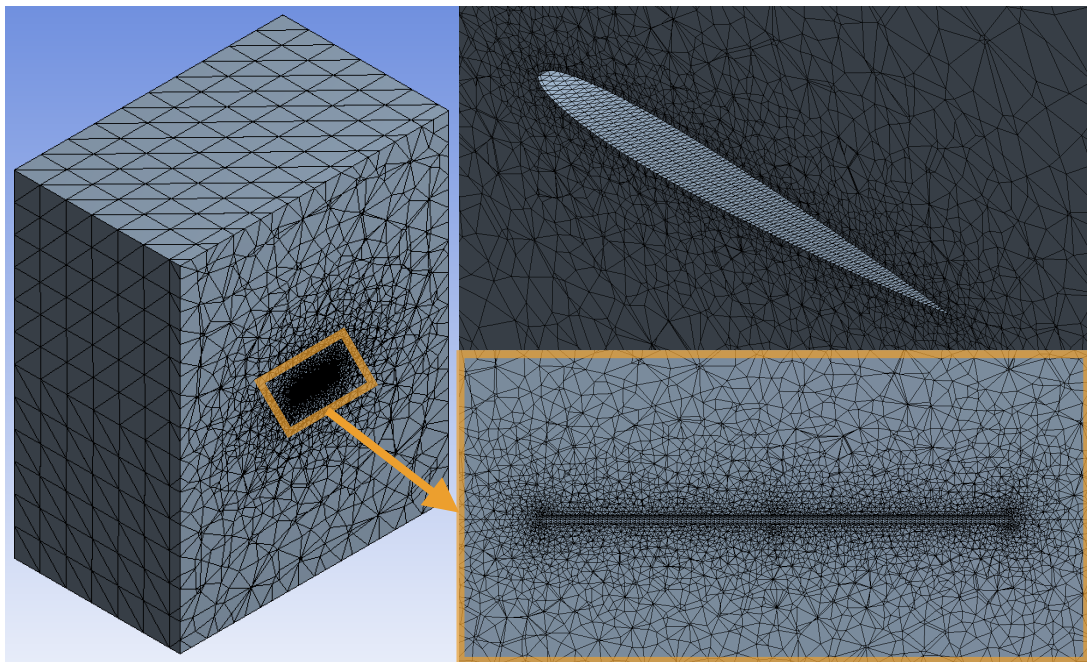


Figure 27, Generated Unstructured Mesh for Smooth Surfaced Hydroplane Model

In the interest of sanctioning useful comparison of the derived results from the smooth surfaced to riblet applied hydroplane cases, the meshing process was instated within the riblet applied hydroplane model through adherence to the above procedures. Furthermore, the edge sizings in application within the flat plate test cases of section 5.4.3 have been considered for incorporation here, with reduced magnitudes in the individual spacing characteristics opted for relief of the generated mesh cell count. In result, the applied sizing configurations can be viewed by the ensuing table, with the corresponding mesh characterised by the successive figure [Table 17, Figure 28], accountable to approximately 2.1 million cells.

Edge	Number of Divisions	Bias Factor
Leading/Trailing Edge	100	No Bias Applied
Curved Side Edges	100	No Bias Applied
Individual Riblet Thickness	4	No Bias Applied
Individual Riblet Height	20	10

Table 17, Characteristics of the Endorsed Unstructured Mesh for Riblet Applied Hydroplane Model

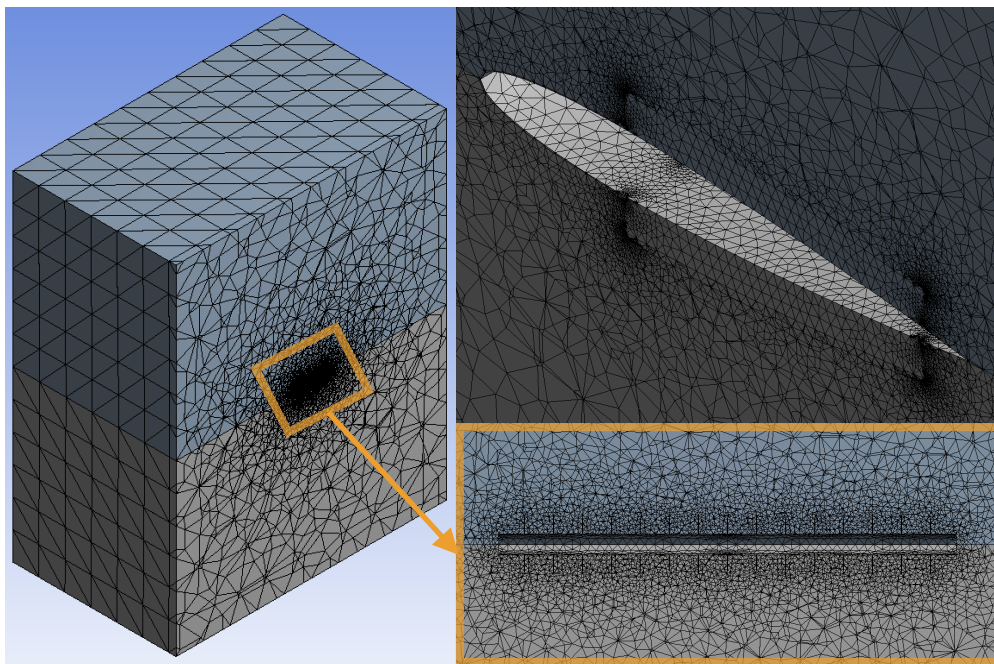


Figure 28, Generated Unstructured Mesh for Riblet Applied Hydroplane Model

5.5.4 Configurations of the Computational Physics

The configurations of computational physics prescribed for the preceding hydroplane test cases have been outlined within this section, analogous with section 5.4.4, where focus is again placed on the attainment of results of adequate accuracy and comparability to previous publications. It must then be noted that both hydroplane models — smooth and riblet applied — have been simulated by duplicate configurations, providing resultant juxtaposition of results.

5.5.4.1 Fluent Solver

The solver setup issued for CFD analysis of the hydroplane models has been correlated in appliance with the setup utilised for the riblet applied flat plate test cases, as although the physical geometries differ entirely, the desired characteristics of the fluid simulation remain intact. This can be perceived by the admission of turbulence within the fluid domain, where a transient simulation will again benefit the formulation of turbulent characteristics, permitting the influence of the riblet structured surface on both lift and drag forces to be translated. In further similarity with section 5.4.4.1 the time step in application was administered through practise of [Eq. 16], due to the acknowledgements of this relation within previous studies. However, by cause of the transcendent nature of the simulation on the computational time required, the number of time steps was reduced in view of scaling back the operation, as is further discussed through section 6.3. Also in accordance with previous studies, a standard of 20 iterations were solved per time step, permitting adequate convergence of the flow simulation.

5.5.4.2 Solution Models

As previously stipulated by section 5.5.2 the simulated fluid flow within each hydroplane model was initially desired to satisfy a Reynolds number of one million, due to the inducement of a purely turbulent, externally flowing fluid at such a characteristic. With this in mind, the k- ϵ turbulence model was correspondingly nominated for simulation of each case, in agreement with previous studies regarding the analysis of flow surrounding three dimensional airfoils [43, 49, 51]. However, in direct consequence to the sizeable reduction of Reynolds number in association with the final flow situation, as well as the requirement for increased boundary layer control over the geometry of examination in the use of any k- ϵ turbulence model, the investigation was permitted to run through LES. The use of this model has been further endorsed by the realisation of similar fluid flow characteristics to previous publications, as is discussed in exceeding detail through section 6.3.

5.5.4.3 Boundary Conditions

The boundary conditions constrained to both hydroplane simulations have been administered due to their absence of interference within the fluid domain, particularly when considering the fluid flow characteristics induced by the hydroplane body. As such, translationally periodic boundary conditions have been further applied here, permitting increased possibility for comparison between the flat plate and hydroplane test cases, while simultaneously emulating greater similarity to realistic application. In agreement with this condition, the outer faces of the fluid domain have each been simulated in such a way, with the faces of the hydroplane geometry selection designated by standard no slip stationary wall condition. The use of translationally periodic boundary conditions results in a requirement for mass flow rate specification, which may be of issue if considering the simulation of fluid flow at fluctuating

angle of attack. However, as this study focuses on the nature of the fluid flow characteristics developed by both smooth and riblet applied hydroplane bodies when subjected to flow at prescribed zero angle of attack, this should not arise as issue here, with the specified mass flow rate again formulated in relation to both the geometry of the fluid domain and the Reynolds characteristic in application. The resultant boundary specifications can be observed via the below figure, with the configuration utilised for the smooth surfaced hydroplane test case provided as a representative form [Figure 29].

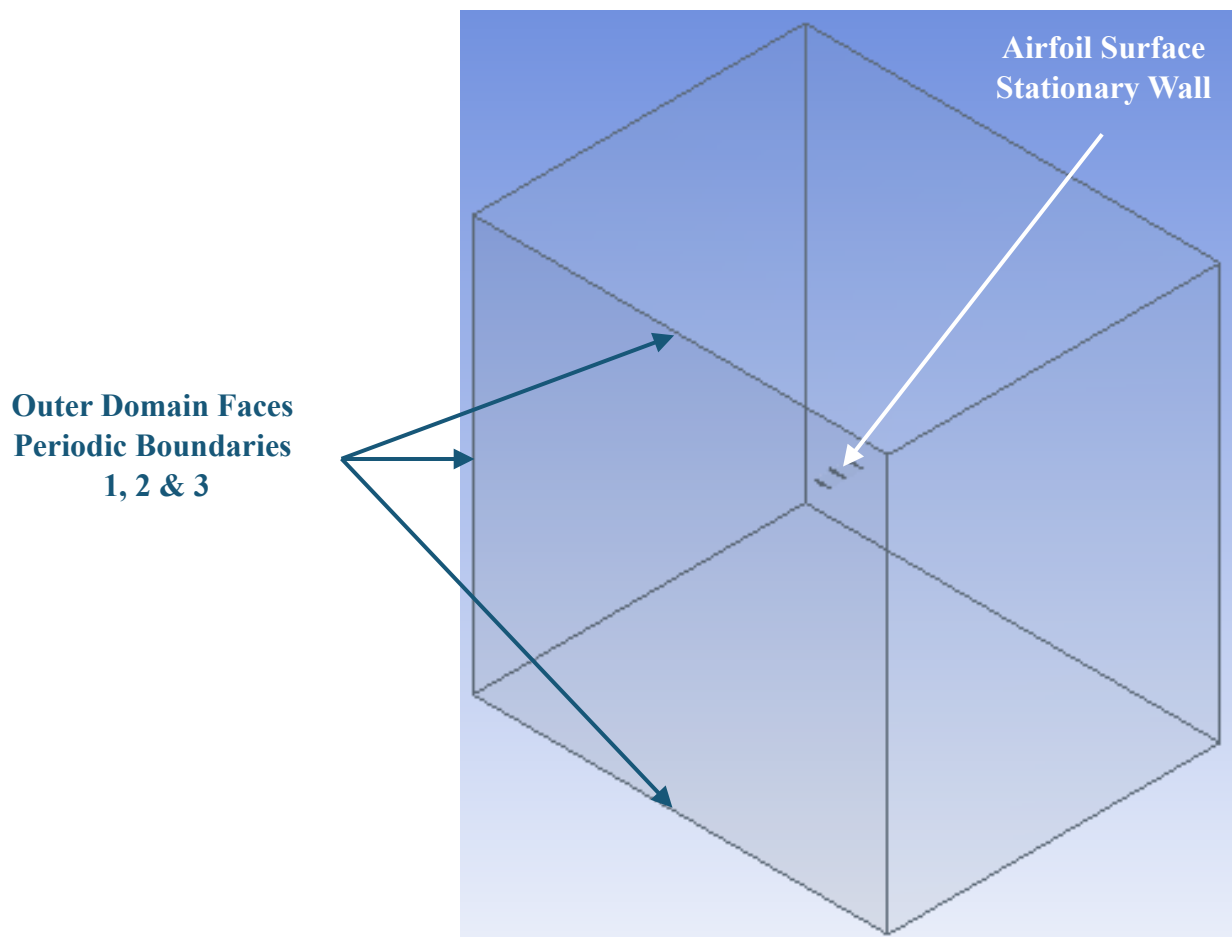


Figure 29, Representation of Applied Boundary Conditions for Each Hydroplane Test Case

5.5.4.4 Solution Methods & Applicable Monitors

The solution methods employed here have been configured as to represent those of the flat plate test cases expressed through section 5.4.4.4, with the below table reiterating the concluding setup [Table 18]. In view of capturing the induced lift and drag characteristics of the hydroplane models in relation to the advancing fluid flow, lift and drag monitors were created as to allow evidential comparison between both cases in accordance with each time step in the flow simulation. A further surface monitor was also issued for each model, attendant to the integral wall shear stress present throughout the proceeding computational solution. Ultimately the simulations were run in accordance with all previous cases, by which, an absolute residual criterion of 10^{-6} was administered, with initialisation of the solution conducted by means of zero conditions.

Fluent Solver	Transient Pressure-Based, Absolute Velocity Formulation
Solution Models	Viscous-LES-Smaorinsky-Lilly
Solution Methods	SIMPLE Scheme, Least Squares Cells Based Gradient, Standard Pressure, Bounded Central Differencing Momentum, Bounded Second Order Implicit Transient Formulation

Table 18, Configurations of Applied Computational Physics for Each Hydroplane Test Case

RESULTS & DISCUSSION

This section of the thesis aims to consolidate the establishment of all garnered results, in accompaniment to interdependent discussions on the nature of the fluid flow characteristics contained within each test case. Presentation of the main outcomes regarding the validation and verification models therefore initiate the chapter in demonstration of their relations to past publications. Finalisation of the validation and verification process then permits evaluation of the results provided by both shark skin inspired riblet applications, specifically, the aforementioned flat plate and airfoil configurations.

6.1 Depiction of Validation & Verification Test Cases

The outcomes of the validation and verification test cases will be presented within this section, where focus is constrained towards highlighting the similarities and consequent discrepancies of the produced results, in comparison to previous publications.

6.1.1 Two Dimensional Cylinder

The primary concentration of the two dimensional cylinder as a validation and verification test case stems from comparison of its induced drag coefficient when subjected to variations in the surrounding fluid flow. Alterations in the flow contained within the stipulated fluid domain are prescribed by means of the characteristic Reynolds number correlated by the geometrical attributes of the cylinder and the corresponding conditions of the encircling fluid. As such, the drag coefficients generated in result of the detailed two dimensional cylinder configurations of section 5.3.1 have been evaluated in their proximity towards previously

published results, through application of the following figure [Figure 30], whereby, comparison of each is conveyed through the succeeding table [Table 19].

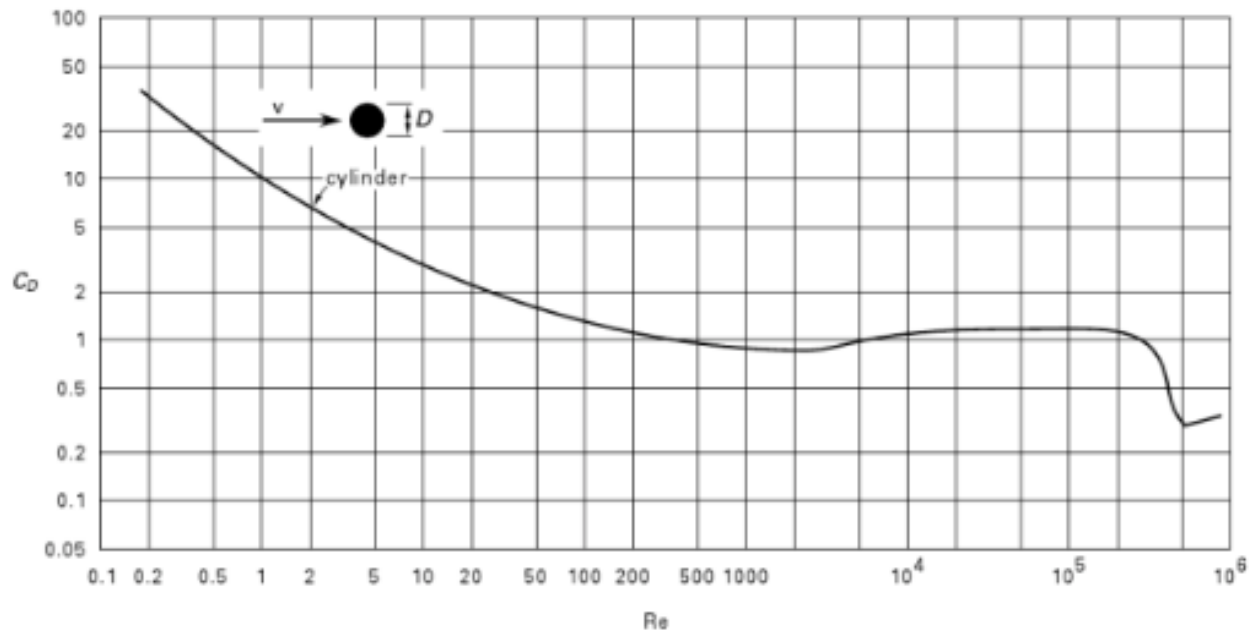


Figure 30, Publicised Drag Coefficient of Smooth Cylinder at Varied Reynolds Number, Adapted From [52]

Reynolds Number, Re	Drag Coefficient	
	Figure 33	Project Test Cases
1	10	12.86
20	2.2	2.03
100	1.3	1.07

Table 19, Comparison of Drag Coefficient-Published to Test Case Results, at Reynolds Number 1, 20 & 100

In assessment of the above juxtaposition it can be deemed that the results of this study are sufficiently well matched by those of previous publications, highlighting the accuracy of their geometrical, mesh and computational configurations while permitting advancement with the main analysis of this study.

6.1.2 Two Dimensional Airfoil

In analysis of any airfoil profile it is quintessential to direct attention towards the lift and drag associations of the particular geometry, under the influence of varying angles of attack to the commensurate fluid flow. In adherence to this, the coefficients of lift and drag have been extrapolated from the fluid flow simulation detailed through section 5.3.2, where comparison of the results attained in response to the induced six degree angle of attack have been outlined in relation to experimental data [Table 20].

Coefficient	Experimental Data	Project Test Case
Lift	0.6630	0.6421
Drag	N/A	0.0123

Table 20, Test Case Lift & Drag Analysis in Regards to Experimental Data, Adapted From [43]

The above table expresses relatively close proximity in the lift coefficient associated with the test case of this study to experimental data, where further refinement of the mesh has previously permitted results of exceedingly small variation. However, reevaluation of this case through increased mesh refinement is not particularly profitable to the main aims of this validation and verification exploration and as such has not been issued within this thesis. In contrast, the results of drag coefficient cannot be compared to actual experimental data, due to the incorporation of an inviscid fluid within this simulation deeming any drag coefficient to be theoretically zero. However, the results here still sanction justification of the applied configurations, with the characteristics of static pressure and velocity magnitude relaying a similar pattern to those achieved by more in depth analysis [43], [Figure 31].

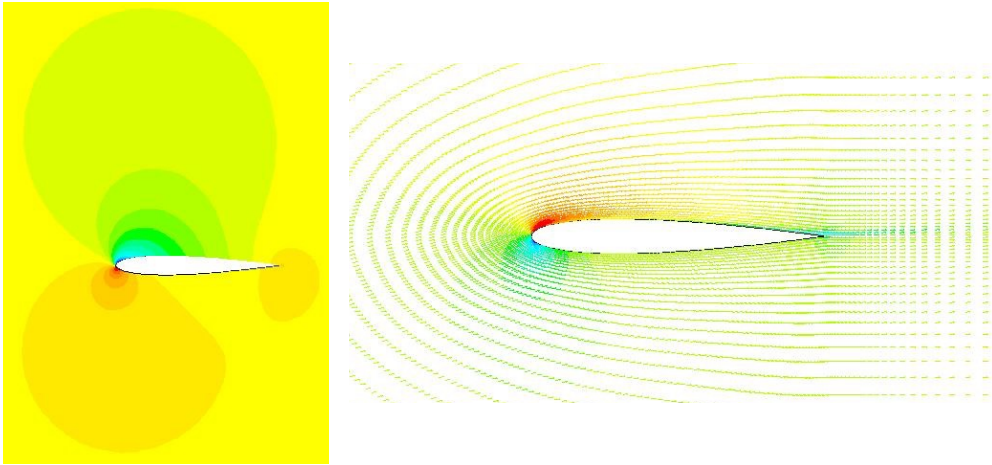


Figure 31, Contour Plots of Static Pressure & Velocity Magnitude Inherent to Test Case Simulation

6.1.3 Driven Cavity Simulation

In evaluation of the results expressed by the lid driven cavity simulation detailed by section 5.3.3, comparison of the instigated velocity of the contained fluid is presented through matching of its horizontal and vertical constituents, in relation to the corresponding geometrical cavity centres, to studies by Bruneau [53], [Figure 32, 33].

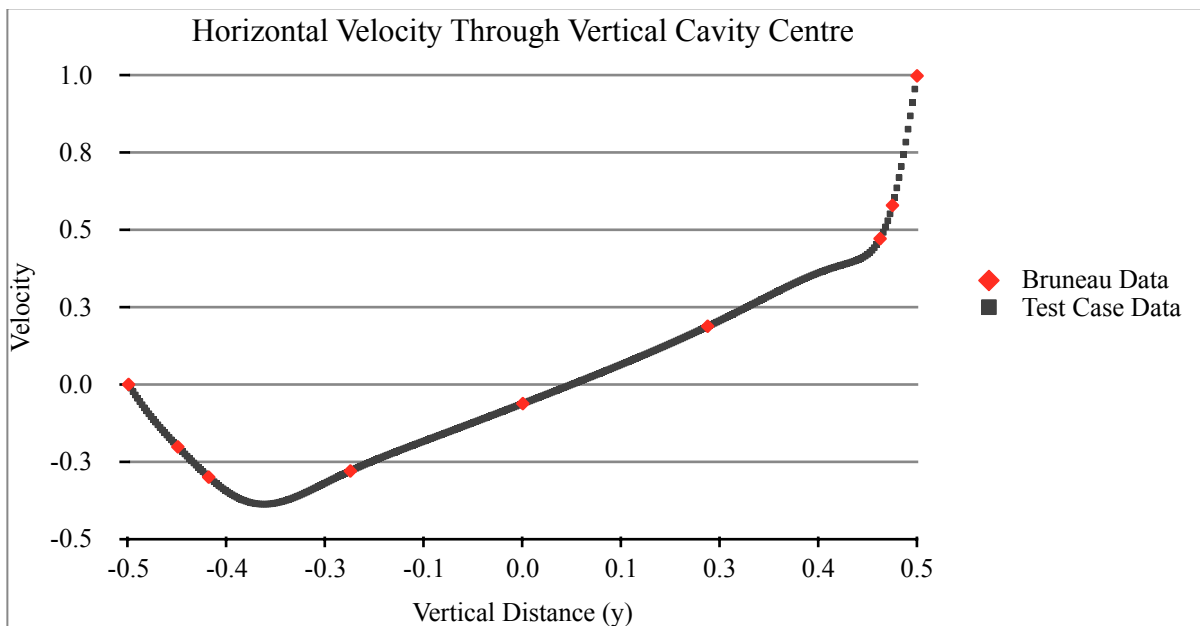


Figure 32, Comparison of Horizontal Velocity Through Vertical Cavity Centre

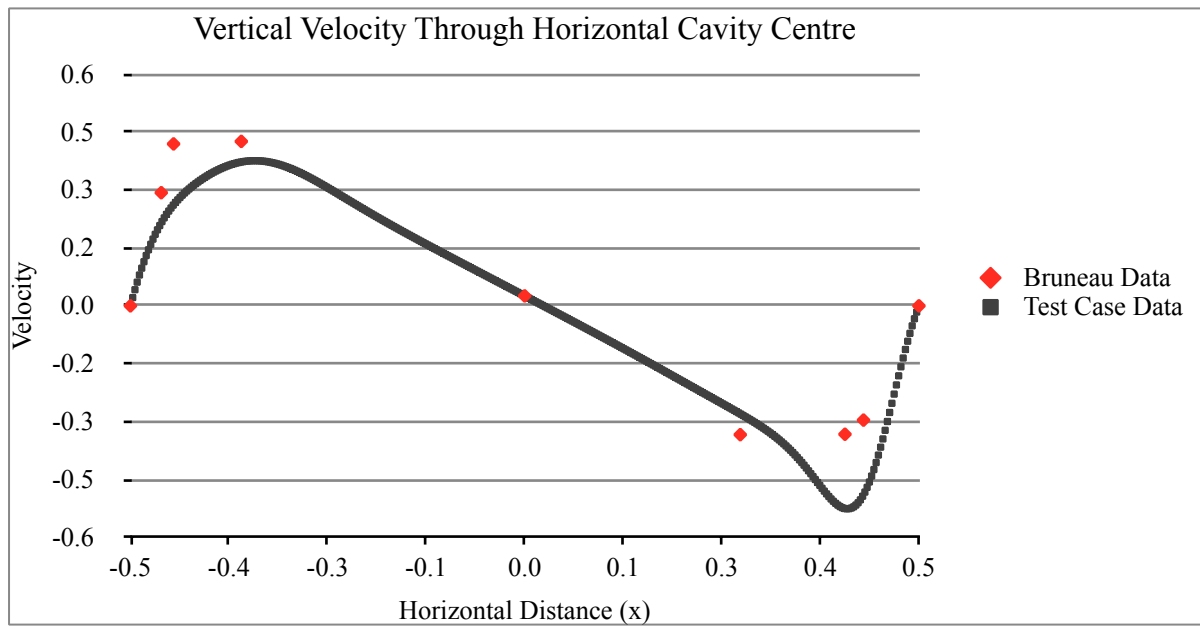


Figure 33, Comparison of Vertical Velocity Through Horizontal Cavity Centre

Examination of the above figures exhibits the accuracy in the development of a horizontal fluid velocity upon crossing the vertical cavity centre, where comparison to published data by Bruneau is indisputably similar. In contrast, although the data presented for vertical fluid velocity, upon intersection with the horizontal cavity centre, follows a similar path to that of Bruneau, variations in the exact magnitudes of velocity along the plane become apparent. The inconsistency presented here can be deemed to stem from varying degrees of mesh refinement, where increased grid processing of this studies test case could allow for results of more significant resemblance to Bruneau's data. Further support towards the approval of the garnered results of this case is expressed through comparison of the contour plots of both static pressure and velocity stream function. The resultant comparisons can be found through the succeeding figures where both plots can be seen to convey similar effects in the developed fluid flow characteristics [Figure 34, 35]. Much of the variation present between the plots is restrained to particular areas of the model,

again dependant on the increased grid refinement presented by Bruneau's simulations, as such, permitting justification of the configurations applied here.

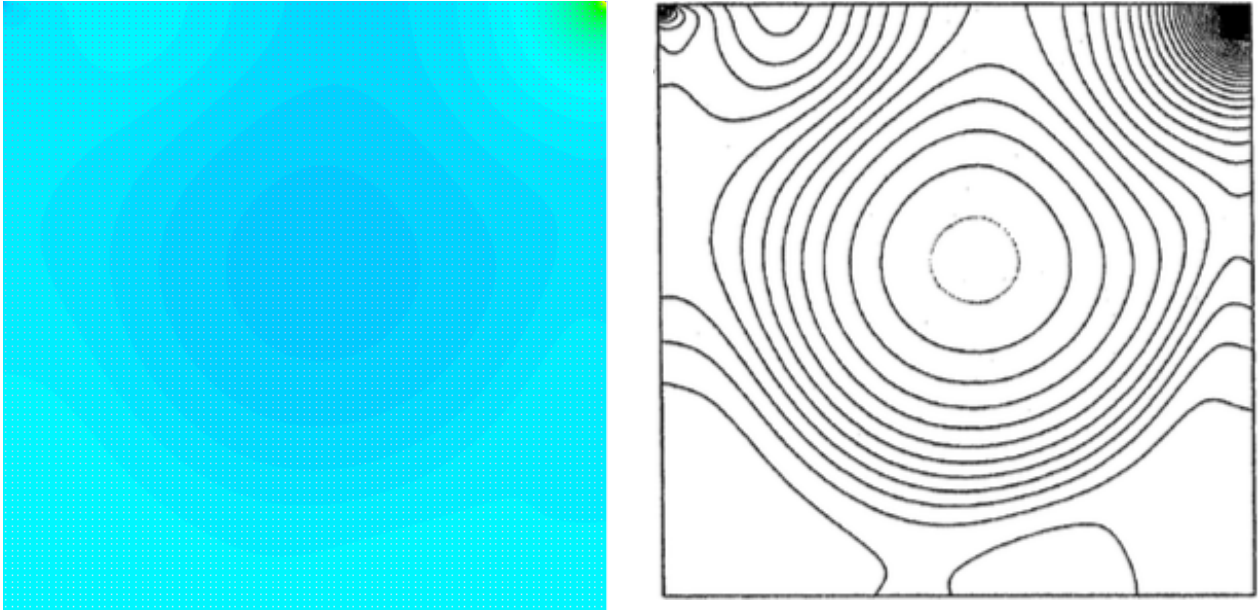


Figure 34, Comparison of Static Pressure Within Cavity, Test Case Contour Plot Left, Bruneau Right

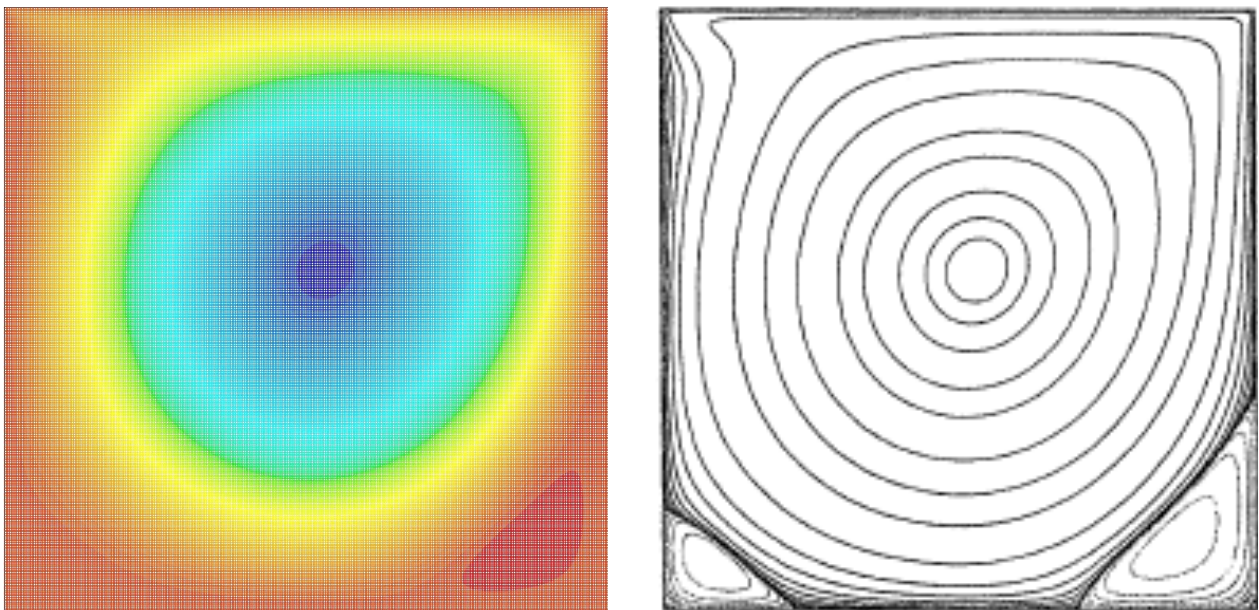


Figure 35, Comparison of Stream Function Within Cavity, Test Case Contour Plot Left, Bruneau Right

6.1.4 Backward Facing Step Simulations

The variations in setup applied to each backward facing step simulation detailed through section 5.3.4, result in a requirement for disparate evaluation between the laminar and turbulent cases. As such, the laminar case is discussed in regard to the horizontal and vertical velocity characteristics present at two intersections along the streamwise direction of the domain, in addition to determination of localised recirculation zones, whereas, focus is placed solely on the location of reattachment regions of fluid along the streamwise direction of geometry when considering the turbulent simulations. For the laminar simulation, the relationship between the test case velocity characteristics and those contained within previous publications [46] is represented through the below figures [Figure 36, 37], namely, comparison of the horizontal and vertical components of fluid velocity present upon intersection of two planes normal to the flow direction.

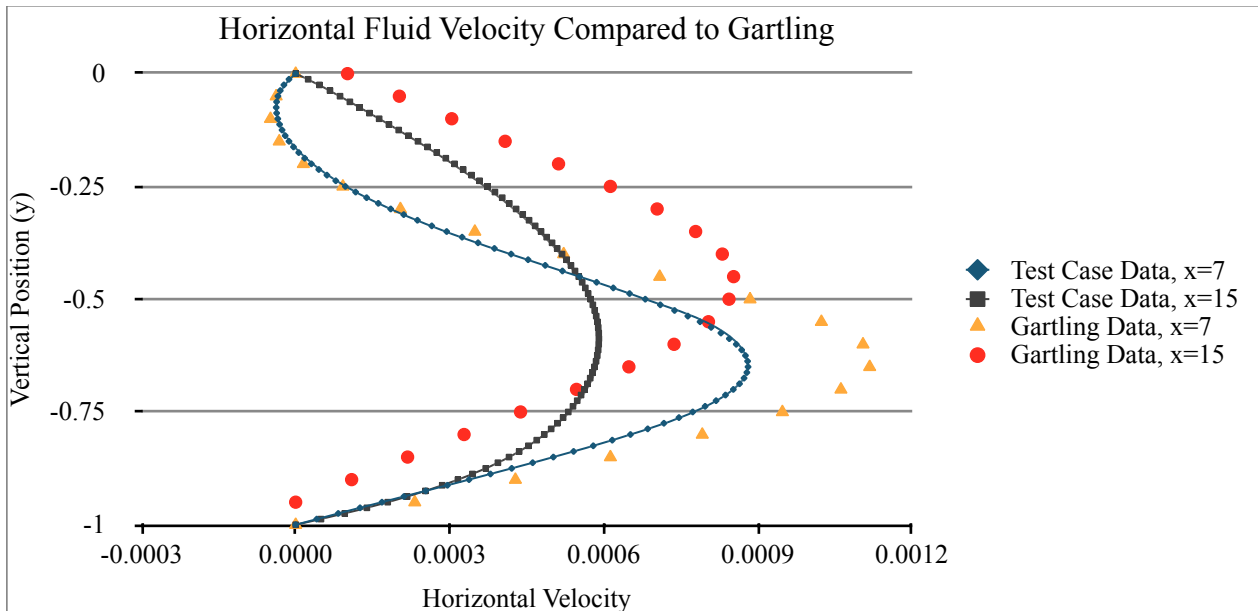


Figure 36, Comparison of Test Case Horizontal Fluid Velocity at x=7 and x=15 to Gartling [46]

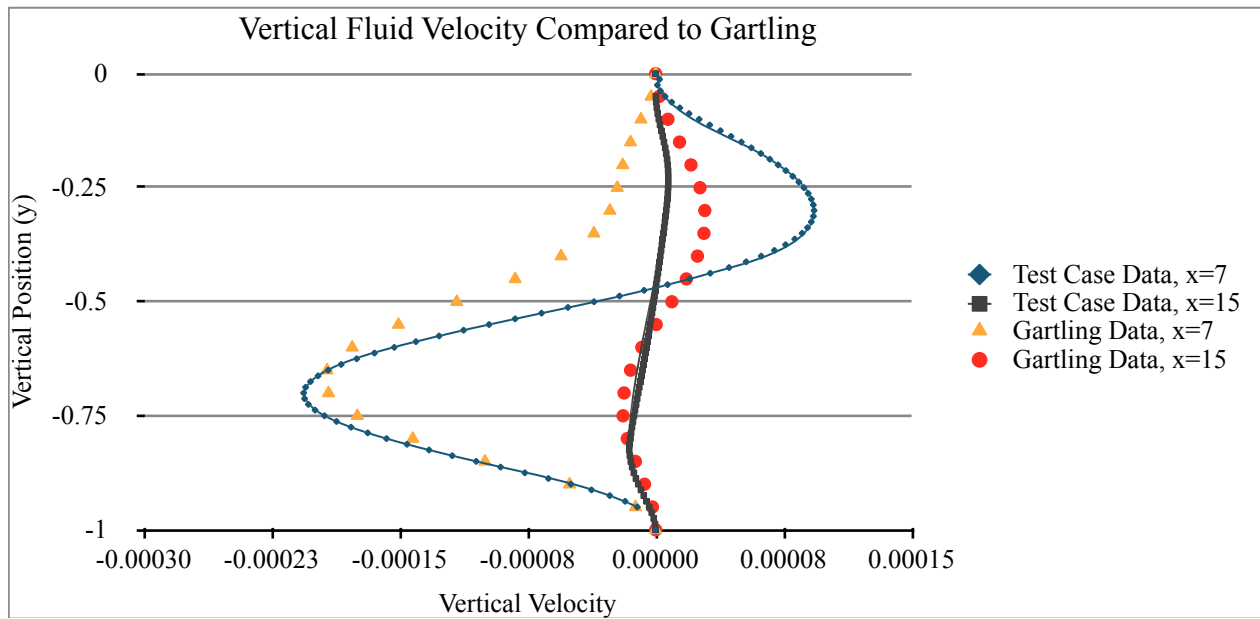


Figure 37, Comparison of Test Case Vertical Fluid Velocity at $x=7$ and $x=15$ to Gartling [46]

The above juxtaposition of figures conveys the discrepancies present throughout the velocity formulations, where although the velocity plots of the test case follow similar paths to those of the Gartling publication, the results are found to vary in the magnitude of each specific velocity component. The most significant variation conveyed by the above figure is that of the vertical fluid velocity present upon intersection with the plane located at seven non dimensional divisions along the streamwise direction of the geometry, where it can be seen to differ widely from the Gartling data, particularly towards the upper region of flow. The variations present here again relate to differences in the mesh configuration applied to each case, with further dissimilarity originating from the time step of application and the overall runtime of each simulation. As such, increased refinement of the applied time step in conjunction with further increases in both the number of time steps and the number of iterations, would permit closer relations to the published works of Gartling. However, this has not been carried out here, as although the data may not provide an exact replica of previous material, the fundamental characteristics of the fluid flow

velocity components have nonetheless been developed by this simulation, with further reiterations of the case not pertinent to this particular study. In addition to the above evaluations of fluid flow velocity, the locations of recirculation zones along the streamwise direction of the fluid domain have been evaluated through use of the below figures, with the zones generated within the test case shown to relate closely to those of past studies [Figure 38, 39]. It must be noted that the vertical divisions present within the below figure each represent one unit of non dimensional spacing, as perceived by $x/H = 1$. This then permits comparison between the locations of recirculation zones within the test case to those of previous publications [54].

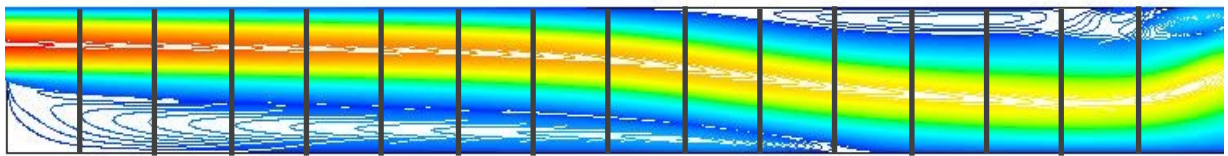


Figure 38, Representative Velocity Magnitude of Laminar Test Case, Containing Non Dimensional Divisions

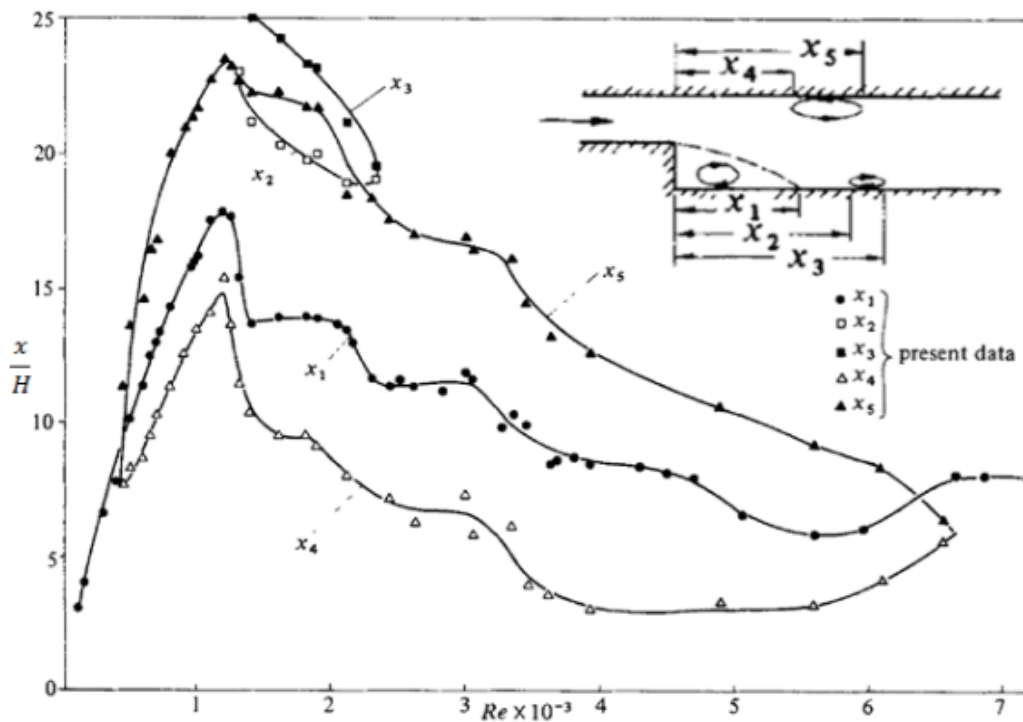


Figure 39, Detachment & Reattachment Locations of Flow Throughout Step Geometry, Adapted From [54]

As previous mentioned, the turbulent cases of backward facing step simulation have been evaluated through comparison of the reattachment locations of fluid in relation to the step geometry. In doing so, the below plots of velocity magnitude convey the location of reattachment present within each test case, where the representative vertical divisions again illustrate one unit of non dimensional spacing ($x/H = 1$) [Figure 40]. The succeeding table then provides evidential similarities between the reattachment locations generated within this study and those outlined by previous research [55], again highlighting the correctiveness of the configurations applied [Table 21].

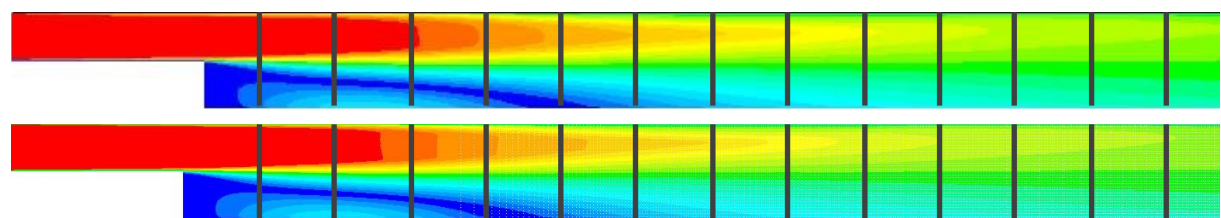


Figure 40, Examination of Primary Reattachment Region of Turbulent Test Cases, With Top Representative of $Re = 93,000$ and Bottom of $Re = 210,000$

Reynolds Characteristic	Location of First Point of Reattachment (x_1)	
	Published Data	Project Test Case
93,000	7	8.1
210,000	7	7.2

Table 21, Comparison of Reattachment Location - Published to Test Case Results

6.2 Exploration of Riblet Applied Flat Plate Test Cases

This section of the thesis aims to present the main findings evident to the shark skin inspired, riblet applied, flat plate test cases, introduced through section 5.4. Contemplation of the influences of each riblet configuration, in relation to the characteristic integral wall shear stress' present over said model and its smooth counterpart, constitute the main outcomes of this study. Whereby, examination of the attributes of the resultant fluid flow contained within each test case permits evaluation of the aforementioned attributes of wall shear stress, in the provision of justification towards the variant nature of the fluid flow characteristics developed in respect to the individual riblet geometry.

6.2.1 Flow Characteristics of Each Model

The characteristics of the fluid flow induced by the three model configurations are outlined within this subsection, with respect to the development of velocity magnitude and tangential velocity within each case. The formation of fluid flow velocity within each test domain was prescribed by the stipulation of a mass flow rate, in conjunction with a predetermined Reynolds number of 4200, as discussed through section 5.4.2. The resultant velocity characteristics of the simulated fluid flow are exhibited by the following figure, whereby, the contour plots of velocity magnitude developed by Model 2 have been issued in representation, due to evident similarities between all three cases [Figure 41]. The below figure showcases the variances in velocity magnitude present throughout the fluid domain, in relation to the upper and lower walls. In doing so, the velocity of the fluid passing through the riblet valleys can be seen to obey a lower magnitude than that at the riblet tips. Not only does this adhere to the outcomes of the literature review section of this thesis [Section 4.3], it also permits the accumulative velocity of flow over the riblet structured surface to be lower than that immediate to the smooth surface on

top of the fluid domain. The resultant influences of this diversification in velocity magnitude on the attributable integral wall shears stress' present over each domain wall are discussed within the succeeding subsection of this review [Section 6.2.2], permitting justification of the coinciding variations in skin friction drag to be expressed.

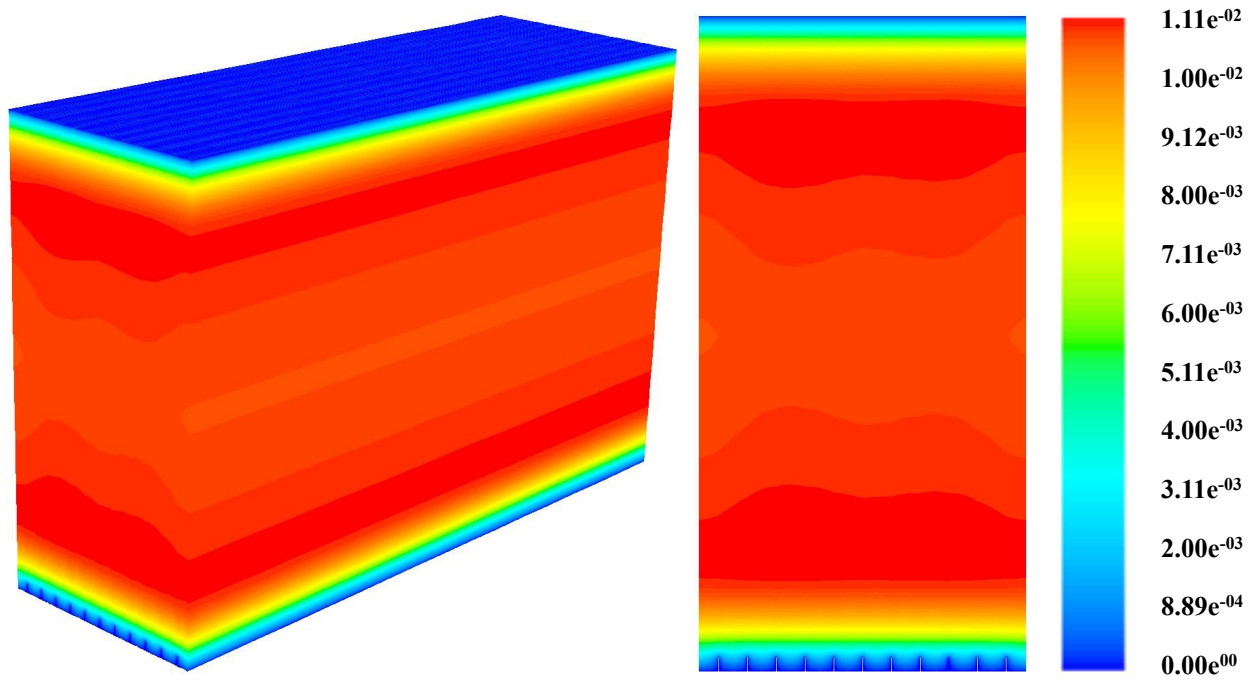
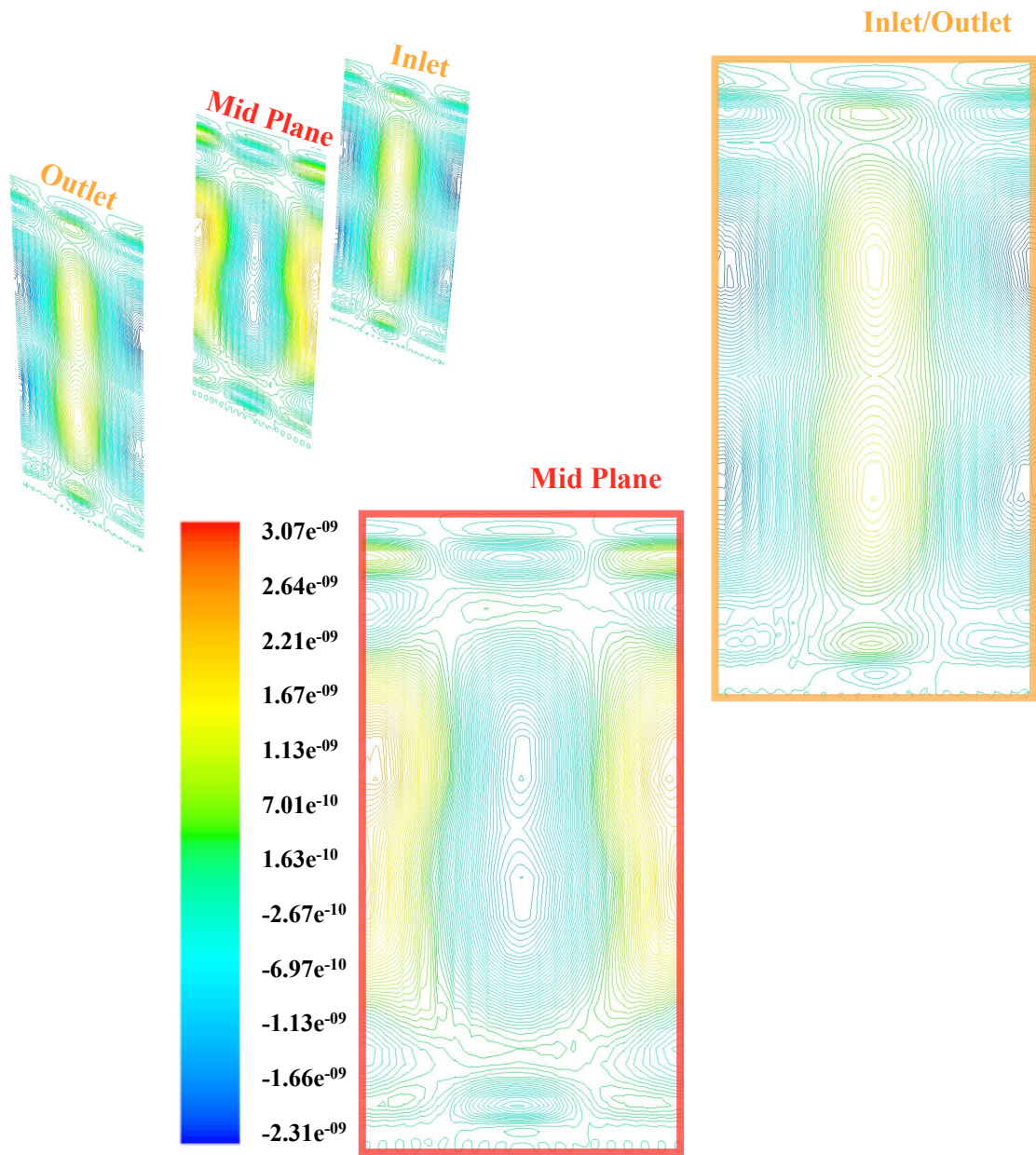
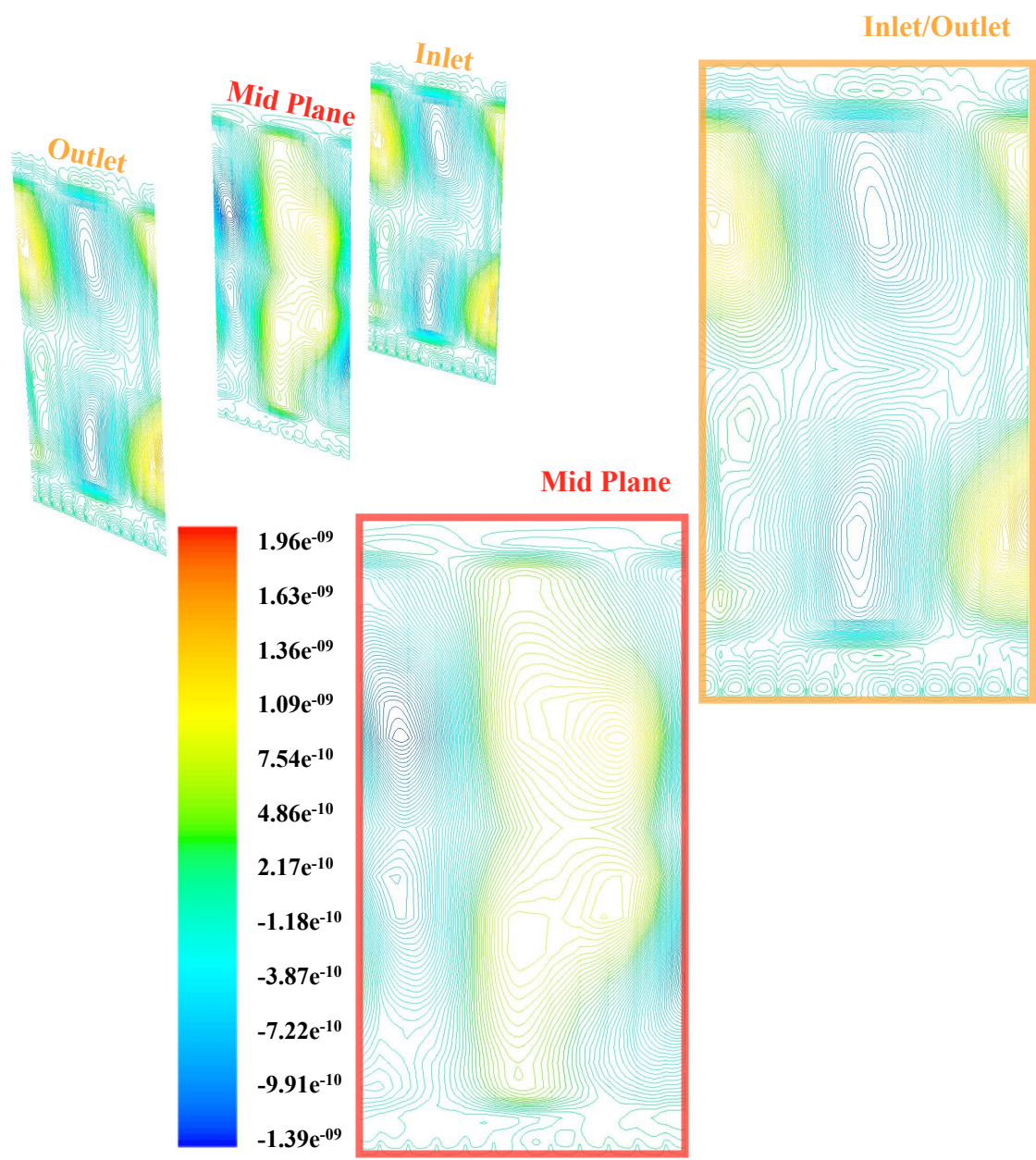


Figure 41, Contour Plot of Velocity Magnitude Inherent to Model 2 Simulation

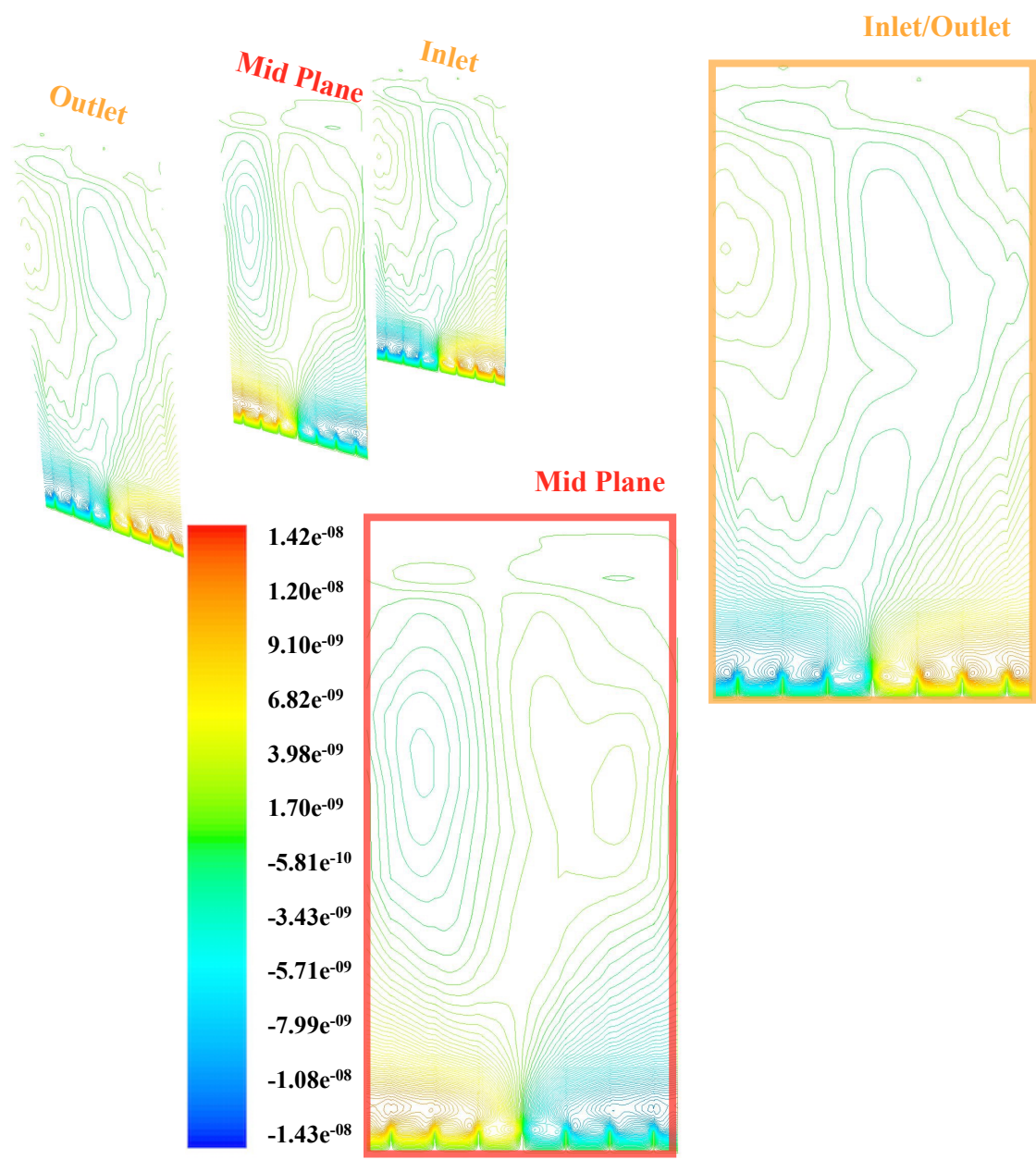
In the provision of subsidiary analysis towards the effects of varied riblet geometries on the advancing fluid flow throughout each model domain, the formulation of secondary vortices along the spanwise direction of the each domain have been evaluated. In view of showcasing these vortices, the contours of tangential velocity have been plotted at three distinct intersections to the streamwise flow, specifically, upon entering the domain, midway through the domain and upon exiting the domain [Figure 42, 43, 44]. These plots have been issued as to highlight the interaction of each individual riblet geometry with the encompassing fluid flow, while simultaneously conveying the variations in secondary vortex formulation throughout the streamwise domain.



**Figure 42, Contour Plot of Tangential Velocity Inherent to Model 1 Simulation,
Representative of Secondary Vortex Development**



**Figure 43, Contour Plot of Tangential Velocity Inherent to Model 2 Simulation,
Representative of Secondary Vortex Development**



**Figure 44, Contour Plot of Tangential Velocity Inherent to Model 3 Simulation,
Representative of Secondary Vortex Development**

The preceding figures convey distinct variations in the formulation of vortices throughout the streamwise flow, where the contours of tangential velocity can be seen to differ considerably between those present upon crossing the inlet/outlet periodic boundaries and those intersecting the mid plane of each domain. This fluctuation in vortex formulation stems from the induced development of turbulence within each case, where although the simulations are permitted to run with identical Reynolds number, the interaction of each riblet geometry within the surrounding fluid introduces distinguished features of the resultant flow. From **[Figure 42]** the behaviour of flow in the region surrounding the riblet structured surface can be seen to transmit similar patterns to that of its smooth counterpart, due to the specific geometry of the riblet structure enforcing little effect on the elevation of vortices from the riblet valleys. As such, although the vortices generated do not rest within the valleys themselves, the increased contact with multiple riblet blades enforces a higher resultant force on the entire riblet wall. In contrast, the flow patterns induced by the simulation of Model 2, conveyed by **[Figure 43]**, exhibit evidential deviations in the regions surrounding both walls of the domain. It can be concluded that although there is visible introduction of turbulent characteristics within the riblet channels, the elevation of larger vortices from the riblet structure, towards the outfield flow, permits the reduced effect of turbulence in accumulation over the riblet wall of this model. The tangential velocity components associated with the simulation of Model 3, as relayed by **[Figure 44]**, showcase the varying nature of the fluid flow direction, with the flow seen to vary in both its streamwise and spanwise direction throughout the domain. Further to this, the increased lateral spacing, present between each riblet blade, permits the development of turbulent vortices on entrance to the riblet valleys within this model, stemming from a lack of interaction between the turbulent flow and multiple blade tips, for extrusion of turbulent characteristics from the riblet wall. As such, summation of these figures, in coalition with the findings of the literature review conducted in start this study,

puts forth the dispute of balancing the interaction of multiple riblets with any one vortex (in aim of lifting the vortex from the riblet valleys) and reducing the resultant force inherent to multiple contacts. Furthermore, due to the latitudinal effects of each riblet structure on the corresponding flow within their encompassing fluid domains, the accuracy of results attained in relation to the smooth surface of the top wall is brought into question. This uncertainty in regard to the reliability of results is based around the perceived flow interaction concerning both walls in the fluid domain, as although each domain geometry has been configured in accordance with previous publications [47], the above figures express some interaction in the fluid characteristics instigated by each wall. As such, further investigation into the fluid flow characteristics developed by each wall may be better represented by simulation of each individual case — separating the smooth and riblet walls into independent simulations. However, due to the increased time and computational demands required for such an exhaustive evaluation, the results presented here have been deemed to be of suitable accuracy for the present study.

6.2.2 *Influence of Riblet Configuration on Drag Reduction*

Upon exploration of the surface monitor reports, detailing the integral wall shear stress development per time step of the conducted simulations, the derived friction drag influences of each model configuration, depicted by the below table, were developed [Table 22].

Model Reference	Lateral Spacing, s^+	Influence on Friction Drag
Model 1	$s^+ = 15$	-2.5%
Model 2	$s^+ = 25$	-10.9%
Model 3	$s^+ = 40$	+10.3%

Table 22, Examination of Individual Riblet Configuration on Friction Drag Influence

In evaluation of these results it must be noted that negative drag influence is representative of drag reduction, while positive drag influence relays an increase in the developed drag over each riblet structured surface, when evaluated in comparison to its smooth counterpart. As such, it is apparent that although both Models 1 and 2 develop a drag reduction, Model 1 comprises considerable advantage. In contrast, the representative geometry of Model 3 leads to a substantial increase in the developed drag force, of similar magnitude to the reduction provided by Model 2. This variation in drag influence conforms to the main findings of the related literature review section of this thesis [Section 4.4], by which, optimisation of the specific geometrical attributes of a particular riblet case results in a conquering geometry, in terms of its advantageous fluid flow effect. The results attained here can be evaluated with respect to the previous section, where the consequences of an increase in wall surface area have to be outweighed by the abilities of the particular structure to enhance the turbulent properties of the surrounding fluid flow. Therefore, the perceived reduction in fluid velocity within the riblet valleys does not solely result in a reduced drag force, as is evident by the results of Model 3, where the substantial increase in surface area of the riblet wall in comparison to the smooth upper wall, culminates in an increased drag force. In addition, the increased lateral spacing attributable to the riblet wall of this model, prohibits the interaction of multiple riblet tips with turbulent vortices, as has been previously depicted, emphasising the effect of turbulent alleviation on the resultant surface friction drag. In view of showcasing the variant nature of shear stress present over the walls of each fluid domain, the following contour plot has been issued [Figure 45]. Although the below plot publicises the specific case of Model 2, it is nonetheless representative of each simulation, due to the inherent similarities of the three test cases. It can therefore be observed that although a higher shear stress is present in relation to the riblet tips, there remains substantially greater region of reduced stress within the riblet valleys when compared to that of the smooth upper surface. As such, it can be

seen that the aforementioned balance of characteristics must permit the accumulative wall shear stress of the riblet valleys to outweigh the increased magnitude applied to each individual riblet tip, as is the case for both Models 1 and 2.

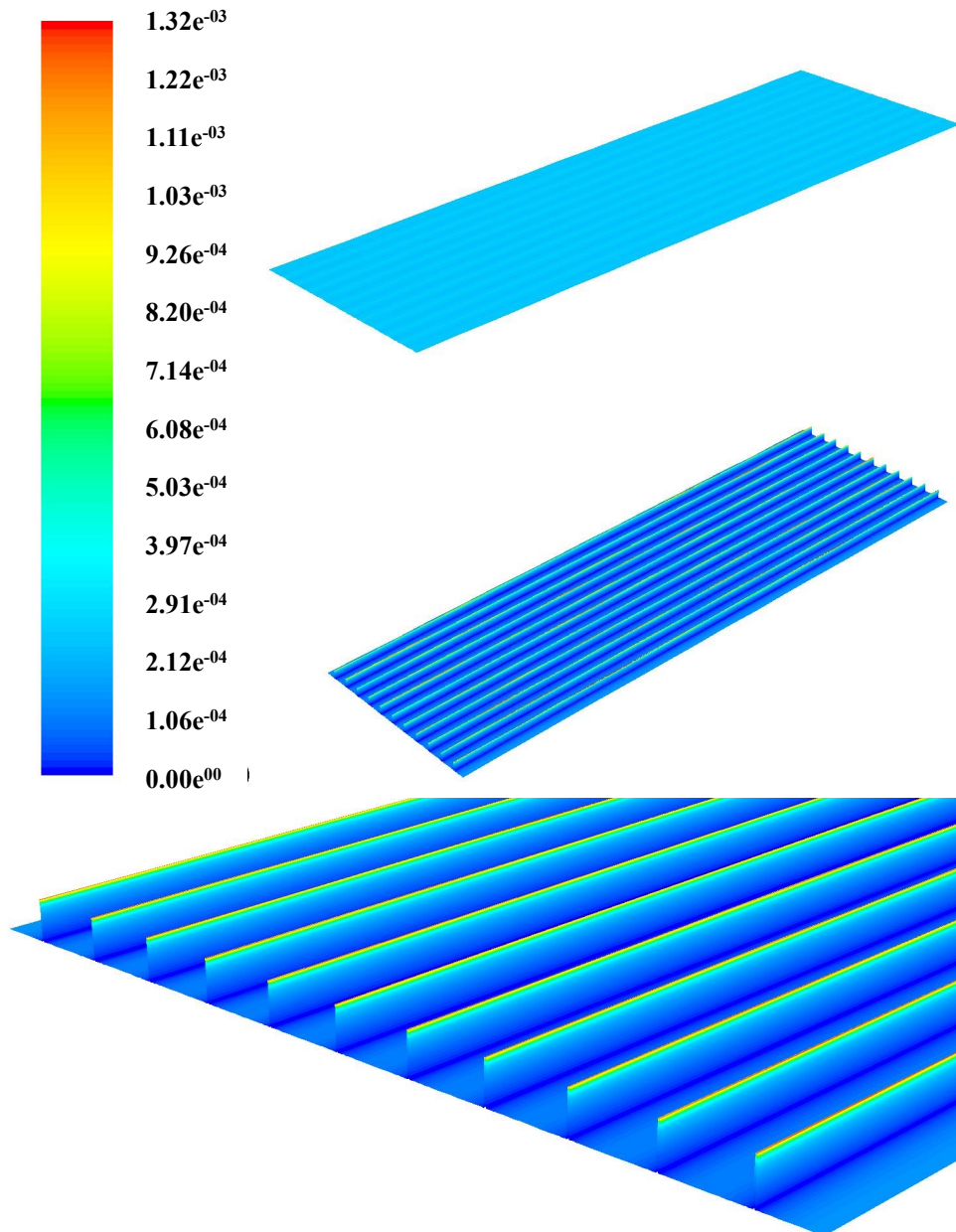


Figure 45, Integral Wall Shear Stress Contour Plot Comparison Between Smooth and Riblet Structured Surfaces Inherent to Model 2 Simulation

The conclusions of the above figure are further endorsed through the ensuing plots of existent integral wall shear stress along the spanwise direction of each fluid domain [Figure 46, 47, 48]. These plots assist in sanctioning the drag influences of each riblet structure, where the characteristics of the above figure are further reiterated in presentation of Model 2 as a prominent drag reducing geometry. This stems from the permission of the models geometrical attributes on a resultantly reduced fluid velocity within the riblet valleys, in manifestation of a subsequent reduction in wall shear stress within these areas, as such, prevailing over the higher wall shear stress of the riblet tips in delivering an overall reduction when compared to results of its smooth counterpart.

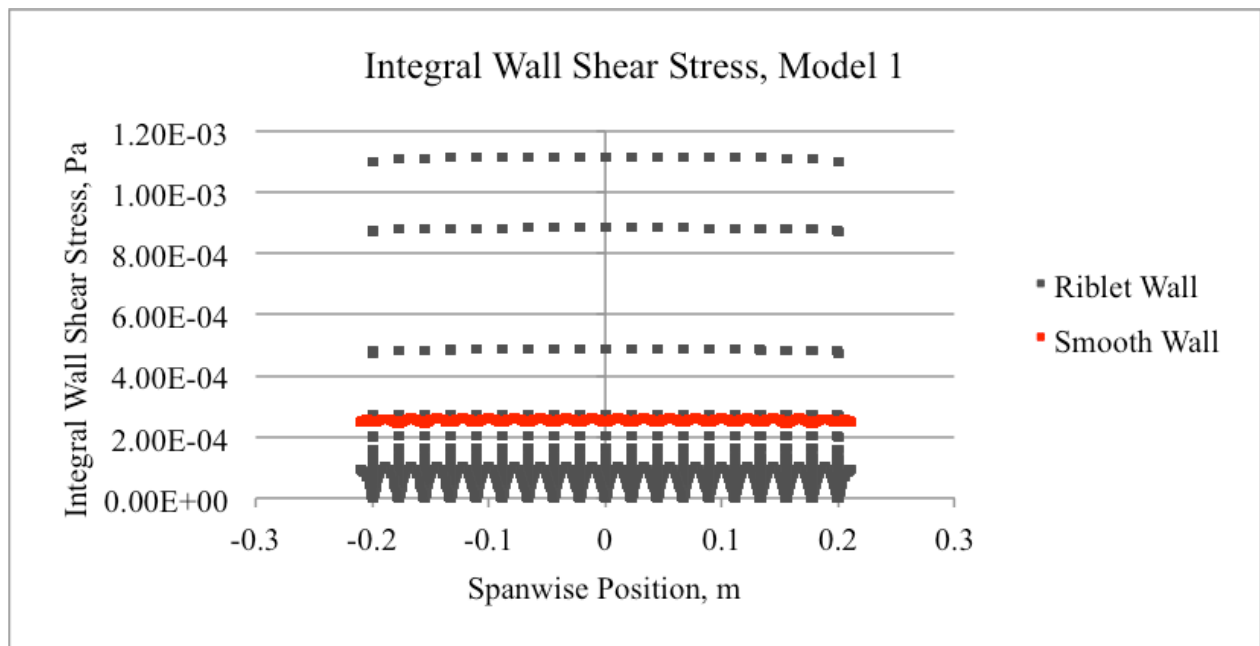


Figure 46, Integral Wall Shear Stress Development in Relation to Spanwise Location of Fluid Domain, Inherent to Model 1 Simulation

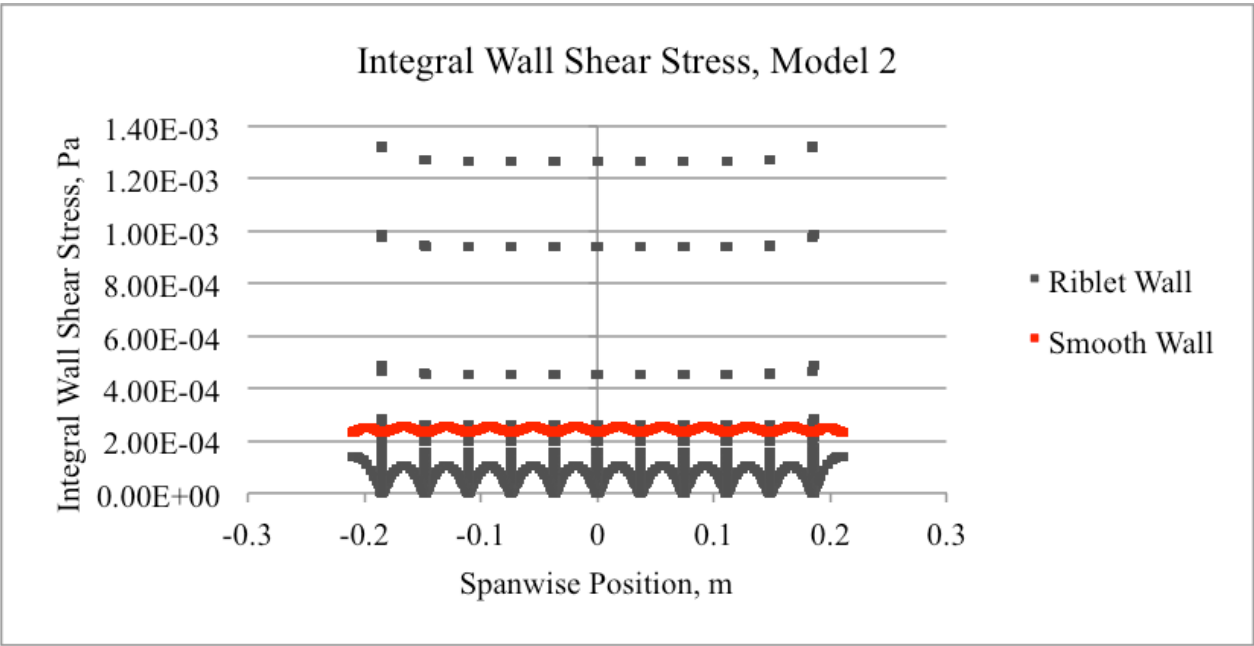


Figure 47, Integral Wall Shear Stress Development in Relation to Spanwise Location of Fluid Domain, Inherent to Model 2 Simulation

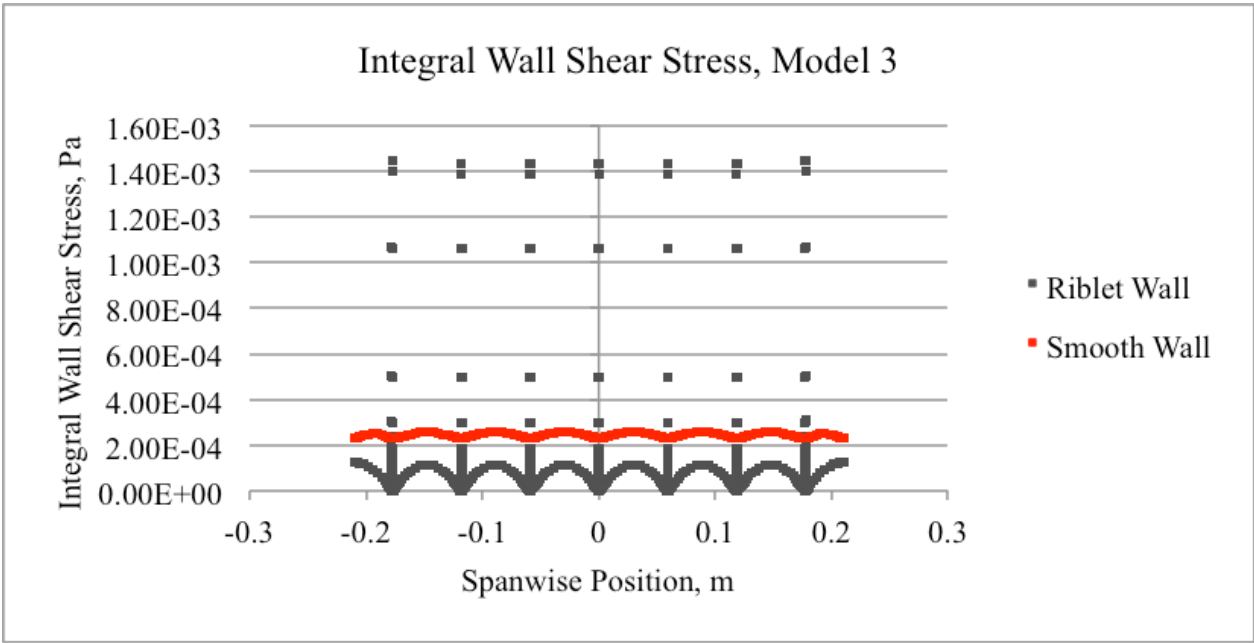


Figure 48, Integral Wall Shear Stress Development in Relation to Spanwise Location of Fluid Domain, Inherent to Model 3 Simulation

6.3 Exploration of Optimised Riblet Application to Hydroplane Simulation

In this final section of project results, the hydroplane simulations detailed by section 5.5 are explored through examination of the inherent fluid flow characteristics of each model domain. This then permits differential evaluation between the influences of each model — smooth and riblet applied — on the provision of lift and drag forces, as such, concluding the main objectives of this study.

6.3.1 *Flow Characteristics of Hydroplane Simulations*

In evaluation of any CFD study which focuses on the induced attributes of fluid flow around airfoil profiles, it is typical to provide comparison of the contour plots of velocity magnitude and static pressure inherent to each individual test case. In adherence to this stipulation, variances of these plots have been issued below for both hydroplane simulations of this study — smooth and riblet applied — highlighting the intrinsic similarities and subsequent deviations of each case [Figure 49, 50]. Although the contours contained in each of these figures may initially present themselves in varying degrees of magnitude, when examined in relation to the integrated magnitude scale, it becomes apparent that the plots emanate a similar pattern. This variation in the magnitudes present within each simulation stems from the increased magnitude induced upon coalition with the leading edge of each riblet blade, enforced by the abrupt change in direction of the hydroplane geometry at this position along its streamwise length. In regard to dissimilarity of the resultant fluid flow velocity magnitudes, the contour plots taken upon intersection of the spanwise mid plane, convey variation in the flow formulation beyond the trailing edge of the riblet applied portion of hydroplane body. It can be seen here that the flow follows a turbulent path in this region, adhering to the combination of flow channelled through each riblet valley and the rushing motion experienced by that upon reaching the trailing edge of

the riblet blades. This characteristic can be deemed to stem from the scaling of Reynolds number to riblet geometry, in appliance within this study as detailed by section 5.5.2, due to the largescale increase of the non-dimensional riblet blade lateral spacing taking effect. As such, if further study could permit the application of a riblet structured surface scaled by a Reynolds number of one million, the simulated effects on the resultant fluid flow would be seen to change drastically from those outlined within this thesis, sanctioning the formulation of characteristics of greater appeal, a factor discussed in greater detail through sections 6.3.2 and 7.2.

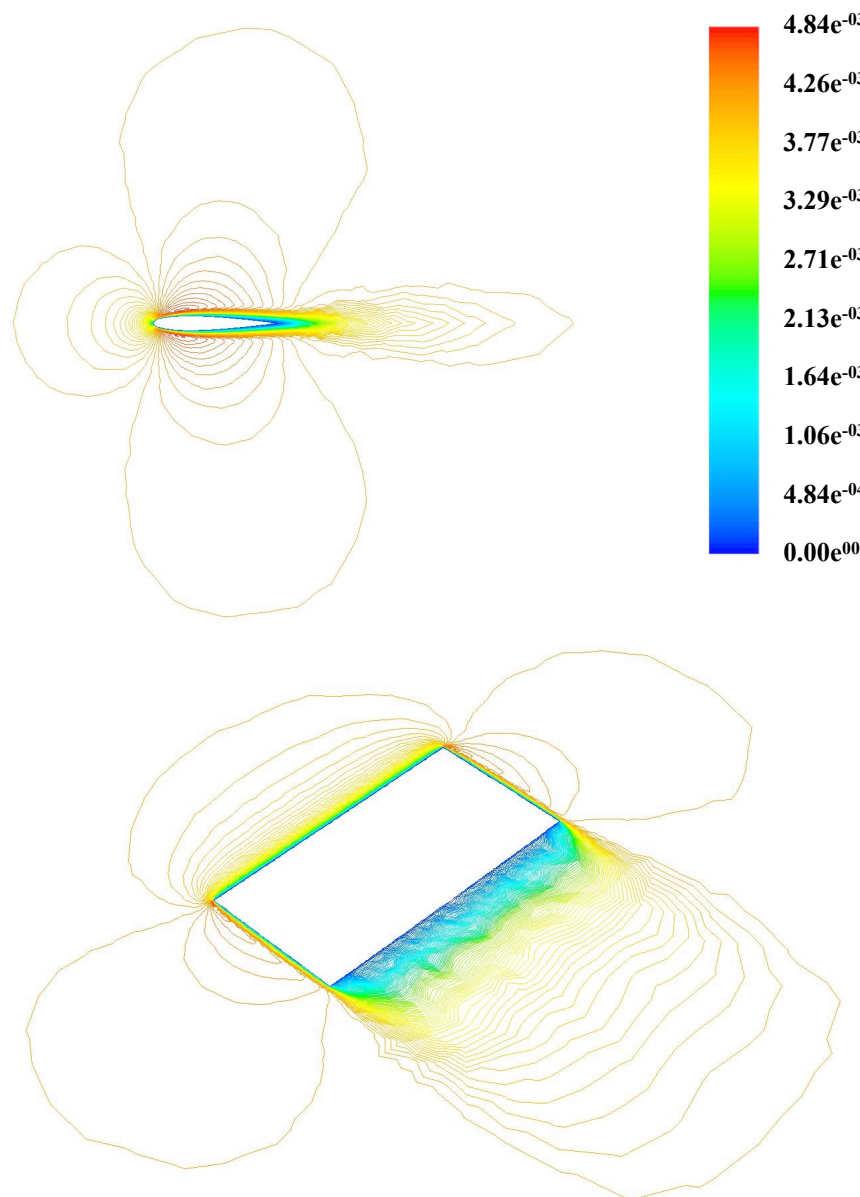


Figure 49, Contour Plots of Velocity Magnitude Upon Intersection with Vertical and Horizontal Centres of Smooth Hydroplane

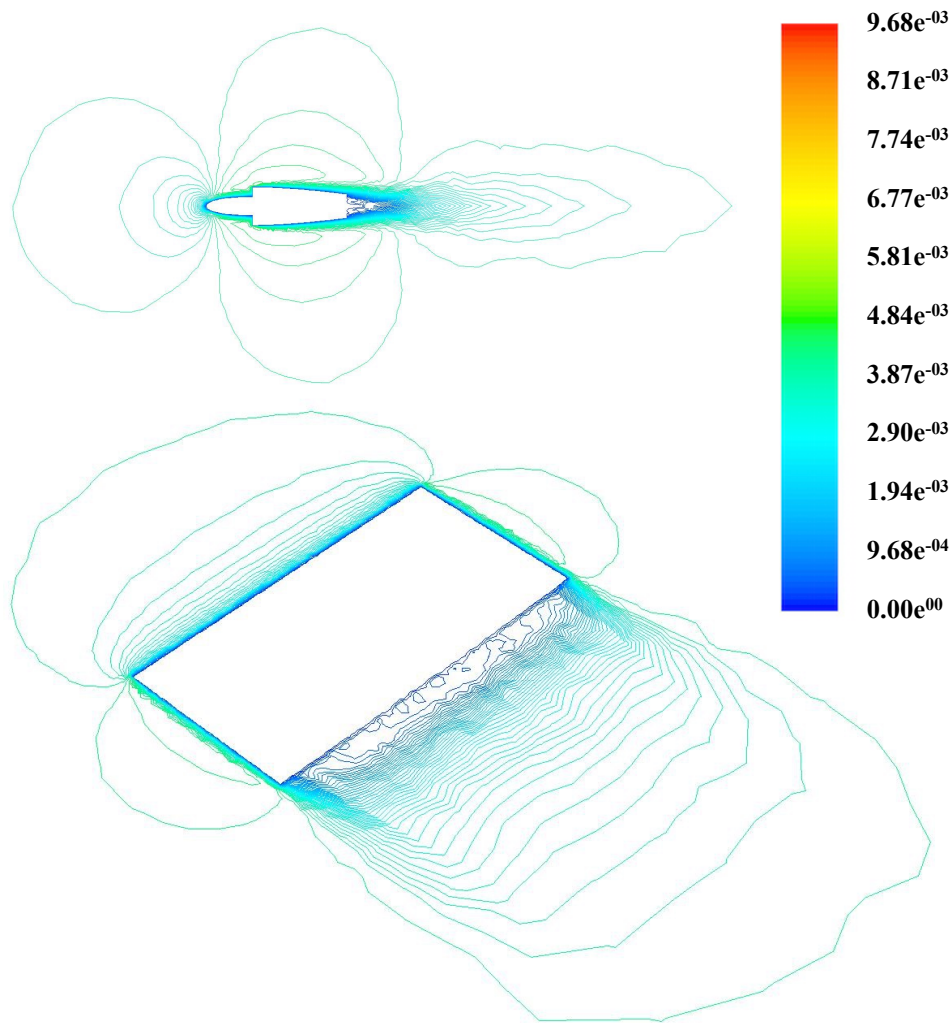


Figure 50, Contour Plots of Velocity Magnitude Upon Intersection with Vertical and Horizontal Centres of Riblet Applied Hydroplane

Although the contour plots of static pressure, in relevance to each simulation, do not provide particular advantage to the outcomes of this study, the inherent plot of the smooth surfaced hydroplane simulation has been provided below, in provision of further justification towards the accuracy and correctiveness of the geometrical, mesh and computational configurations in application [Figure 51]. This plot has been correlated in reference to previous studies, with the resultant static pressure of the fluid domain shown to convey similar development to that of not

only the two dimensional airfoil test case utilised in validation of this study [Section 6.1.2] but also to [43].

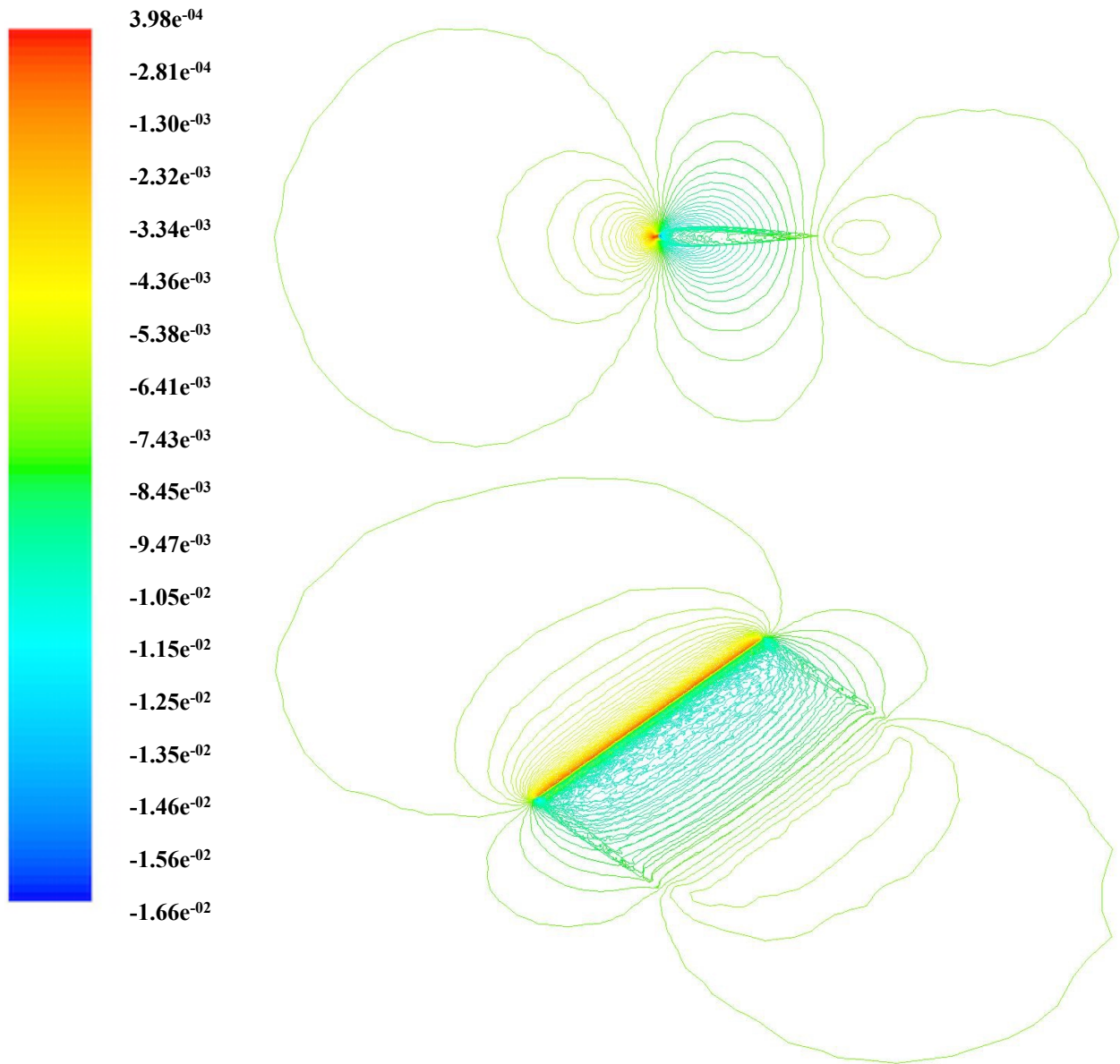


Figure 51, Contour Plots of Static Pressure Upon Intersection with Vertical and Horizontal Centres of Smooth Surfaced Hydroplane

In addition to the above, the succeeding figure expresses comparison of the pressure coefficient inherent to both smooth and riblet applied hydroplane test cases [Figure 52]. By evaluation of this

figure, the proximity of results is clear between both test cases and in regard to [43] with exception of the leading and trailing edges of the riblet blades, where a significant increase in pressure is recorded. The increased magnitude of pressure coefficient native to these two locations along the streamwise hydroplane length, can again be deemed attributable to the notable increase in riblet geometry that was unfortunately obligatory for this study to proceed.

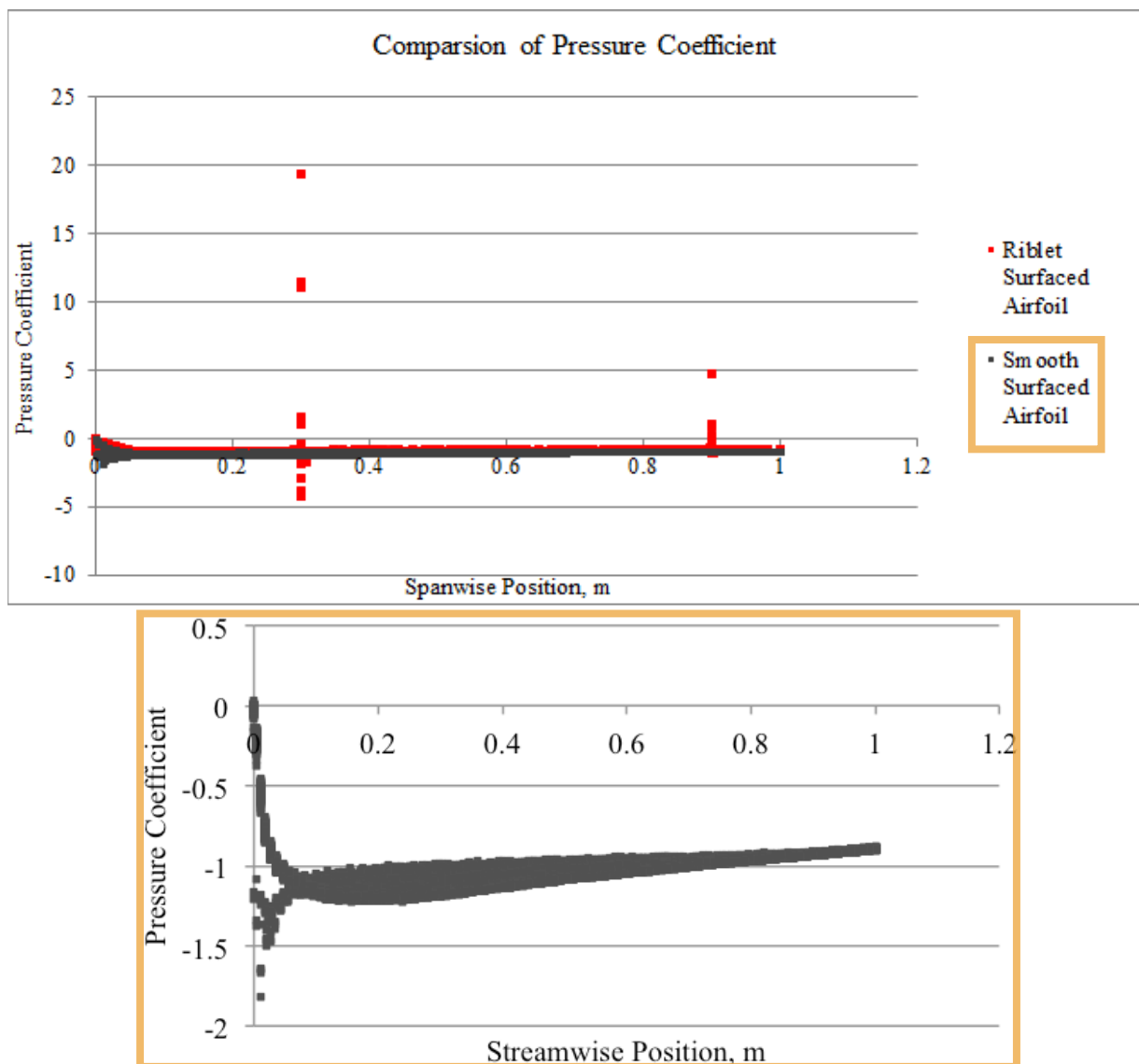


Figure 52, Pressure Coefficient Development in Relation to Streamwise Location of Hydroplane Body, Comparison of Smooth and Riblet Applied, with Further Inspection of Smooth Hydroplane

6.3.2 *Influence of Riblet Application on Drag Reduction*

Upon evaluation of the influential nature of the endorsed riblet application on skin friction resistance reduction, the characteristics of lift and drag coefficient have been discussed in addition to the formation of wall shear stress on each hydroplane body. Unlike the flat plate test cases evaluated through section 6.2, the application of an optimised blade riblet configuration, utilising the $s^+ = 25$ non dimensional characteristic, has not yielded drag reducing effect. The results of the induced lift and drag coefficients inherent to each test case — smooth and riblet applied — have been outlined within the below table, conveying the significant increase in drag force consequential of riblet application [Table 23].

Coefficient	Smooth Surfaced Hydroplane	Riblet Applied Hydroplane
Lift	0.0010937052	0.00072480413
Drag	0.13530812	0.20071335

Table 23, Evaluation of Lift and Drag Coefficients Inherent to Smooth and Riblet Applied Hydroplane Cases

This unfortunate outcome is resultant of a number of issues contained within the riblet structure configuration in application here, as has been previously touched on throughout the proceeding section of results. The significant increase in the final riblet geometry enforced in order to ease both the mesh and computational consumption of the study, in permission of any progression towards attaining simulation results, culminates in solutions of severe dissimilarity to those perceived by externally turbulent characteristic (for example $Re = 1,000,000$). The application of a riblet structure correlated by a Reynolds number of one million would permit the incorporation of said structure at micro scale towards the hydroplane geometry, allowing the riblet blades to introduce similar effects to those outlined by the flat plate test cases of section 6.2. However, in

order for any similarity to occur between the effects induced by the riblet structures of the flat plate test cases and those of the hydroplane application, the advancement of flow throughout each fluid domain must prescribe to consequently similar degrees of turbulent interaction. With this in mind, although the boundary conditions in application to each of these cases have been issued in conjunction with one another, the variation in domain configuration enforced on the hydroplane simulations results in these cases no longer emanating the effects of flow through a channel. A direct ramification of this is that the prescription of a Reynolds number of 4200 towards the hydroplane test cases then simulates a laminar flow regime, in comparison to the turbulent regime inherent to the flat plate test cases at identical Reynolds setup, due to the boundary conditions now permitting exploration of externally flowing fluids (translationally periodic boundaries remove wall effects). Therefore, in summarisation of the above findings it becomes apparent that although the boundary conditions applied to the hydroplane test cases allow for more realistic exploration of the developed fluid flow characteristics (hydroplane use is predominately an external fluid situation — within seas, rivers, lakes etcetera), the stipulation of a Reynolds number lower than one million culminates within the laminar regime, prohibiting the desired effects of any riblet structure simulation on interference with turbulent flow characteristics, for induced drag reduction. The arguments contained above then answer any doubt as to why no drag reduction could be induced by the riblet applied hydroplane test case, as the necessary stipulations enforced for any progression towards CFD analysis of the case, results in disadvantageous repercussions with respect to the required permission of turbulent interference by the riblet blades. In provision of further clarification of this outcome, the acquired integral wall shear stress plots of both hydroplane test cases have been issued below, in accompaniment to the comparable plots of each along the spanwise direction of the hydroplane body [Figure 53, 54, 55]. These figures provide similar clarification to those outlined in respect to the flat plate test

cases. However, due to the position of the riblet structures leading edge along streamwise length of the hydroplane body, there is substantial increase in the developed wall shear stress over the individual leading edge of each riblet blade, leading to subsequent increase in the accumulative wall shear stress of the entire body. This is then another issue that could be alleviated through simulation of a riblet applied structure in scale with a Reynolds number of one million, as the reduced geometrical properties of each riblet blade, in conjunction with an entirely turbulent regime, is considered to provide reduced significance on the inflicted wall shear stress upon leading edge collision. However, further exhaustive study into the location of riblet application would also be required in order to provide said structure as a wholly comprehensive solution to skin friction drag.

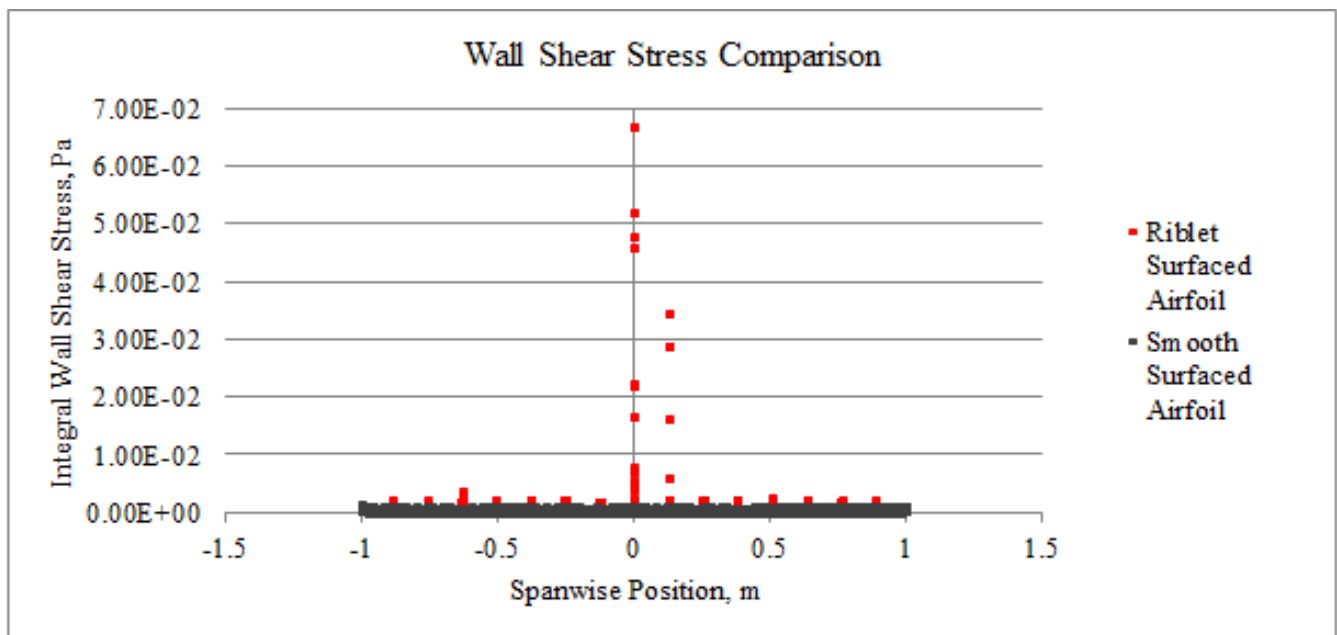


Figure 53, Integral Wall Shear Stress Development in Relation to Spanwise Location of Hydroplane Body,

Inherent to both Smooth and Riblet Applied Test Cases

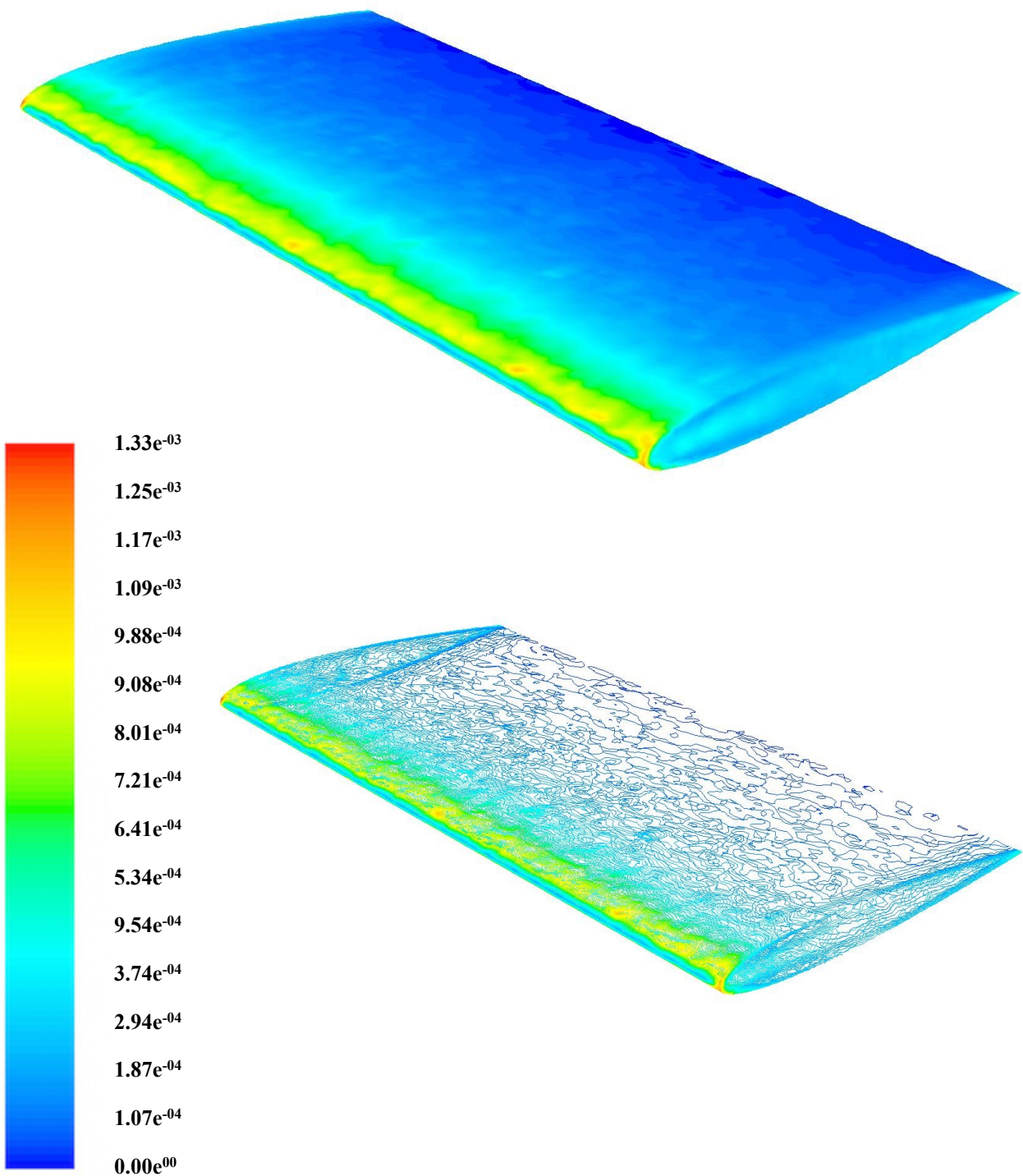


Figure 54, Contour Plot of Integral Wall Shear Stress Development, Inherent to Smooth Surfaced Hydroplane
Test Case

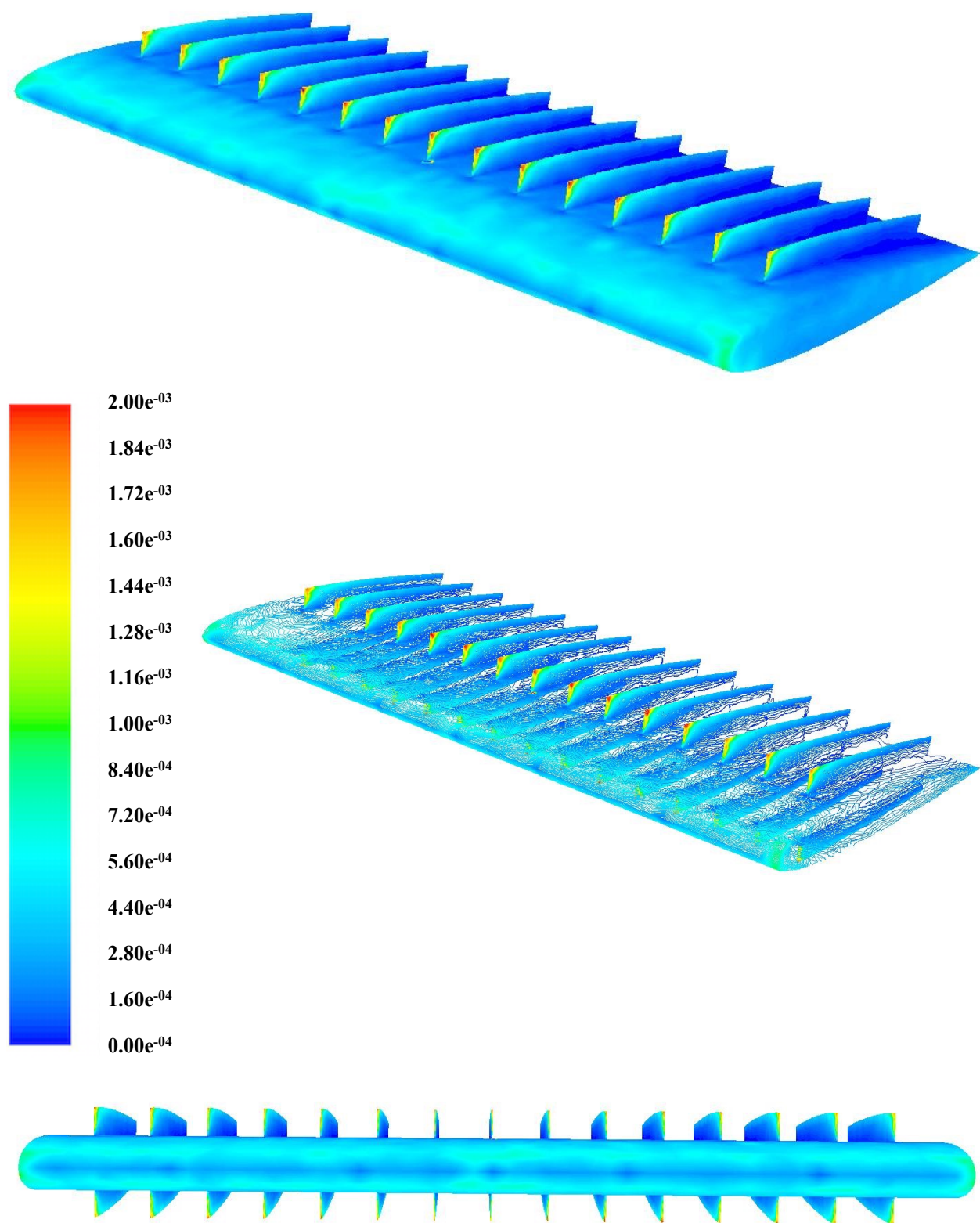


Figure 55, Contour Plot of Integral Wall Shear Stress Development Inherent to Riblet Applied Hydroplane
Test Case

INTERPRETATION OF PROJECT RESULTS

7.1 Imposed Knowledge Contributions of Study

The imposed knowledge contributions of this study are best construed through evaluation of each analytical outcome, as such, forming summation of the proceeding chapter of articulated results [Chapter 6]. In doing so the following points have been deemed to present the main findings evident within the conducted analysis of this study.

i. The analysis conducted in realisation of this study has presented the fundamental requirement of balance between the geometrical attributes of the developed shark skin inspired riblet structure and the advantageous interaction of said structure within any surrounding fluid. The structure itself must then meet the demands of instigation of multiple features upon contact with the encompassing fluid, in order to present any notable reduction in the skin friction resistance of it's encompassing body. The evidential properties of the resultant balance are outlined by the following figure, showcasing the need for multiple features to present themselves in order for the riblet structure to be deemed successful in its main objectives as a passive drag reducing method [Figure 56].

ii. A secondary area of contributory knowledge stems from evaluation of results outlined in regard to hydroplane application by an optimised riblet structure [Section 6.3]. It can henceforth be expressed that in order for comprehensive investigation of the effects induced by riblet application to be take place, the structure must be scaled at fully turbulent Reynolds characteristic. The scale of this analysis would either have to be met with the resources of vast computational processors or through further exhaustive evaluation into methods of reducing the overall operation, a factor discussed within the areas of further study outlined by this thesis [Section 7.3].

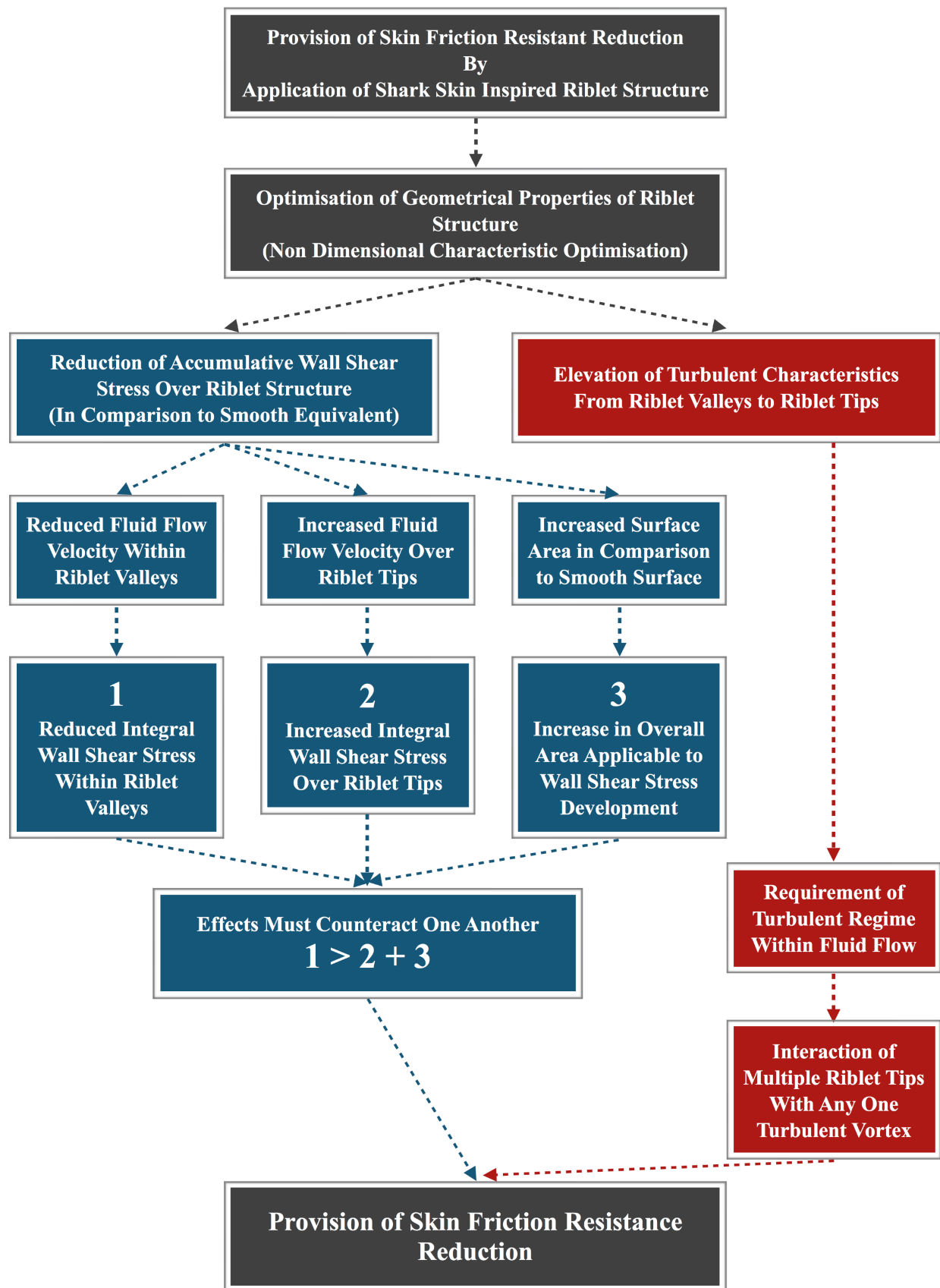


Figure 56, Balance of Introduced Characteristics of Riblet Structured Surface in Provision of Drag Reduction

7.2 Evaluation for Industrial Application

The effects presented by the flat plate test cases of this study convey the evidential advantageous effect that application of an optimised riblet structured surface can have on the encompassing body. This is presented in adherence with the findings articulated throughout the justification of this project and the succeeding review of past literature, contained by sections 2 and 4, whereby, a reduction of friction drag has been obtained in the region of 10% when compared to a smooth equivalent surface. The attainment of results of similar magnitude by application of such a surface towards industrial integration, in particular towards marine utilisation, emphasises the resultant effects of reducing surface friction resistance on fuel consumption, marine emissions and vessel maintenance. However, if future integration of shark skin inspired designs are to become reality, the need for further, fully comprehensive, theoretical and experimental study has also been shown to be of necessity, through evaluation of the applied hydroplane test cases of this study. This demand for further exploration presents to a new era of sharkskin development, where analysis of the effects of sharkskin application on simplified geometries is now proven to have significant beneficial relation towards the bodies accumulative drag, as such presenting a need for future study to focus on the effects of such structures within ‘real life’ application. This conclusion has been drawn due to the need to relay the findings of past research towards analysis on not only realistic structures, for example the hydroplane geometries examined within this study, but also within realistic flow conditions, in order to evaluate the abilities of such structures within their destined commercial practice. In advancement with analysis of this magnitude, the need for future study to explore new methods of analytical investigation also becomes apparent, due to the large time and computational burdens exhibited by CFD analysis of any complex geometry. In doing so, this permits the realistic application of shark skin inspired surfaces within the commercial sector to be evaluated

in greater accuracy, in turn, advancing their widespread introduction towards thorough practical trials and the perceived subsequent integration of varied marine employment.

7.3 Development for Future Study

The scope of related future study in reference to this project is vast, with variations possible in almost every aspect of the conducted analysis. For instance, the presented research has been conducted through optimisation of a blade riblet configuration, characterised by variances of the applied non dimensional lateral spacing. However, future study of alternate riblet configurations, either stemming from those of past publications (such as sawtooth, scalloped or the three-dimensional designs discussed through section 4.4.2) or from the creation of a unique riblet configuration (encompassing a range of attributes endorsed by these designs), would certainly permit evaluation of each through in depth comparison, in the provision of a fully optimised geometry for increased levels surface friction resistance reduction. In further keeping with the configurative process applied in creation of the investigated riblet structure, alterations to secondary riblet characteristics could also be implemented. These additional alterations stem from the inclusion of a fixed non dimensional lateral spacing to height and thickness ratio within this study, where modification to these parameters could also take place for any riblet design. The above alterations would permit evaluation of an extensive range of riblet structures, allowing the individual characteristics of each to be expressed in relation to practical implementation. In doing so, it would be expected for particular configurations to present themselves for inclusion in varying realistic applications, such as towards internal or external flow regimes.

Evaluation of the computational analysis pertinent to this study, results in a range of possible adaptations of the theoretical process. As such, future study of the cases presented by this thesis, could be conducted through implementation of varying turbulence models. However,

as has been discussed previously, the need for further mesh refinement of each case is fundamental to attaining advantageous results with other models, for example the desired k- ϵ turbulence model. In addition, further study could be attempted in the analysis of additional flat plate test cases, scaled by a Reynolds characteristic of one million. This would then alleviate the computational demands of the generated mesh, when compared to testing upon hydroplane application, allowing the fluid flow influences of varying riblet configurations to be evaluated in their drag reducing effect, at significantly increased turbulent regime. Conclusion of this additional testing would then provide judgement as to whether application of an optimised structure is necessary for hydroplane evaluation, by which, further optimisation of the individual case would be permitted in allowing said analysis to commence. In adhering to such computationally intensive investigation, the possible outcomes of the study would have to be considered at the cost of analytical progression, forming a matter of future study in itself.

Additional areas of possible advancement with this study then rest in the experimental application of the optimised riblet structure at non dimensional spacing of $s^+ = 25$, through the multiple methods outlined by section 4.6. Upon attainment of experimental results, the particular advantages of the individual case could be more exhaustively evaluated, promoting the structure further as a desirable drag reducing method.

PERSONAL DEVELOPMENT & REFLECTION

This study has widely developed many aspects of my academic capabilities, with each individual chapter of this thesis presenting a unique opportunity for advancement of varying aspects of professional evolution. Upon commencement of this study, the need for justification of the desired research allowed me to progress in my understandings of the commercial sector, by which, the much publicised reduction in skin friction drag applicable to shark skin inspired surfaces is not solely sufficient for project initiation. The resultant outcomes of this section relay the importance of achieving a balance between an advantageous result and the cost and time demands in attainment of said result, which has allowed me to develop my opinions on what constitutes to an acceptable degree of risk. Furthermore, this section and the succeeding literature review have enabled me to progress as a researcher, with the importance of collating knowledge and figures from a wide spectrum of past publications proven to be of resultant benefit, allowing the personal reflections expressed to be of sound acquirement.

Upon inception of this project, my personal knowledge of not only the Ansys Workbench 15.0 suite, but of the CFD field was inherently noviciate. However, through exploration of the articulated findings evident in reviewing past publications, in conjunction with the validation and verification process of this study, I have been able to substantially broaden my knowledge of each simultaneously. In turn this has allowed me to conduct the required CFD analysis inherent to this investigation with an increased perception on the accuracy and correctiveness of the geometrical, meshing and computational configurations in application at each stage, in presentation of a unified research viewpoint. In conclusion, this study has presented itself as an infallible opportunity to advance as a student, researcher and academic professional, with further advancement in the direction of my future career.

PROJECT CONCLUSIONS

In final conclusion of this thesis, the formerly articulated objectives of the study, outlined by section 3, have been revisited. The succeeding summarisation then details the evidential performance of the final study in adhering to these initial stipulations of direct aim.

- I. Upon initiation of this project, a detailed investigation into the unique influences of shark skin inspired surfaces on the resultant friction resistance of an enveloped body has been conducted, showcasing the mechanisms presented by said structures in conjunction with the previously publicised characteristics of geometrical configuration. In result of this analysis, three distinct models of varying non dimensional riblet lateral spacing have been developed through optimisation of past publications. Furthermore, examination of the theoretical models applicable to investigation of riblet structured surfaces has been outlined, with the resultant findings utilised in establishment of the configurations applied for each of the main test cases presented by this thesis.
- II. A comprehensive validation and verification process has been undertaken in further presentation of the theoretical models applicable to study of riblet structures, with additional knowledge presented towards advancing as a CFD researcher, and user of the utilised software. In addition, the provision of results of sufficient proximity to those of past research presents evidential backing of the conducted analytical configurations, highlighting the accurate nature of the cases presented.
- III. Articulation of the analytical methodology employed in creation of riblet applied flat plate test cases has been issued extensively, with the corresponding outcomes of

computational analysis presenting a conquering geometrical configuration ($s^+ = 25$), in relation to its individual influence on integral wall shear stress development. The characteristics of the induced fluid flow throughout each model domain have also been evaluated, in view of explanation towards their differential effect on skin friction resistance.

- IV. The application of the above mentioned optimum riblet geometry has then been detailed in regard to its application towards a hydroplane body. This exploration was conducted as a means of presenting said structure as a notable drag reducing method for inclusion on realistic geometries, however, the necessary actions applied in order to proceed with the investigation under the computational and time constraints of the study, have resulted in disadvantageous effect on the evaluated results.
- V. Detailed interpretations of the results pertinent to each test case of this study have been issued in the provision of clear evidence towards the projects finding and subsequent knowledge contributions, whereby, the scope of possible future study has been discussed.

Collation of the above conclusions present indisputable similarity to the aforementioned objectives stipulated upon commencement of this study. In result, it can be deduced that the articulations contained throughout this thesis adhere to all aspects of the planned study, in turn, relaying a comprehensive viewpoint on the role of sharkskin inspired surfaces on surface friction reduction and their consequent development for commercial endorsement.

Word Count to this Point Approximately 21950

REFERENCES

1. Rebecca Paul. (2013). *Design Inspiration from Nature - Biomimicry for a Better Planet* Read more: *Finding Inspiration in Nature* . Available: <http://inhabitat.com/finding-design-inspiration-in-nature-biomimicry-for-a-better-planet/3/>. Last accessed 25th November 2014.
2. Frank E. Fish, Donna M. Kocak. (2011). *Biomimetics and Marine Technology: An Introduction*. Available: <http://darwin.wcupa.edu/~biology/fish/pubs/pdf/2011MTSJ%20Intro-.pdf>. Last accessed 30th July 2014.
3. Stephanie Vierra. (2014). *Biomimicry: Designing to Model Nature*. Available: <http://www.wbdg.org/resources/biomimicry.php>. Last accessed 26th Nov 2014.
4. Yee Shin Khor, Qing Xiao. (2011). CFD Simulations of the Effects of Fouling and AntiFouling . *Elsevier Ocean Engineering*. 38 (38), 1065-1079.
5. Clark, R. (2015). *Electronmicrograph Shows Sharkskins Dermal Denticle*. Available: <http://exploregram.com/2of2-an-electronmicrograph-shows-the-sharkskins-the-which-are-the-2/>. Last accessed 20th April 2015.
6. Brian Dean, Bharat Bhushan. (2010). Shark-skin Surfaces for Fluid-Drag Reduction in Turbulent Flow: A Review. *Philosophical Transaction of The Royal Society*. 368 (1929), 4775-4806.
7. Lee, S.J, S.H, Lee. (2001). Flow Field Analysis of a Turbulent Boundary Layer over a Riblet Surface. *Experiments in Fluids*. 30 (2), 153-166.
8. Bechert, D, et al. (1997). Experiments on Drag-Reducing Surfaces and their Optimisation with an Adjustable Geometry. *Journal of Fluid Mechanics*. 338 (1), 59-87.
9. Adrian Bishop. (2012). *Shark-Skin Boosts Swim Speed Cuts Drag*. Available: <http://www.earthtimes.org/nature/shark-skin-boosts-swim-speed-cuts-drag/1818/>. Last accessed 19th November 2014.

10. Smith, R. (2009). *Swimsuits Banned as 'Technology Doping'*. Available: <http://www.symscape.com/blog/swimsuit-banned-as-technology-doping>. Last accessed 1st April 2015.
11. Robert Roy Britt. (2005). *Fake Shark Skin Could Make Navy Fleet Faster*. Available: <http://www.livescience.com/312-fake-shark-skin-navy-fleet-faster.html>. Last accessed 19th November 2014.
12. Bechert, D., et al., Biological surfaces and their technological application- laboratory and flight experiments on drag reduction and separation control. AIAA paper, 1997a. 1960.
13. Bureau of Transportation Statistics F41 Schedule P12A. (2015). *Airline Fuel Cost and Consumption (U.S. Carriers - Scheduled)*. Available: <http://www.transtats.bts.gov/fuel.asp>. Last accessed 1st April 2015.
14. Zabi Bazari, Tore Longva. (2011). *Assessment OF IMO Mandated Energy Efficiency Measures For International Shipping*. Available: <http://www.imo.org/MediaCentre/HotTopics/GHG/Documents/REPORT%20ASSESSMENT%20OF%20IMO%20MANDATED%20ENERGY%20EFFICIENCY%20MEASURES%20FOR%20INTERNATIONAL%20SHIPPING.pdf>. Last accessed 26th November 2014.
15. BP Workbook of Historic Data. (2012). *Crude Oil Prices from 1961 to 2011*. Available: http://en.wikipedia.org/wiki/Price_of_oil#/media/File:Crude_oil_prices_since_1861.png. Last accessed 12th March 2015.
16. Pedersen, M.F. (2008). *International: IMO Marine Engine Regulations*. Available: <https://www.dieselnet.com/standards/inter/imo.php>. Last accessed 12th March 2015.
17. Global Security . (2012). *Polymer Drag Reduction*. Available: <http://www.globalsecurity.org/military/world/ssn-drag-reduction-polymer.htm>. Last accessed 12th March 2015.
18. Munson, B.R., et al., Fundamentals of fluid mechanics2009: Wiley.
19. Zaid B. Jildeh. (2014). *Which Turbulence Model Should I Choose for my CFD Application?*. Available: <http://www.comsol.com/blogs/which-turbulence-model-should-application/>. Last accessed 12th March 2015.

20. Kline, S., et al., The structure of turbulent boundary layers. *Journal of Fluid Mechanics*, 1967. **30**(04): p. 741-773.
21. Starling, I., and Choi, K. S., 1997. Non-linear Laminar-Turbulent Transition over Riblets. *Proceedings of the Laminar Flow Workshop*, Queen Mary and Westfield College, London.
22. Bechert, D, M.Bruise & W. Hage. (2000). Experiments with Three-Dimensional Riblets as an Idealised Model of Shark Skin. *Experiments in Fluids*. 28 (5), 403-412.
23. Sheng, J., Malkieland, E., and Katz, J.,. (2009). Buffer Layer Structures Associated with Extreme Wall Stress Events in a Smooth Wall Turbulent Boundary Layer. *Journal of Fluid Mechanics*. 633 (1), 17-60
24. Bacher, E. V., and Smith, C. R. (1985). A Combined Visualisation-Anemometry Study of the Turbulent Drag Reduction Mechanisms of Triangular Micro-groove Surface Modifications. *American Institute of Aeronautics and Astronautics*. 85 (0548), 1.
25. Rohr, J., et al., A comparison of the drag-reducing benefits of riblets in internal and external flows. *Experiments in fluids*, 1992. **13**(6): p. 361-368.
26. Bechert, D. W., and Bartenwerfer, M. (1982). The Viscous Flow on Surfaces with Longitudinal Ribs. *Journal of Fluid Mechanics*. 206 (1), 105-129.
27. Bushnell, D. M. (2003). Aircraft drag reduction – A review. *Proceedings of Institute of Mechanical Engineers*. 217 (G), 1-17.
28. Vukoslavcevic, P., Wallace, J. M., and Balint, J. L. (1992). Viscous Drag Reduction using Stream-Wise Aligned Riblets. *American Institute of Aeronautics and Astronautics*. 30 (1), 1119-1122.
29. Wilkinson, S. & B. Lazos. (1987). Direct Drag and Hot-Wire Measurements on Thin Element Riblet Arrays in IUTAM. *Symposium on Turbulence Management and Relaminarization*. 12 (p), 19-23.
30. Choi, H., P. Moin, and J. Kim, Direct numerical simulation of turbulent flow over riblets. *Journal of Fluid Mechanics*, 1993. **255**: p. 503-503.

31. Dr. Q Xiao (2014). *Theory and Practice of Marine CFD*. University of Strathclyde: Glasgow. 1.
32. John.D.Anderson, JR (2013). *Computational Fluid Dynamics: The Basics with Application*. 6th ed. Maryland: McGraw Hill Education. 3-278.
33. Oyewola Miracle Olanrewaju. (2004). Influence of Riblets on the Suction Effect on a Turbulent Boundary Layer. *Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering*. 1 (1), full text
34. CFD Online. (2013). *Turbulence Modelling*. Available: http://www.cfd-online.com/Wiki/Turbulence_modeling. Last accessed 12th April 2015.
35. André Bakker. (2006). *Applied Computational Fluid Dynamics*. Available: <http://www.bakker.org/dartmouth06/engs150/12-les.pdf>. Last accessed 26th November 2014.
36. Bao-wei Song, Zhan-yi Liu, Ting Xu, Hai-bao Hu, Ming-ming Huang. (2009). *A method for testing drag-reduction on riblet surfaces based on the Spalding formula*. Available: <http://link.springer.com/article/10.1007/s11804-009-8073-5>. Last accessed 16th November 2014.
37. Boden, F. (2014). *Particle Image Velocimetry (PIV)*. Available: <http://aim2.dlr.de/measurement-techniques/particle-image-velocimetry-piv/index.html>. Last accessed 12th November 2015.
38. Nadu, T. (2008). *LDV - Laser Doppler Velocimetry*. Available: http://velocimetry.net/ldv_principles.htm. Last accessed 3rd November 2014.
39. Ansys . (2015). *Ansys Workbench Platform*. Available: <http://www.ansys.com/Products/Workflow+Technology/ANSYS+Workbench+Platform>. Last accessed 3rd April 2015.
40. Abaqus Verification Manual. (2013). *Time Accurate Laminar Vortex Shedding*. Available: <http://abaqus.ethz.ch:2080/v6.11/books/ver/default.htm?startat=ch03s03abv179.html>. Last accessed 15th April 2015.
41. *Has Your Simulation Converged?*. Available: <http://support.esi-cfd.com/esi-users/convergence/>. Last accessed 20th March 2015.

42. Douvi C. Eleni*, Tsavalos I. Athanasios and Margaris P. Dionissios. (2012). Evaluation of the turbulence models for the simulation of the flow over a National Advisory Committee for Aeronautics (NACA) 0012 airfoil. *Journal of Mechanical Engineering Research*. 4 (3), 100-111.
43. Mullen, B. (2014). *Flow Over an Airfoil*. Available: <https://confluence.cornell.edu/SIMULATION/Flow+over+an+Airfoil+-+Numerical+Results>. Last accessed 24th March 2015.
44. Airfoil Tools. (2015). *NACA 0012 AIRFOILS (n0012-il)*. Available: <http://airfoiltools.com/airfoil/details?airfoil=n0012-il>. Last accessed 12th April 2015.
45. Kim, S.-W., A fine grid finite element computation of two-dimensional high Reynolds number flows. *Computers & fluids*, 1988. **16**(4): p. 429-444.
46. Gartling, D.K., A test problem for outflow boundary conditions—flow over a backward-facing step. *International Journal for Numerical Methods in Fluids*, 1990. **11**(7): p. 953-967.
47. Jimenez, J. and P. Moin, The minimal flow unit in near-wall turbulence. *Journal of Fluid Mechanics*, 1991. **225**(213-240).
48. Larsson, J and Karwai, S. (2010). Wall-modeling in large eddy simulation: length scales, grid resolution and accuracy. *Center for Turbulence Research*. 39 (1), 40-46.
49. Gde A. Prabhawatya Poundra 1 , Alfred Agry Mohamad 1 , Ali Munazid 1 , and Bagiyo Suwasono. (2014). HYDROPLANE DEVELOPMENT AS DIVING PLAN EQUIPMENT ON A SUBMARINE. *Naval Architecture and Ship Building Engineering*. 1 (1), 1-15.
50. Dinesh Kumar G, Vignesh Kumar R, Ajay Kumar.M, Srikanth J.V.R and Soorya Prasad. (2013). Numerical Investigation of Viscous Drag Reduction Using Riblets. *Advances in Aerospace Science and Applications*.. 3 (2)
51. Johan Hoffmana, Johan Janssona, Niclas Janssona. Simulation of 3D unsteady incompressible flow past a NACA 0012 wing section. *Computational Technology Laboratory, Report KTH-CTL-4023*.

References

52. Lindeburg, M.R., Mechanical engineering reference manual 1994: Professional Publications.
53. Bruneau, C.-H. and M. Saad, The 2D lid-driven cavity problem revisited. Computers & fluids, 2006. **35**(3): p. 326-348.
54. Armaly, B., et al., Experimental and theoretical investigation of backward-facing step flow. J. Fluid Mech, 1983. **127**(473): p. 20.
55. Kim, J.J., Investigation of separation and reattachment of a turbulent shear layer: flow over a backward-facing step. 1978.