POLITECNICO DI TORINO

Master of Science in Mechanical Engineering

Master Thesis

Development of a CFD method for bicycle design

Superadvisor:

Prof. Renzo Arina

Michael Adomeit

Candidate:

Craig Euan McLennaghan

ACADEMIC YEAR 2020/2021

Abstract

This thesis details the process undertaken to establish a method of CFD analysis for two-dimensional profiles to establish drag coefficient using SolidWorks Flow Simulation. This research was undertaken in collaboration with Canyon GmbH to develop a method of analysis to influence product design.

Starting using a reference body of known *Cd* value, a methodology was defined through research to characterise mesh specifications, to create profile specific quantifiable mesh and calculation controls. These studies were namely focused on the determination of the *Cd* value and further the effective modelling of the expected airflow behaviour for turbulent flows, energy dissipation and frictional forces present. The outcome from this testing was to be able to create a 'best practice' for 2D flow simulation for bluff bodies.

This 'best practice' was then applied to other defined bluff bodies deemed to be similar geometrically to the types of bodies seen in bicycle design. By assessing the ability of SolidWorks Flow Simulation to determine the *Cd* value for the bluff bodies to an error factor, an analysis was then carried out to assess the validity and effectiveness of the methodology.

The preliminary outcome was that a methodology was determined that was able to produce a valid solution for the preliminary development case, this methodology was then applied to case profile studies to assess the ability of the methodology to fit the scope predetermined; to model and simulate a generic profile effectively with accurate and valid results.

The findings of the research concluded that SolidWorks Flow Simulation can be utilised for research and development activities for bicycles, however more work is required to constrain the methodology for more difficult aerodynamic shapes, particularly profiles which show separation before the end of the profile, or boundary layer sensitive cases.

Content

Abstract 2
Content 3
1. Introduction 5
1.1. Low speed aerodynamics theory 6
1.1.1. Drag force 6
1.1.2. Boundary layer 7
1.1.3. Separation 7
1.1.4. Turbulence 8
1.2. Cycling aerodynamics overview 9
1.2.1. Bicycle Aerodynamics 9
1.2.2. Design for aerodynamics 10
1.3. CFD for external aerodynamics 10
2. Characterising method effectiveness 11
2.1. Force in X and coefficient of drag 11
2.2. Turbulent Kinetic energy dissipation 12
2.3. Frictional Force 12
3. Development of methodology 13
3.1. Overview 14
3.1.1. Approach 14
3.2. Development of standard methodology 15
3.2.1. CAD model 15
3.2.2. Domain 15
3.2.3. Solver selection 16
3.3. Boundary conditions 17
3.4. Initial studies 18
3.4.1. Domain initial sensitivity study 18
3.5. Mesh initial sensitivity study 29
3.5.1. Initial Mesh 29
3.5.2. Conclusion from initial study 32
3.6. Optimisation 32
3.6.1. Y+ studies 32
3.6.2. Meshing controls 38
4. Application and validation of developed method 39
4.1. Assessment of methodology 41

4.1.1. CAD import 41
4.1.2. Domain Generation 41
4.1.3. Mesh Generation 42
4.2. Initial study 44
4.3. Results 44
4.3.1. NACA profile 44
4.3.2. Eppler profile 47
4.4. Investigation into the optimisation of the methodology 49
4.4.1. Y+ optimisation 50
4.4.2. Conclusion of optimisation 56
5. Conclusion 57
6. Bibliography 58
Acknowledgements 60

1. Introduction

In the 21st century, aerodynamics and cycling have become almost synonymous with one and other. Through the transfer and adaptation of aerospace knowledge it has been possible to improve the basic design of the bicycle and reconsider this once basic proprietary method of transport as a complex and highly efficient machine, meticulously designed for optimum efficiency and maximal output.

When one considers the history of both the sport and the industry, the connection between performance and aerodynamic efficiency is evident. From the beginning of bicycle racing, competitors have always searched for ways to find the edge over the competition by seeking to improve the aerodynamic properties of both rider and bicycle. This predominantly manifested itself in the early days of bicycle racing through the adoption of different positions and latterly in the improvement of airflow characteristics across the various products developed in the cycling industry. Through these iterations of innovation, the modern racing bicycle was born and with it, modern bicycle racing too, which lead to a demand for manufacturers to search for new ways to gain the competitive edge.

In a bid to remain competitive in both the cycling market and the sport, manufacturers chase increasingly more elaborate ways to optimise their products through weight reduction, improved aerodynamic efficiency and refined mechanical characteristics. One such example of this adoption of technology has been the use of Computational Fluid Dynamics (CFD) to influence the design of cycling products with the end goal of improving the aerodynamic efficiency and lowering the losses in energy from the users work input through airflow energy dissipation and turbulence creation.

This master’s thesis looks to explore the utilisation of CFD in cycling, investigate the applicability and apply a methodology that will enable CFD to be utilised by Canyon GmbH within its research and development activities for bicycles.

1.1. Low speed aerodynamics theory

This section will cover the main concepts of aerodynamic behaviour which are applicable and important to consider for cycling. The fluid dynamic behaviour of airflow that a cyclist will experience (maximum realistically achievable is around 35 m/s) falls into the domain of low speed aerodynamics and hence several topics will be addressed specific to the flow characteristics associated with those airflow speeds [1] [2]

1.1.1. Drag force

Drag force represents the total force acting upon a body when moving through a fluid, this is easily achieved experimentally by measuring the total load present on a body under investigation for fluid flow. This total load is a summation of pressure differences acting across the bicycle and rider. As airflow strikes the front facing surfaces of a body, the airflow loses a large portion of its velocity, which leads to the flow losing dynamic pressure through the equation [3] [4]:

$$q = \frac{1}{2}\rho u^2$$

Where q, the dynamic pressure is influenced by the density ρ and the square of flow velocity u.

This reduction in dynamic pressure is complemented by an increase in the static pressure on this same area of reduced velocity flow through Bernoulli's theory [4]:

$$Total\ Pressure = Static\ pressure + Dynamic\ pressure$$

Hence it can be observed that variations in airflow velocity across a body equate to variations in dynamic pressure and hence static pressure in the airflow surrounding the body. It is the difference in static pressures upstream (in front of a body in airflow) and downstream (behind a body in airflow) that creates a force acting upon the body, against the direction of airflow travel.

The drag equation captures the effects of the high pressure on the frontal area of the body, along with the expected low-pressure area rearwards of the profile, which hence gives a force vector. This drag force is most useful in shape profile optimisation by using this force to characterise the aerodynamic efficiency of an object through the coefficient of drag C_D, which is a dimensionless coefficient representing the drag or resistance of an object moving through a fluid. This coefficient is defined in the following equation [5]:

$$C_D = \frac{F_D}{\frac{1}{2}\rho A v^2}$$

Where:

F_D is the drag force acting on the body

ρ is the mass density of the fluid

A is the frontal or reference area of the profile

v is the relative velocity between the fluid and the body moving through the fluid

1.1.2. Boundary layer

In understanding deeper the concept of airflow velocity and how it acts on a body, one key concept is the boundary layer [6, 7]. As fluid flow comes into contact with a surface, the first molecular layer of the fluid attaches itself to the surface of the body through fluid viscosity, from the initial layer at velocity zero, the subsequent airflow velocities grow according to the friction between the flow and its subsequent layers according to the shear forces present. These velocities grow until reach the normal airflow velocity at a specific point. This region of velocity gradients is referred to as the boundary layer [6] [7] [8].

Figure 1, shows diagrammatically this variation from surface to free stream and where the boundary layer presents itself for a fluid flow [6].

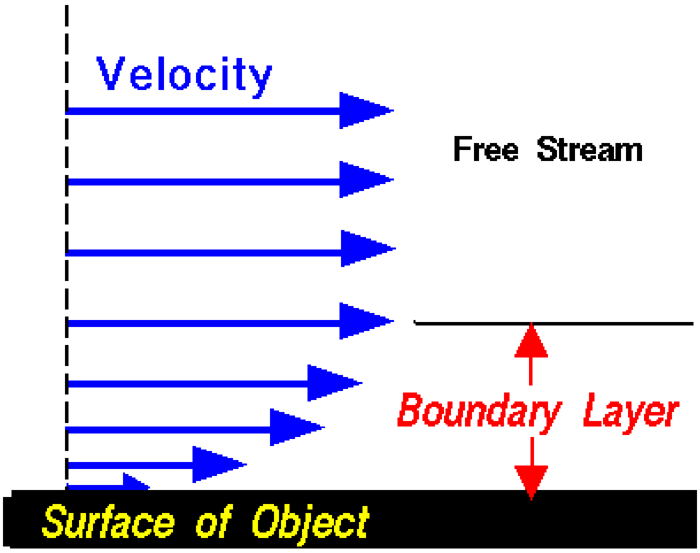

FIGURE 1, BOUNDARY CLASSIFICATION ACCORDING TO SOURCE

Hence, the size and behaviour of the boundary layer is dependent on the fluid velocity, fluid viscosity and surface parameters of the object to which the air flows over. Different shaped profiles and bodies exhibit different boundary layer behaviours which are fundamental to understanding external aerodynamics and its implications in CFD [6] [9].

1.1.3. Separation

When the boundary layer is travelling along a surface and encounters an adverse pressure gradient created by the outer free stream flow, typically at a point of higher relative angle between the external flow and the surface to which the fluid is attached [10]. This causes the closest layers of fluid flow to reverse and this flow reversal causes detachment of the boundary layer to the surface of the body. Once separated, eddies and vortices are likely to form and the characteristics of these eddies and vortices become dependent on the fluid flow and fluid properties [11].

The behaviour of fluid travelling along a specific body is therefore characterised by these velocity gradients, and how the geometry of a body encourages flow separation at a particular point, under specific conditions of fluid parameters. Figure 2 [12] details the fluid following a path until it experiences the negative pressure gradient as described.

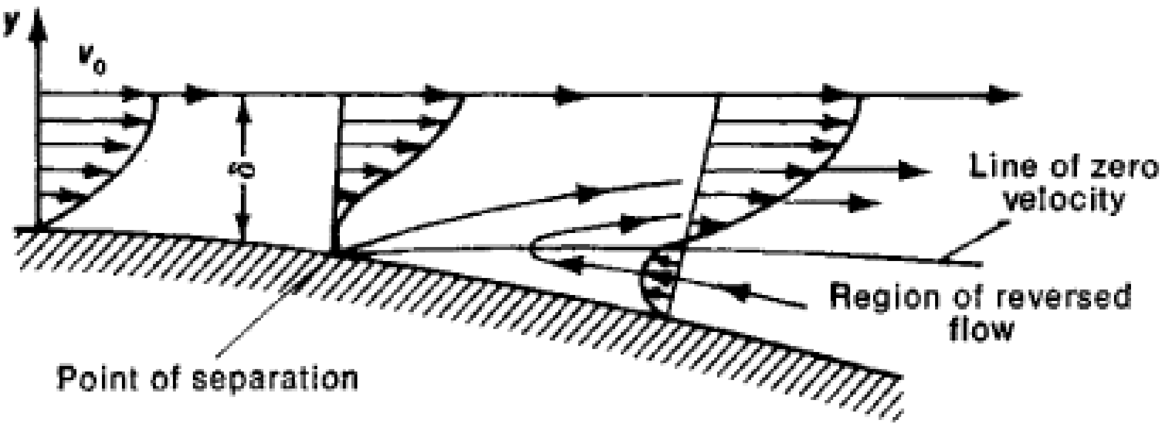

FIGURE 2, DETAIL OF THE POINT OF SEPARATION WITH THE INCREASE IN RELATIVE ANGLE BETWEEN AIR FLOW AND SURFACE

1.1.4. Turbulence

Turbulence refers to flow that is characterised physically and mathematically as chaotic in terms of its changes in velocity and pressure. Turbulent flows are created by the kinetic energy in some portions of flow being able to overcome the damping of the viscosity of the fluid, creating deviations from the fluid flow path [13].

This summation of forces can be best described by the Reynolds number, a non-dimensional quantity that characterises a flow through the ratio between the inertial forces and viscous forces present in the flow, this is described in the following equation as follows [13]

$$Re = \frac{Inertial\ forces}{Viscous\ forces} = \frac{vL}{\nu}$$

Where v is the relative fluid velocity, L the characteristic linear dimension and ν the kinematic viscosity. This number is then used to characterise whether a flow is turbulent (high number and therefore inertial force dominant) or laminar (lower number and therefore dominated by viscous forces) for a specific fluid behaviour. This can also be applied locally to define the behaviour of a specific point in a flow to better characterise the behaviour and changes in fluid dynamic properties.

From understanding the effect that the Reynolds number has on a flow, and how the flow is effected by the geometry of the body, a relation can be observed between flow characteristics downstream of the body (and thus created by the body) and the forces presented upon the body that moves through a given fluid.

1.2. Cycling aerodynamics overview

Aerodynamic drag is the single most important factor to consider when aiming to improve a bicycles performance in terms of energy efficiency. Anywhere between 60-90% of the overall resistive forces experienced by a cyclist at commonly experienced cycling speeds, can be attributed directly to aerodynamic drag [1].

Of the total aerodynamic forces, a significant percentage of the force is attributed to the cyclist and the aerodynamic phenomena induced by airflow across the cyclist's body. The overall contribution of the bicycle in drag generation is around 20-30% for a nominal air flow speed of around 12 m/s [14]

Despite the lower percentage of drag force that comes from the bicycle itself, a great deal of time and effort goes into improving bicycle aerodynamics as significant performance gains can be made through optimising the product to improve airflow characteristics.

1.2.1. Bicycle Aerodynamics

For high performance bicycle applications, a mean fluid flow value of 12 m/s is typically accepted, this value is generally accepted in industry to be used when developing and analysing aerodynamic performance.

As far as simulation / replication of fluid flow behaviour in a real-world environment, several approaches have been taken and modified to consider the following:

- Wind gusts of varying intensity
- Different angles of attack due to varying environment wind velocities
- Different angles of attack from the bicycle relative orientation
- Varying conditions air flow conditions (riding in the slip stream of another rider or in a bunch group)

Therefore, prior to developing the simulation it is necessary to define specific criteria for the air flow properties:

- Air flow velocity
- Bicycle / profile orientation
- Gas properties (density, temperature, humidity etc)

1.2.2. Design for aerodynamics

To best design products when considering aerodynamics, the industry focuses on a trade-off between aerodynamic efficiency and mechanical properties such as weight, stiffness, or practicality. In recent years, more focus has been placed on developing products which can offer effective aerodynamic efficiency whilst ensuring that the bicycle remains usable and rideable from a customer perspective.

Due to this trade-off, the shape of profiles often in the modern cycling industry can be different to those found in aerospace / automotive applications, particular examples of this are industry wide, with focus on utilising shortened aeronautical air foil profiles being commonplace. Further, efforts are focused on adapting many occurring profiles on the product to change them from being blunt to bluff, in order to improve the aerodynamic efficiency through reducing the coefficient of drag.

This means that for the scope of the described research in this paper, the expectation is to achieve a simulation methodology that will provide appropriate insights into the development, modification and optimisation of bodies and profiles which tend towards the bluff range, but not fully bluff when compared with pure aerospace derived air foils.

1.3. CFD for external aerodynamics

Commonly, established methods and techniques for assessing external aerodynamics come from applications either in the automotive or aerospace industry. These techniques cover design assessment, analysis, diagnosis, and optimisation of technologies and are well equipped to be utilised for the application used for this research.

A variety of methods are used, either to study an entire vehicle overall, or to focus specifically on elemental areas of a body to obtain key insights on aerodynamic performance. The choice of methodology in CFD is always limited by several factors:

- Available computational power
- Available storage for input/output data
- Amount of resources available to develop a methodology
- Level of required insight into aerodynamic performance

Therefore, varying degrees of approach are available and utilised for CFD and external aerodynamics from highly complex full vehicle diagnoses to basic profile flow characterisation.

For applications involving lower speed aerodynamics, the Reynolds number (Re) influences the suitability of specific models which can be applied. For lower speed aerodynamics the Reynolds number is lower ($5.3 \times 10^3 < Re < 2.0 \times 10^4$) and therefore specific observations and ideas can be applied [15] [16]: Most commonly utilised for low speed modelling is Reynolds-Average Navier-Stokes equations (RANS) whereby averaged flow fields are determined and computed, with specific models being applied for the numerical simulation of the turbulence [15].

2. Characterising method effectiveness

In undertaking CFD analysis, it is necessary to determine parameters that will adequately define the accuracy and relevancy of the final solution. These parameters can be used as target values taken from literature to add numerical validation to a particular analysis or can be used as a monitor to determine convergence in a solution.

Initially the development of this method of analysis involved the usage of literature and test cases to develop an appropriate CFD method in two dimensions and hence it was required to validate values and performance against accepted literature results using known benchmark cases (bodies with specific known characteristics).

Specifically, coefficient of drag or Cd was the main characteristic used in terms of defining the analysis accuracy against literature values, whilst Turbulent Kinetic Energy and Frictional Force where used to characterise the effectiveness of the mesh and analysis internally within the CFD computations, and further characterise and control convergence within the static solutions.

2.1. Force in X and coefficient of drag

Drag force represents the total force acting upon a body when moving through a fluid, this is easily achieved experimentally by measuring the total load present on a body under investigation for fluid flow, and can be determined also in CFD as the total force acting along the direction of travel of the body. The equation therefore captures the effects of high pressures expected on the frontal area of the body, along with the expected low-pressure area rearwards of the profile, which hence gives a force vector [5] [9].

This drag force is most useful in shape profile optimisation by using this force to characterise the aerodynamic efficiency of an object through the coefficient of drag C_D, which is a dimensionless coefficient representing the drag or resistance of an object moving through a fluid.

In using CFD programs, this drag force and Cd can be used to determine numerically the effectiveness and accuracy of an analysis by comparing them to known values. Particularly for this project, the analysis method was first established using drag coefficients of known profiles and applying these to newly formed, unknown profiles. This allowed a good degree of validity to be undertaken from the experimentally derived, literature defined drag coefficients.

Furthermore, the drag force and Cd values are used in CFD to determine the convergence of the solution, that these can be used to mathematically characterise stability in the solution of the fluid flow being modelled.

2.2. Turbulent Kinetic energy dissipation

In Reynolds-Averaged Navier-Stokes simulations, another characteristic equation that is important in determining convergence of a particular solution in a particular domain is that of turbulent kinetic energy dissipation (TKE). TKE characterises the mean kinetic energy per unit mass linked to eddy formation in turbulent flows and thus through measuring the velocity fluctuations across a fluid flow, can define the amount of energy which is 'dissipated' through the forces apparent in turbulent fluid flows (namely through fluid shear and frictional forces for low speed aerodynamics) [17].

This can be averaged across a domain by finding the mean turbulence stresses and hence can be applied to define convergence and stability in a calculation solution. For the analyses undertaken, this was used as convergence control to further improve the accuracy and relevance of each analysis, and further to validate the physical properties that where being modelled.

2.3. Frictional Force

Frictional Force describes the averaged frictional force of fluid interacting with the skin of the body which will subsequently create the boundary layer around an object, where the velocity gradients of flow velocity are occurring. This in part characterises the drag force present on a body, but only considers the frictional forces occurring and not the pressure forces acting upon a body [18].

This is also used in convergence control for the analyses to further ensure a stable and valid solution can be achieved. Frictional force is also used to validate the effectiveness of how a mesh can capture the details of fluid in close proximity to the body, by isolating the changes in frictional force with the increase and decrease of cell density around the boundary layers, a better picture can be obtained surrounding the mesh quality.

3. Development of methodology

To aid with development activities at Canyon, it was determined to develop a standard methodology that could be utilised for dealing with the most frequent occurring aerodynamic problems in cycling aerodynamics. The scope of developing the standard methodology was for the following reasons:

- To investigate the feasibility and utility of using SolidWorks flow simulation within the R&D department
- To determine settings which could be applied to specific profiles
- To establish rules and conditions for specific geometrical / airflow conditions

As already discussed in section 1.2, the most commonly occurring profiles in cycling applications vary from standard round cylindrical profiles, to full aerospace inspired profiles, with a general tendency to achieve a compromise between aerodynamic efficiency and mechanical properties.

With this degree of diversity both in the flow conditions along with the type of aerodynamic phenomena that may be observed across different applications, it was therefore necessary to constrain the methodology:

- To be generic as possible within the scope of expected cases
- To impose geometric qualifications and rules
- To investigate rules for resolving issues with the method
- To establish if a generic system of CFD can capture the necessary flow phenomena for typical cases

3.1. Overview

Development of the method follows similar ideas taken from the aerospace, automotive and cycling industries for determining aerodynamic performance. The method was expected to fit into the overall protocol that Canyon adopt when undertaking aerodynamic research.

SolidWorks Flow Simulation software was utilised for the development of the methodology, this was used as it provided an economic and easy to use software with readily available licenses within the company. The aim of this project is to provide a quick and easy CFD methodology which can provide a base overview and specific insights into product development, and therefore it was determined that using such a software would be sufficient for the required tasks.

3.1.1. Approach

The methodology would utilise two-dimmensional (2D) profiles that could be imposed within the simulation as representations of a portion of the specific profile to be assessed. This is a typical approach utilised in industries and is an effective method to investigate basic aerodynamic performance whilst remaining economically efficient. As the main scope of aerodynamic research in the cycling industry is generally for drag reduction, relevant and reliable conclusions can be made already by investigating 2D profiles.

To develop the method, a cylindrical tube profile was used to develop the generic settings which could be transferred to other test cases. This was done under the following hypotheses:

- Using a cylindrical profile would be the least aerodynamically efficient profile encountered in cycling applications:
    - This profile would produce the highest amount of turbulence and aerodynamic drag
    - This would be the most difficult to model because of the increase in turbulence and other fluid dynamic issues
- The cylindrical profile is well studied for CFD and fluid dynamics and thus it can be easily validated and verified through comparing with literature findings

To further simplify the methodology, a steady state solution was sought, this means that only the average flow field is determined, and conclusions are made about the overall profile performance and flow averages rather than specific details about the flow conditions. This approach was deemed to be acceptable and sufficient to provide insights into performance aerodynamically and investigate the influence that different profiles will have on a flow field.

3.2. Development of standard methodology

The basis of the entire methodology resides around developing a repeatable and controllable CFD method, where key factors and controls are established and detailed within SolidWorks Flow Simulation to provide relevant and accurate results when compared with reality.

As already discussed, the CFD software utilised is SolidWorks Flow Simulation Software, this software is commonly used for undertaking basic analyses for design insights and therefore is well equipped to satisfy the scope of this project.

With the entire project being undertaken using a high-powered desktop computer, it was necessary to attain a resource efficient solution whilst at the same time providing accurate and relevant results.

The following section of this document will detail the work done to create the CFD methodology and further assess its validity. Values and settings specific to the CFD method will be left out in the interest of intellectual property protection.

3.2.1. CAD model

For the base development, a cylindrical profile was used, however the method of developing the CAD model remains consistent for all the profiles studied.

Each CAD file was created in SolidWorks CAD software and then exported into the Flow Simulation environment, thanks to the affinity of the software provider, this was easily done through import functions in SolidWorks.

For the aerospace profiles, rather than importing a base CAD shape, the shape was developed using computationally generated coordinates for each respective profile. These are added as a basic line profile within SolidWorks CAD and then mirrored and extruded to create the shapes.

3.2.2. Domain

The domain refers to the total fluid area which will be modelled, minus the area determined to be the profile, it is necessary to appropriately define and constrain the domain.

Controls for the domain are established using offset values from the investigated profile, for a 2D domain, the domain will actually have a thickness making it 3D from the perspective of domain geometry, but 2D constraints are anyway placed on the solver to ensure that only 2D fluid behaviour will be observed.

Before investigating furthermore complex controls for the numerical solution, it was determined to investigate the basic sensitivity of the solution to domain proximity, this allows for a more efficient and effective modelling of the key flow geometry within the system as this is crucial to obtaining a valid and accurate solution.

3.2.3. Solver selection

For CFD applications, it is important to select the correct physical models to be applied to the computational domain. In the case of turbulent aerodynamic modelling, there are many available solvers and models which can be applied.

Whilst in general there are several custom or advanced methodologies which can be applied, the use of SolidWorks Flow Simulation means that the available models are limited compared to other advanced software. For the scope of the research however, the following insights where to be determined, as mentioned previously:

- Evaluation of a given design
- Design optimisation
- Validation and verification of physical results:
    - Coefficient of drag
- Aerodynamic behaviour indications / estimations

With these specific goals in mind, it was therefore determined to assess the available models and methods to develop a methodology which could provide outputs that would satisfy the determined conditions.

Navier-Stokes Equations

SolidWorks Flow Simulation utilises the finite volume method (FVM) to solve Navier-Stokes equations. The following passage describes the conservation laws for mass, angular momentum, and energy:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u_i) = 0$$

$$\frac{\partial \rho u_i}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho u_i u_j\right) + \frac{\partial p}{\partial x_i} = \frac{\partial}{\partial x_j}(\tau_{ij} + \tau_{ij}^{R}) + S_i$$

$$\frac{\partial \rho H}{\partial t} + \frac{\partial \rho u_i H}{\partial x_i} = \frac{\partial}{\partial x_i}(u_j(\tau_{ij} + \tau_{ij}^{R}) + q_i) + \frac{\partial p}{\partial t} - \tau_{ij}^{R}\frac{\partial u_i}{\partial x_j} + \rho\varepsilon + S_i u_i + Q_H$$

Where:

u is fluid velocity

x is displacement

S is mass-distributed external force per unit mass

ρ is fluid density

q is diffusive heat flux

As can be seen in the prior passage, the system of Navier-Stokes equations operates on the principles of conservation laws for mass, energy, and momentum.

In the case of Newtonian fluid behaviour, such as experienced for cycling applications, the following equation is used to describe the shear stress tensor:

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\frac{\partial u_k}{\partial x_k}\right)$$

Interestingly, the software utilises one system of equations for both Laminar (lower Reynolds number) and Turbulent (higher Reynolds number) cases, this means that for the application in question, it is necessary to utilise extra equations that can capture turbulence and its mathematical complexity. In SolidWorks Flow Simulation, the equations for turbulence are known as the Favre-averaged Navier-Stokes. The full closer of the of mathematical system of equations used the k-ε turbulence model whereby turbulence kinetic energy and the dissipation rate are used.

The turbulent kinetic energy is described as follows:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i k) = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right) + S_k$$

Whilst the dissipation term is:

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial}{\partial x_i}(\rho u_i \varepsilon) = \frac{\partial}{\partial x_i}\left(\left(\mu + \frac{\mu_t}{\sigma_e}\right)\frac{\partial \varepsilon}{\partial x_i}\right) + S_e$$

3.3. Boundary conditions

Boundary conditions where defined for the domain. These boundary conditions are defined in accordance with literature-based practices and prior research undertaken within Canyon GmbH. The boundary conditions defined specifically of interest for this research are:

- Domain air velocity
- Domain turbulent factors
- Air flow orientation

Boundary conditions were imposed for the domain inlet and outlet, along with specific coefficients for the domain wall boundaries and the profile itself. These boundary conditions are mostly imposed using the SolidWorks Flow Simulation standard boundary condition values when wall / inlet /outlet is defined.

3.4. Initial studies

Sensitivity studies where carried out for the cylinder benchmark body to determine the effectiveness and accuracy of the simulations undertaken. This was undertaken through the presumption that if fluid behaviour could be adequately and accurately modelled using a less aerodynamically efficient profile (higher CD value and larger amount of turbulence creation) then an effective and accurate model could also be used for analysing the tube profiles, which were aerodynamically shaped.

3.4.1. Domain initial sensitivity study

In external flow fluid dynamic studies domain refers to the specific controlled area of fluid used for an analysis. Sizing of the domain is a crucial factor when modelling fluid behaviour, predominantly due to the fluid dynamic properties being dependent on both the geometry of the body being analysed and the geometry of the fluid and its proximity to walls and inlets/outlets. As previously discussed, this wall proximity is crucial to defining fluid behaviour and can have a significant effect on the outcome of a specific study.

In CFD, domain refers to the computational grid in which fluid is present and modelled. Just as in real-world fluid dynamic studies, the same effects of proximity need to be considered for CFD as the solver attempts to replicate fluid behaviour as closely as possible using the meshing methodology previously discussed.

Procedure

A basic study was carried out to characterize the effects of domain on the simulation, and further obtain an optimal domain size in which it would be possible to sufficiently negate the effects of wall proximity whilst at the same time allow for an efficient domain size with respect to computational load.

A standard mesh density is used without control planes, refinements, or any other mesh alterations. This was done i to negate the effects that other parameters would have on the simulation and be able to purely study and validate the domain size optimisation. A 'standard' domain was calculated using the ratio of domain length/height against profile length/height, to undertake the sensitivity study these ratios where then altered to provide variation.

Each of the studies undertaken (Upstream, Downstream and Wall gap) where assessed by variating the domain size from a standard basic domain size which was calculated according to literature evidence. The iterations for domain size are presented here against the length and height (chord and span) of the body under analysis, to create generic settings which are profile specific for any expected 2D profile to be analysed.

Upstream study

Figure 3 and 4 show the variation between the shortest and longest domain lengths upstream

Here it can be seen clearly the difference in a short (low proximity) and long (high proximity) domain for the upstream domain.

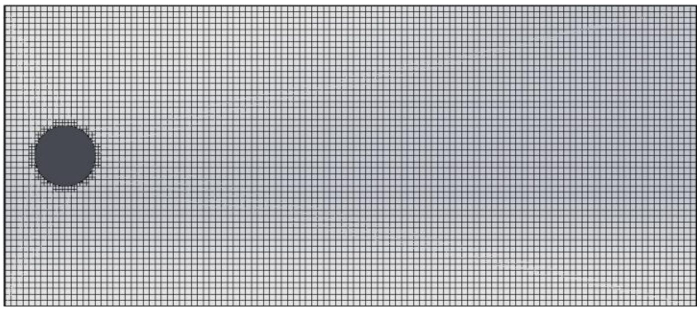

FIGURE 3, SHORTENED UPSTREAM DOMAIN

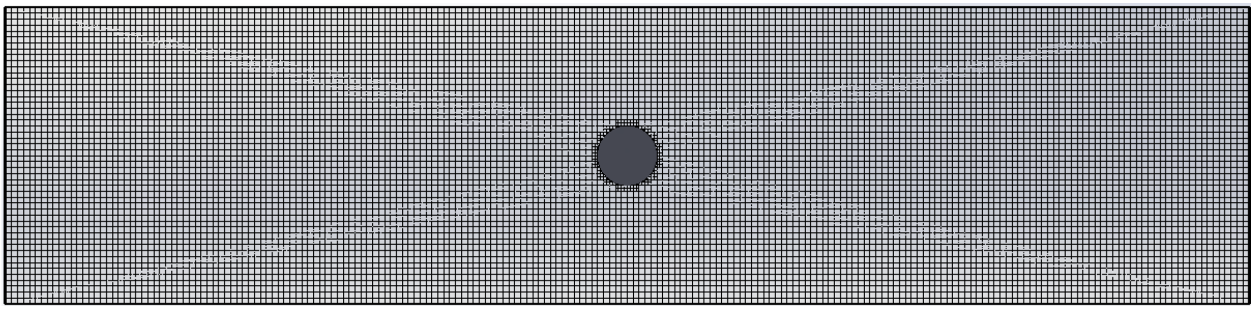

FIGURE 4, LENGTHENED UPSTREAM DOMAIN

Results

Figures 5 and 7 present the static pressure results for the shortest and longest upstream domain respectively, with figures 6 and 8 the velocity plots.

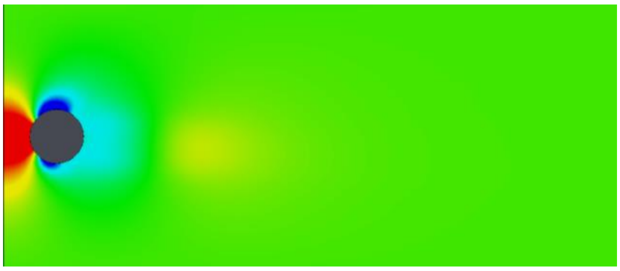

FIGURE 5, REPORTED PRESSURE PLOT SHORT DOMAIN

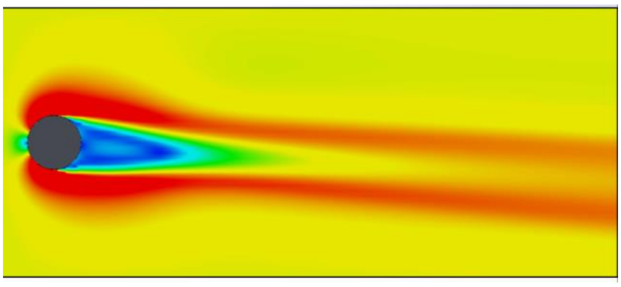

FIGURE 6, REPORTED VELOCITY PLOT SHORT DOMAIN

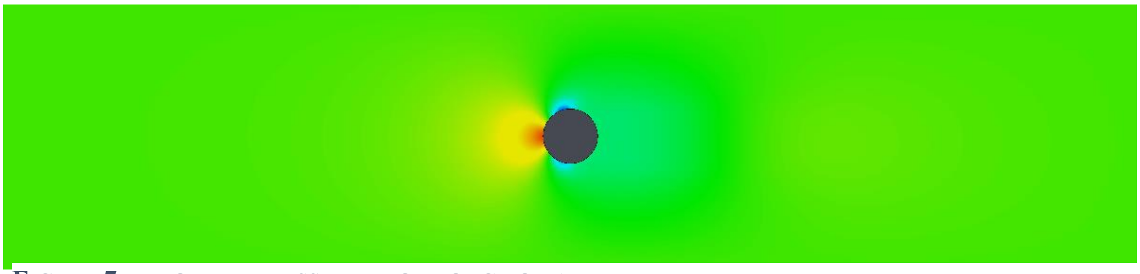

FIGURE 7, REPORTED PRESSURE PLOT LONG DOMAIN

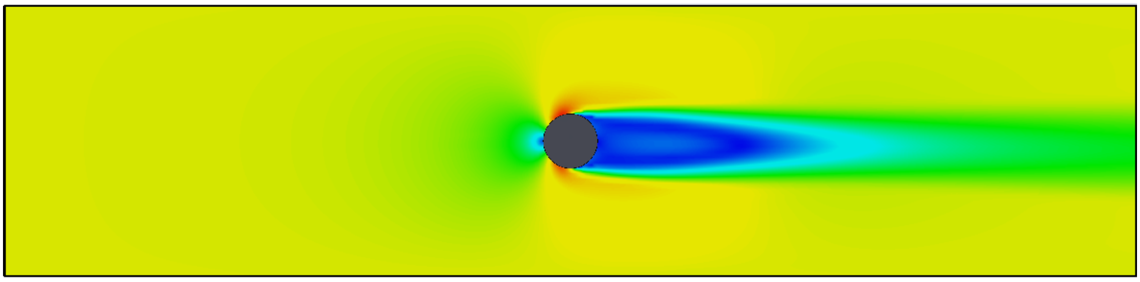

FIGURE 8, REPORTED VELOCITY PLOT LONG DOMAIN

The effect of upstream pressure can be observed to be different for the lower upstream length, indicating an impingement of the flow characteristics around the body, due to the ineffectiveness of the solution for this specific domain size

Figure 6 further confirms the ineffectiveness of the shortest domain size in modelling airflow, as a deviation in the fluid flow path is observed downstream of the body. Below in chart 1, 2 and 3 the values for the validation parameters are defined for Force in X direction, Turbulence Kinetic Energy Dissipation and Frictional Force respectively.

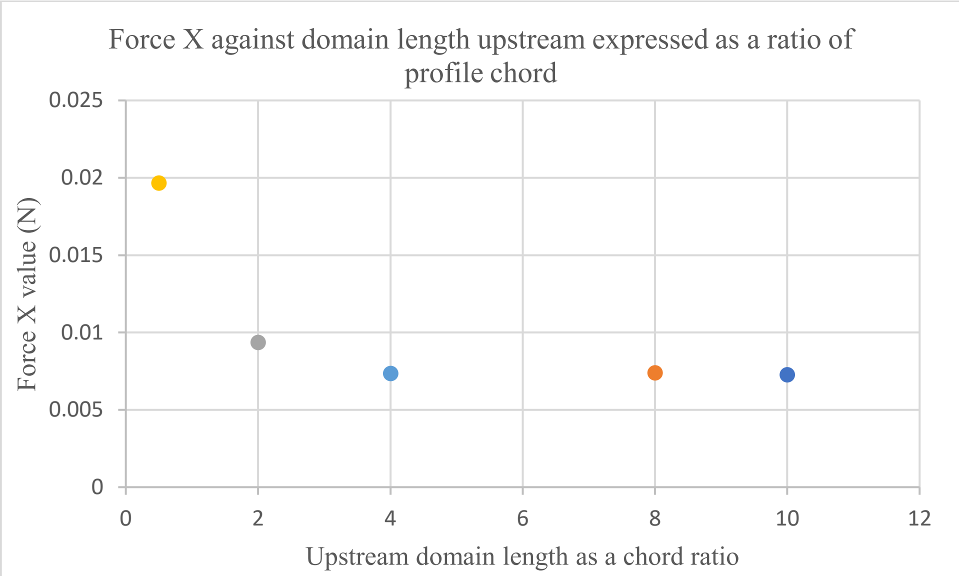

CHART 1, FORCE X SENSITIVITY OUTPUT FOR DOMAIN LENGTH UPSTREAM

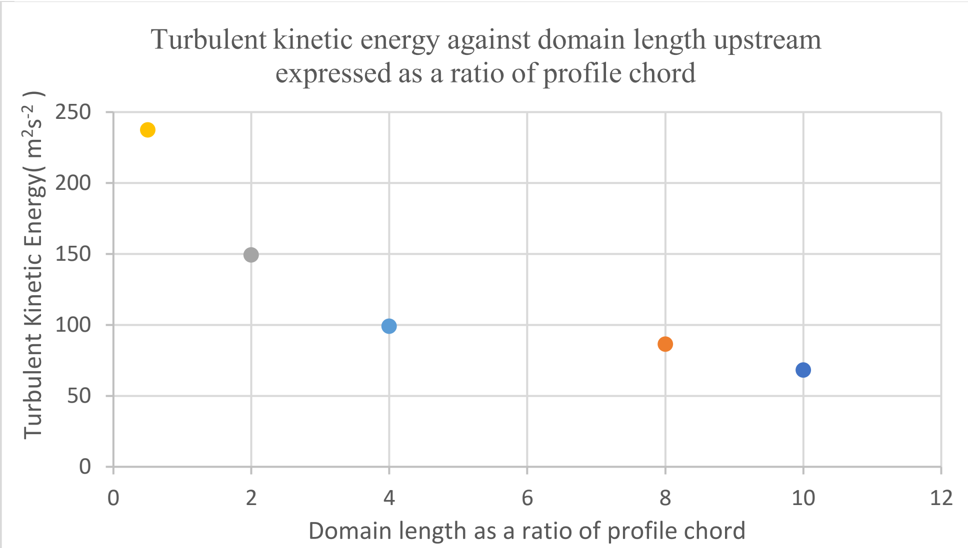

CHART 2, TKE SENSITIVITY OUTPUT FOR DOMAIN LENGTH UPSTREAM

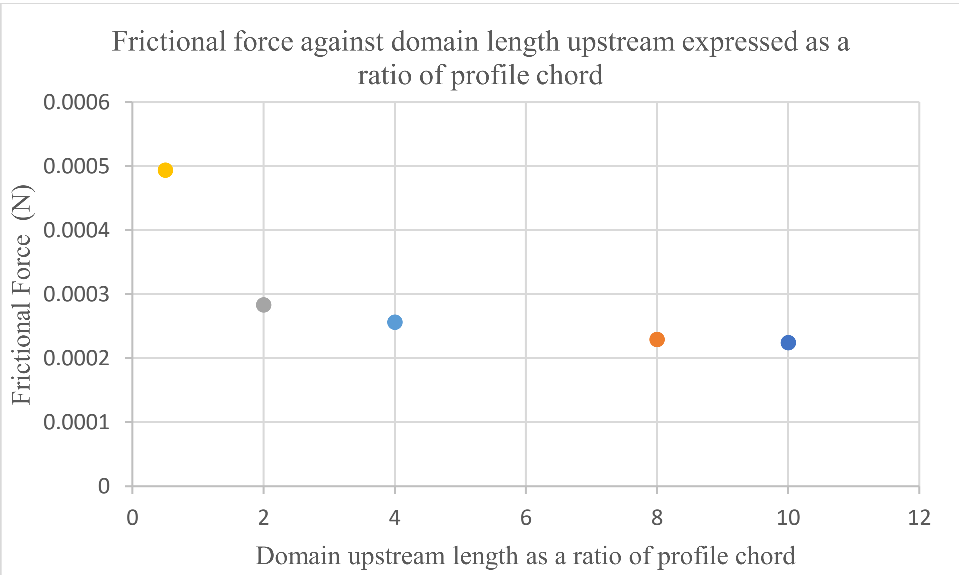

CHART 3, FRICTIONAL FORCE SENSITIVITY OUTPUT FOR DOMAIN LENGTH UPSTREAM

From the output charts over page, it is evident that a convergence is achieved in the Force in X direction long before the final upstream domain length test iteration at 4 times the Frictional force and turbulent kinetic energy dissipation are observed to not to converge at the same point with an increase of 12.5% in frictional force between the converged force X domain (4 times chord length) and the final analysis (10 times the chord length domain) and an increase of 31.3% in turbulent kinetic energy dissipation between those same increments.

Downstream study

Figure 9 and 10 represent the mesh of the smallest and largest domain lengths assessed.

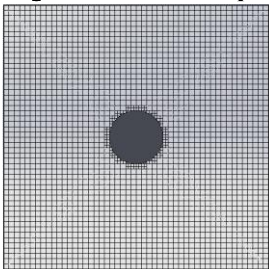

FIGURE 5, EXAMPLE MESH FOR SMALLER DOMAIN DOWNSTREAM

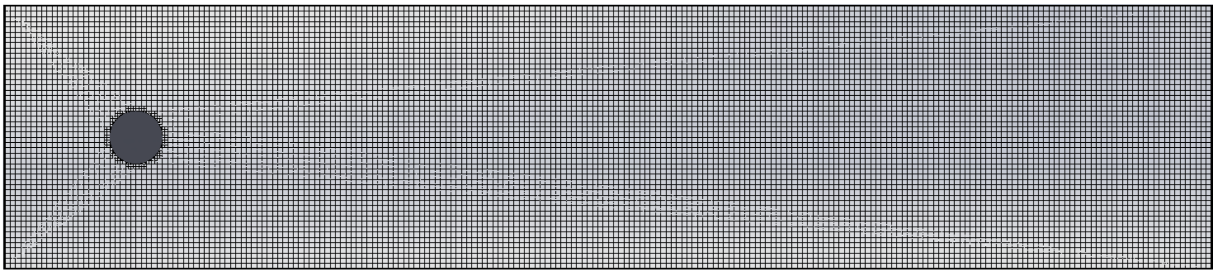

FIGURE 6, EXAMPLE MESH GRID FOR LARGER DOMAIN DOWNSTREAM

Downstream domain length is an important factor in modelling the flow regime downstream of the profile and thus has a large influence on overall fluid dynamic behaviour and thus Cd. Therefore, a larger range of values where selected to experiment to ensure that the correct downstream proximity was established for the flow case.

Results

Figure 11 and 13 report the pressure results for smallest and largest domains, with figure 12 and 14 for the velocity.

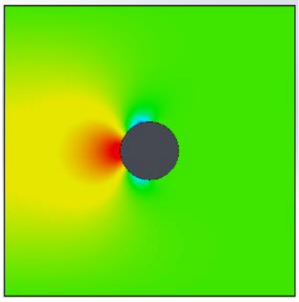

FIGURE 11, OUTPUT FOR PRESSURE PLOT SHORTER DOMAIN EXAMPLE

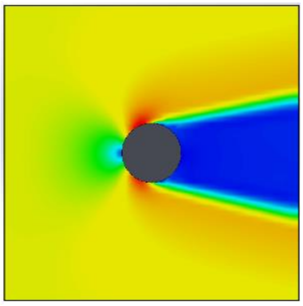

FIGURE 7, OUTPUT FOR VELOCITY SHORTER DOMAIN EXAMPLE

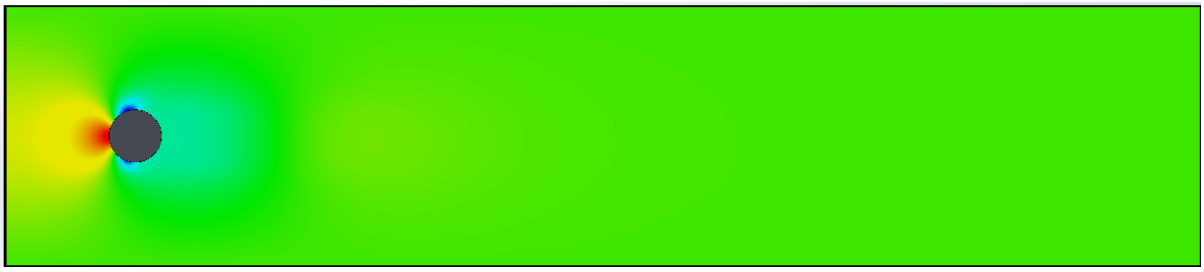

FIGURE 13, OUTPUT FOR PRESSURE LONGER DOMAIN EXAMPLE

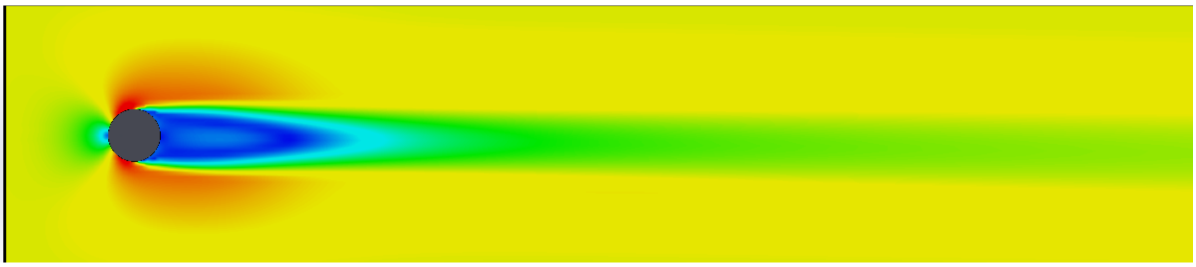

FIGURE 8, OUTPUT FOR VELOCITY LONGER DOMAIN EXAMPLE

Here in the present figures 11-14, a correlation can be observed between the quality and validaity of fluid dynamic behaviour and the domain length, this is confirmed by theoretical and literature expected cases.

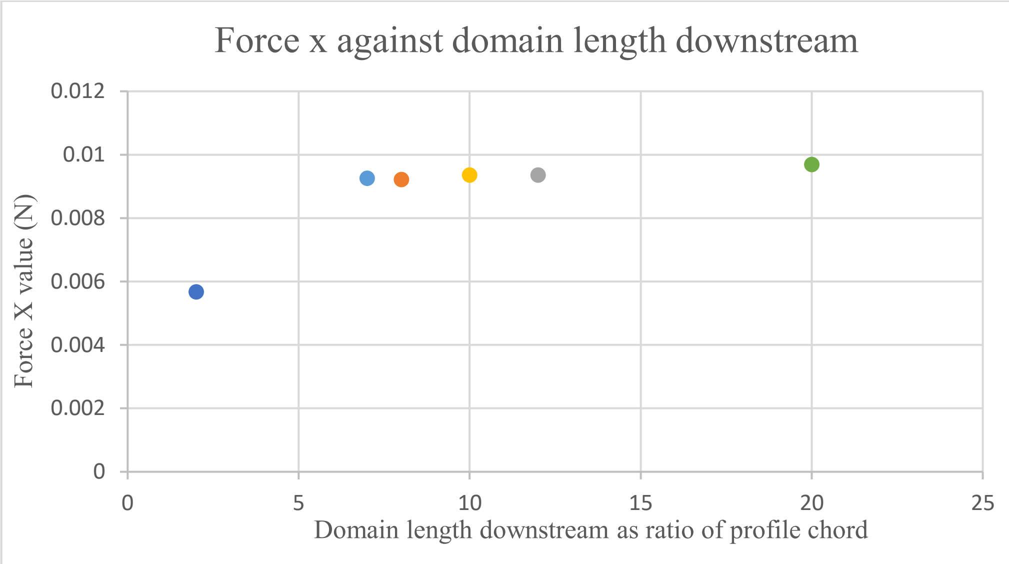

Chart 1, output for Force X sensitivity study for domain length downstream

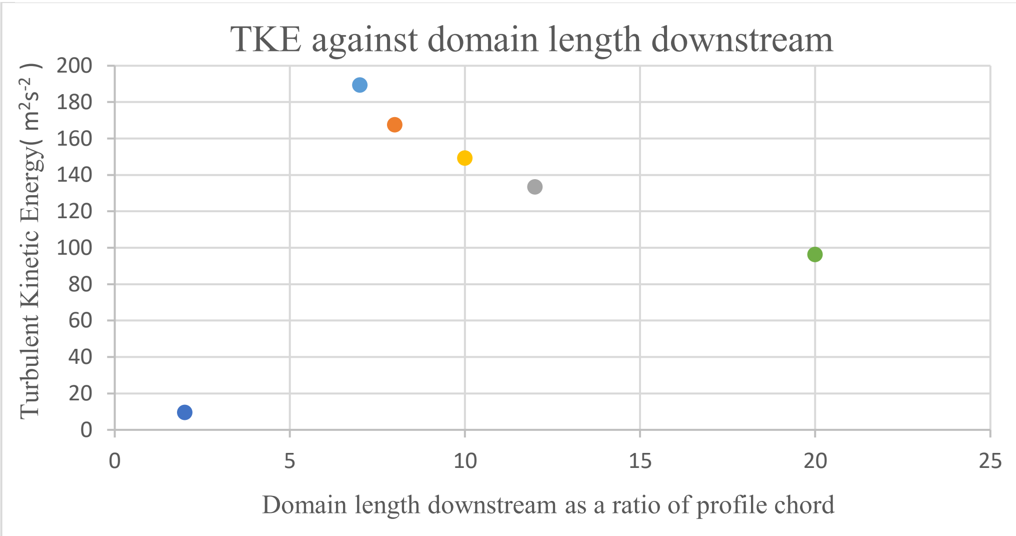

Chart 5, output for TKE sensitivity study for domain length downstream

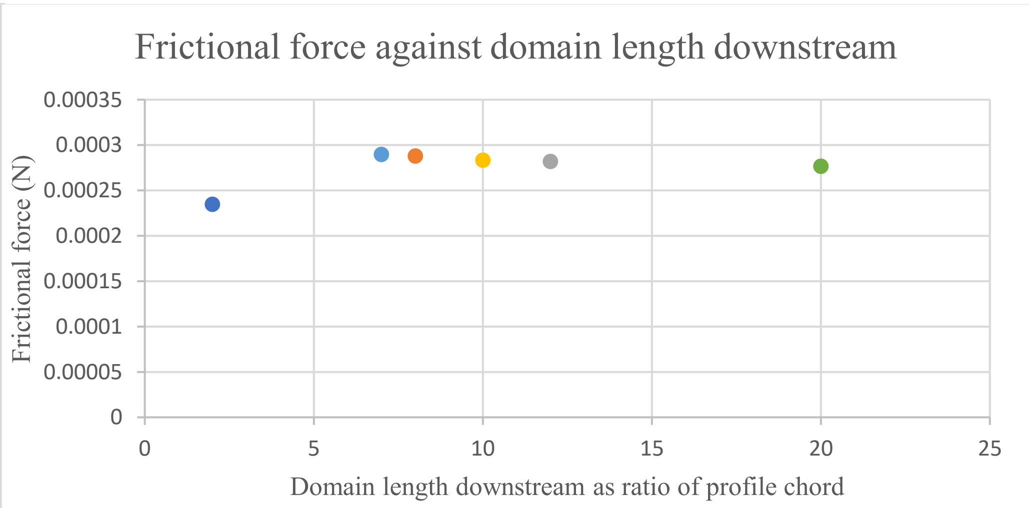

Chart 6, output for frictional force sensitivity study for domain length downstream

As can be observed in the previous charts and plots, improved solution quality was obtained for a specific domain sizing, further this solution quality equalised at a specific point according to domain size against the profile chord, therefore this setting was carried forward to be implemented in the further studies.

Wall gap study

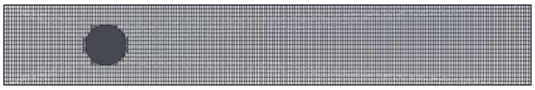

FIGURE 9, EXAMPLE OF COMPUTATIONAL GRID FOR CLOSER DOMAIN CLEARANCE FOR WALL GAP

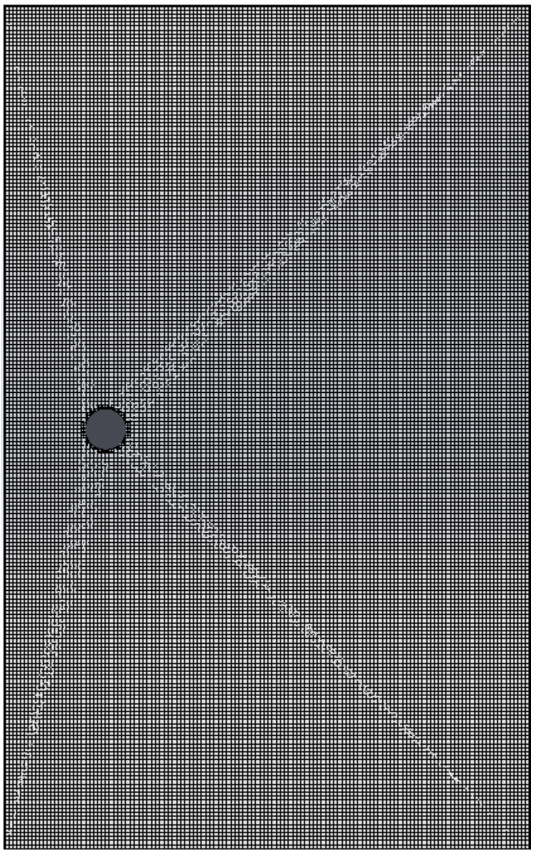

FIGURE 10, EXAMPLE OF COMPUTATIONAL GRID FOR FURTHER CLEARANCE FOR WALL GAP

Figure 15 and 16 show examples of the most narrow and widest domain, as opposed to the upstream and downstream study, this study would investigate the sensitivity of the domain according to the profile thickness and not chord.

RESULTS

Here the results for pressure in figure 17 and 18:

FIGURE 11, PRESSURE PLOT FOR NARROW DOMAIN EXAMPLE

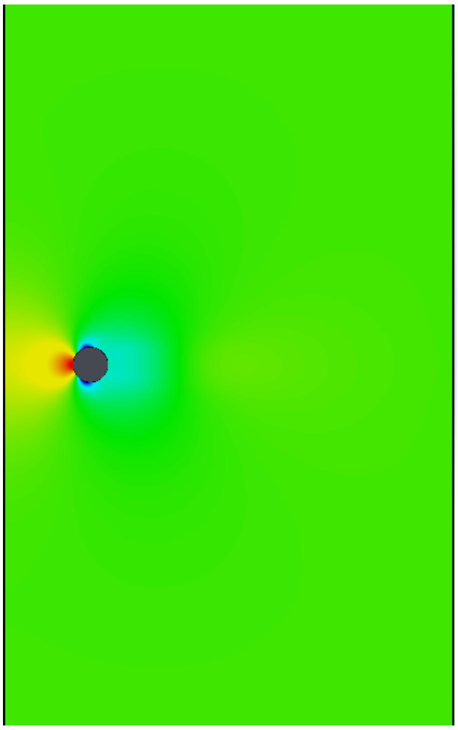

FIGURE 12, PRESSURE PLOT FOR WIDER DOMAIN EXAMPLE

Velocity results are reported in figure 19 and 20:

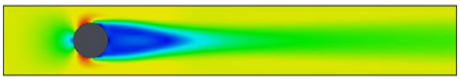

FIGURE 19, VELOCITY PLOT FOR NARROW DOMAIN EXAMPLE

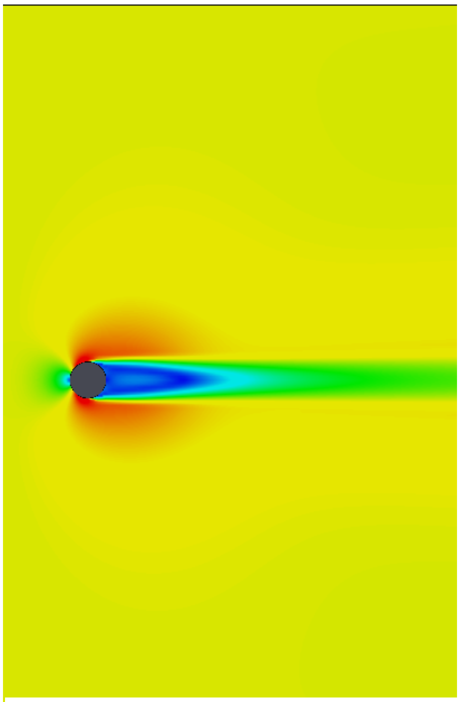

FIGURE 20, VELOCITY PLOT FOR WIDER DOMAIN EXAMPLE

From the qualitative results for the domain fluid flow a conclusion can be made about the sensitivity to wall domain proximity, below in chart 7, 8 and 9 the reported outputs

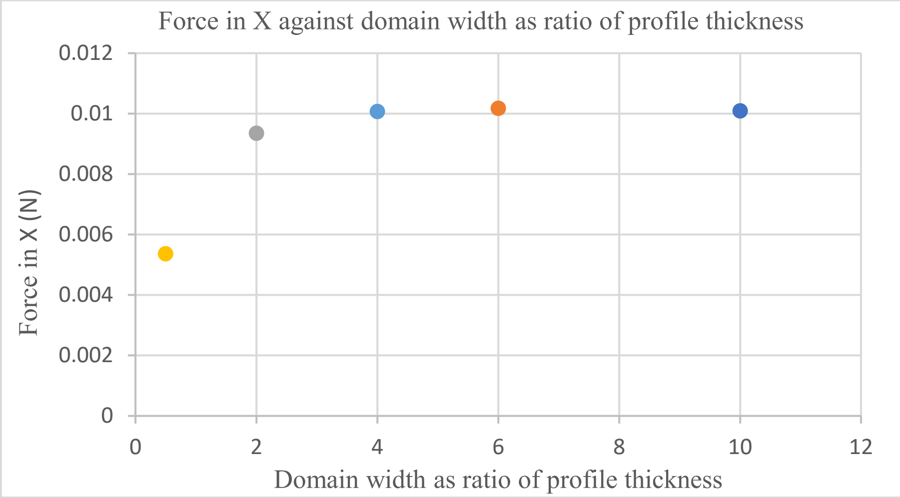

CHART 7, OUTPUT FOR FORCE X FOR DOMAIN SENSITIVITY STUDY

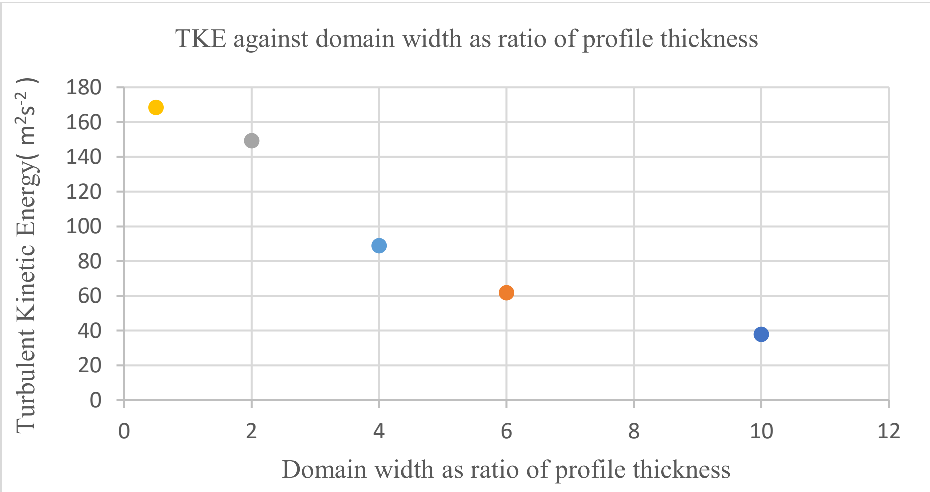

CHART 8, OUTPUT FOR TKE FOR DOMAIN SENSITIVITY STUDY

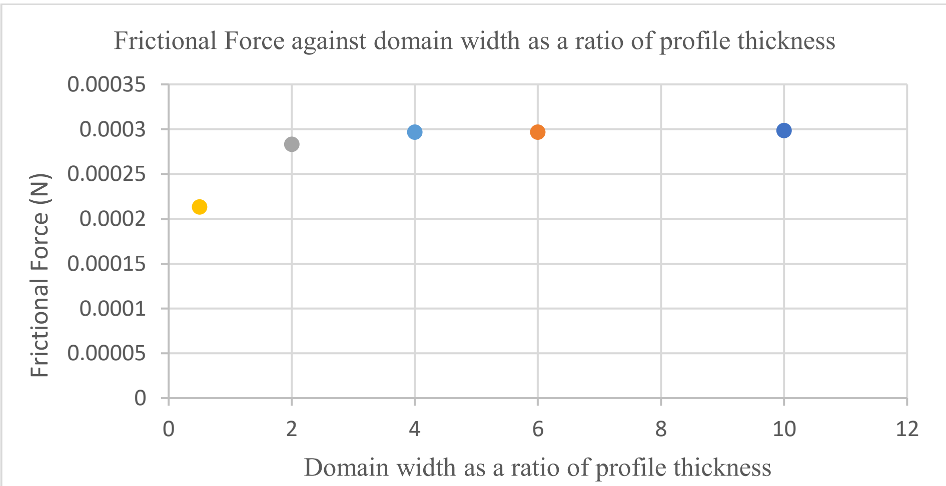

CHART 9, OUTPUT FOR FRICTIONAL FORCE FOR DOMAIN SENSITIVITY STUDY

Again, like the studies for upstream and downstream sensitivity, a specific optimum was obtained between domain sizing and accuracy of the solution. These results are imposed as profile specific rules that can be implemented for domain width and it is expected that these will yield valid results for further investigated profiles.

Conclusion

Overall domain sizing has been achieved for the cylindrical profile and sensitivity for the domain fluid flow accuracy and validity. It is expected that by obtaining settings for the domain sizing for the most complex flow field (cylindrical profile), the same settings will be valid and effective for other profiles that will be tested and validated in the future. Work is still required to improve other characteristic controls for the computational domain, and these are detailed in the further studies.

3.5. Mesh initial sensitivity study

The mesh refers to the discretisation of the fluid domain into small calculatable portions known as cells, in each cell the solver calculations are imposed. The total combination of all the cells is known as the mesh. For CFD applications the mesh is vital to achieving a useful solution and must be defined for the methodology.

The main parameters used to determine mesh quality are:

- Calculation time of solution
- Accuracy of solution
- Iterations required to converge the solution
- Ability of the mesh to capture specific fluid dynamic details

Therefore, the following aspects must be considered to optimise the solution and thus the overall mesh quality:

- Type of cells
- Cell count
- Cell dimensions (X and Y in the case of this 2D case)
- Cell density
- Cell aspect ratio
- Growth rate, the rate at which cells change in dimension from a specified region / cell / object
- Size ratio between one cell and surrounding cells

Meshing is one of the crucial factors in undertaking CFD analysis and highly influences the accuracy, relevancy, and efficiency of a simulation. The goal in undertaking mesh optimisation was to understand the effects of utilising the different mesh controls of solid works and be able to apply a generic rule that would be profile specific and applicable to different shapes in different conditions.

From literature [15] [16] the main discussion on determining an adequate mesh is to control the cells in specific areas of interest for a particular analysis. For most typical CFD analyses for low speed external flows, the areas of particular interest for analysis and design are the boundary layers and downstream fluid behaviour, hence the mesh must be suitably altered in such a way to enable more computational power and resolution is designated to these areas, through mesh controls.

3.5.1. Initial Mesh

A basic cell density analysis was undertaken to try to ascertain the influence of mesh density on the SolidWorks Flow Simulation solver and how the expected behaviours of fluid dynamics would be captured using mesh discretization of the problem.

Procedure

Control of mesh size specific to the domain parameters was undertaken to ensure that cell ratios could be defined in the X and Y directions for the mesh, and further a study with fixed cell aspect ratio were undertaken to investigate the effects of X density, Y density and general density, to fully characterise the effect of cell density. This was undertaken to find a relation for cells specific to profile geometry. The same parameters as used previously were utilised to characterise the effectiveness and validity of each simulation against the literature findings and accepted results for the specific shape.

Figure 21 and 22 detail the density study for X direction density from lowest number in X to highest, figure 23 and 24 detail the meshes for the density study in the Y direction, figure 25 and 26 provide the overview of general mesh density. These studies where undertaken to assess both mesh density and cell aspect ratio influence on the output accuracy and validity.

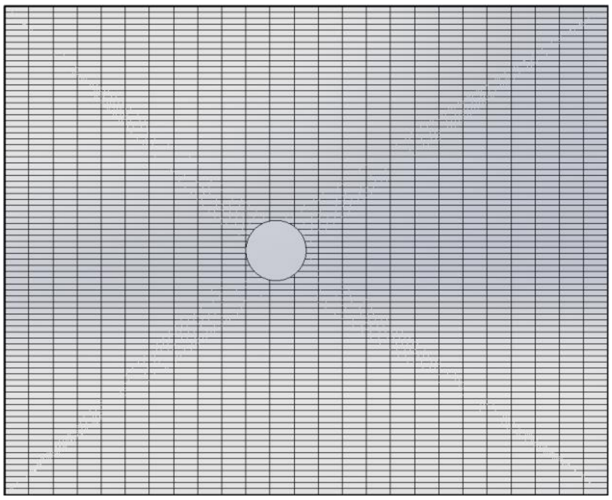

FIGURE 21, X DIRECTION LOW DENSITY GRID

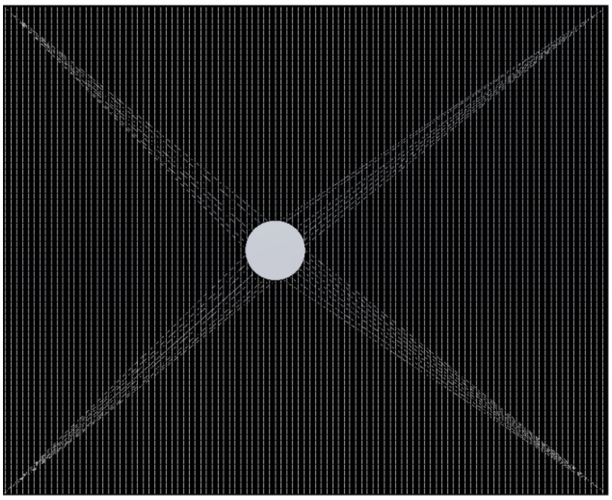

FIGURE 22, X DIRECTION HIGH DENSITY GRID

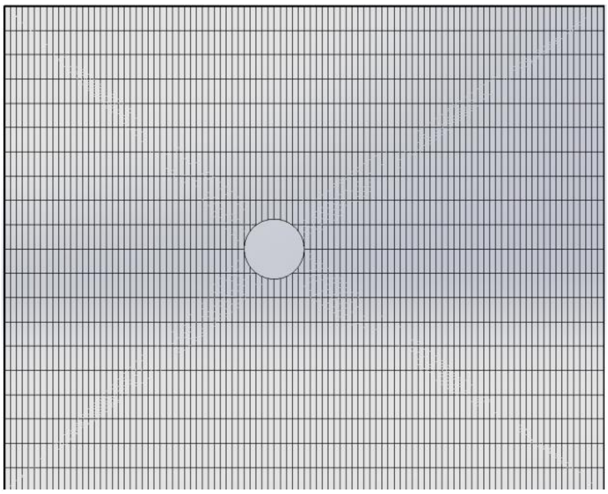

FIGURE 23, Y DIRECTION LOW DENSITY GRID

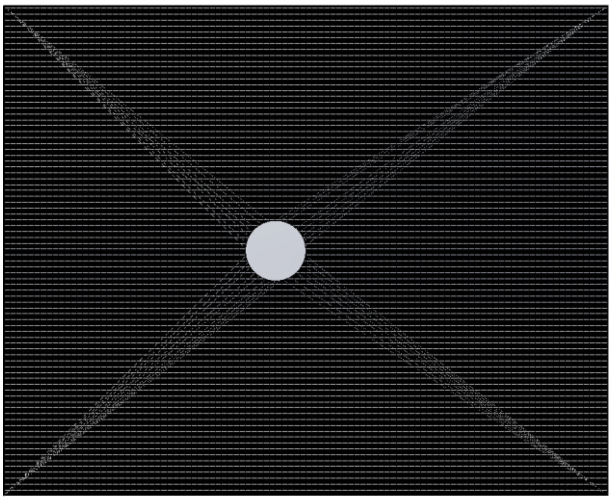

FIGURE 24, Y DIRECTION HIGH DENSITY GRID

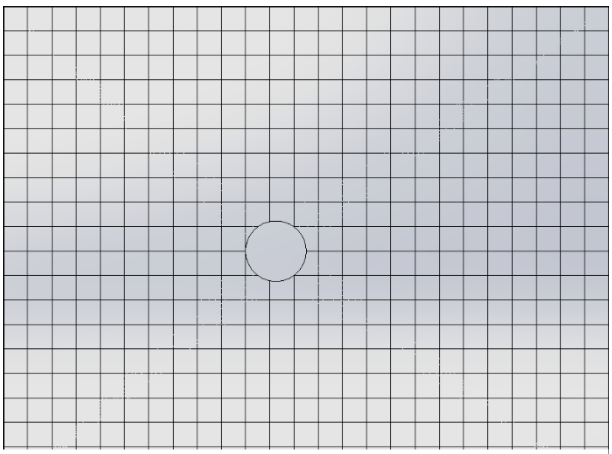

FIGURE 25, GENERAL LOW-DENSITY STUDY GRID

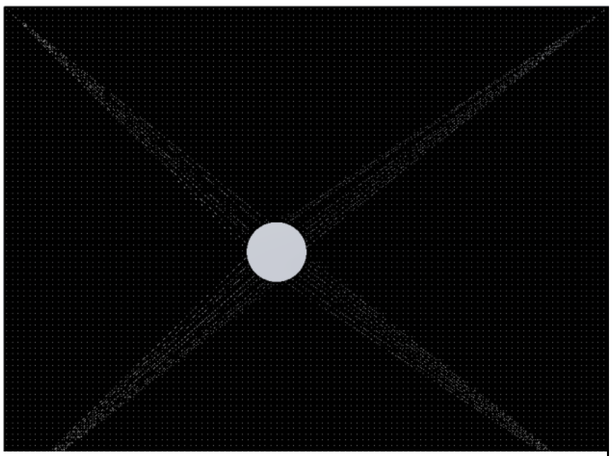

FIGURE 26, GENERAL HIGH-DENSITY STUDY GRID

Results

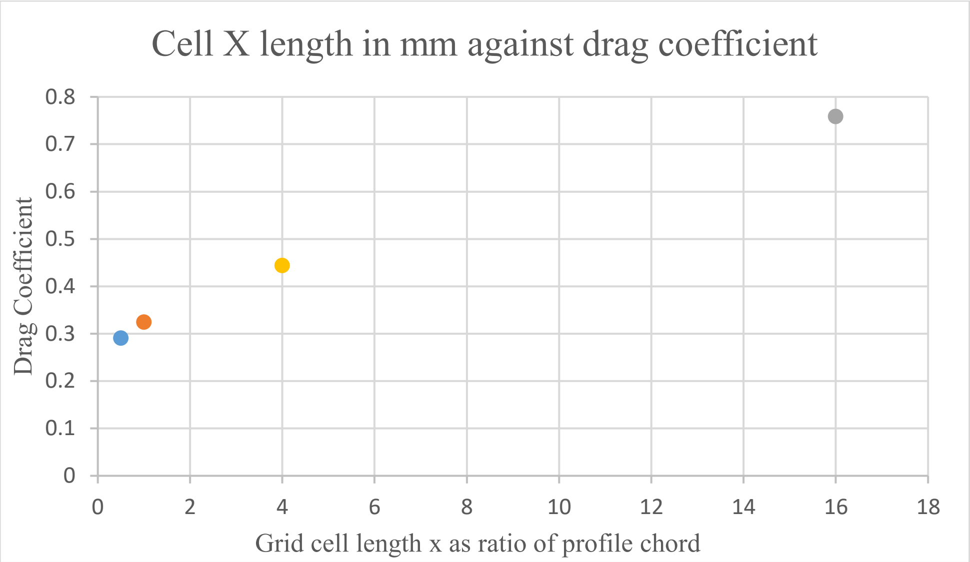

CHART 10, REPORTED OUTPUT FOR CELL LENGTH IN X DIRECTION FOR DRAG COEFFICIENT

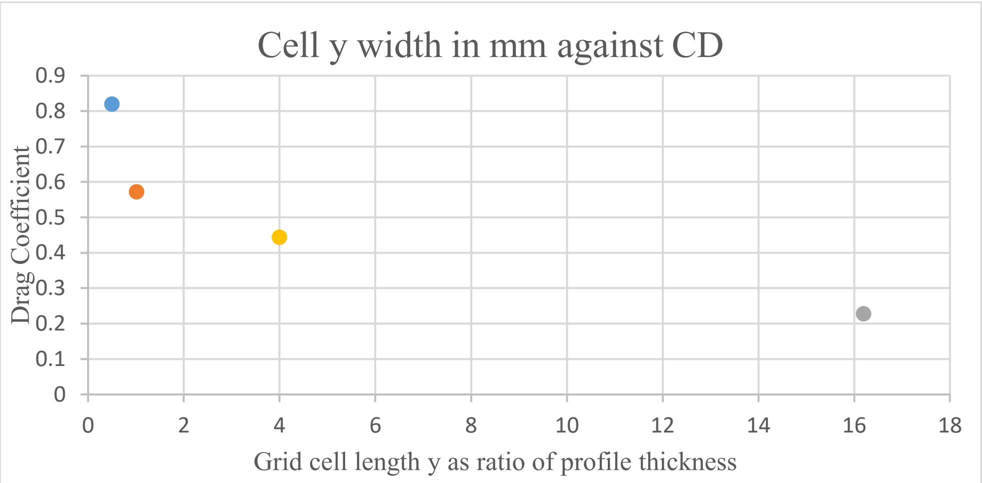

CHART 11, REPORTED OUTPUT FOR CELL LENGTH IN Y DIRECTION FOR DRAG COEFFICIENT

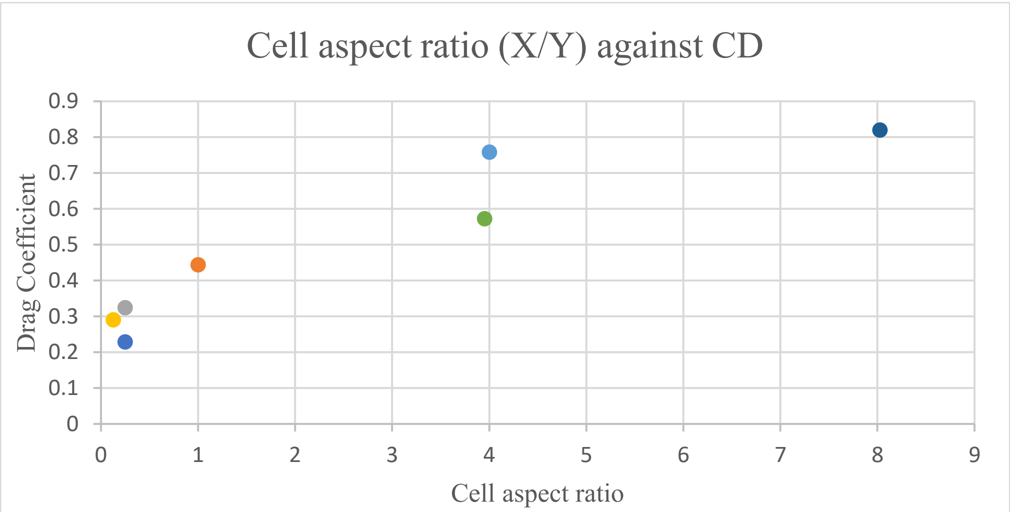

CHART 12, REPORTED OUTPUT FOR GENERAL CELL ASPECT RATIO

3.5.2. Conclusion from initial study

To conclude the findings from the initial studies carried out, correlation was identified between the quality of the simulation and specific settings for both the domain and mesh controls. These settings where applied through specific geometrical characterisations to allow the method to be applied to different geometries.

When considering the overall quality of the solution, a considerable amount of improvement is required, for the final achieved domain and mesh combination, several issues where found within the solution, namely:

- Stability of the steady state solution
- Percentage error for the Cd value

Therefore, it is necessary to further the solution by applying further controls to the method prior to investigation using other profiles.

3.6. Optimisation

To further improve the control of the method and improve the accuracy, it was determined to undertake optimisation activities. This involved more advanced mesh and calculation controls to obtain a well-defined system that was to be developed for generic profile analysis.

3.6.1. Y+ studies

One important aspect of obtaining valid and accurate CFD solutions is considering the ability of the computational domain to replicate the physics of the fluid adequately. Specifically, for the application in question, a sizeable portion of the flow field behaviour is determined by the ability to adequately model the boundary layer around the profile.

By effectively meshing around the boundary layer it is therefore possible to propagate fluid dynamic parameters and values outwards into the remainder of the computational domain, therefore attention must be paid to adequately characterise the mesh at the boundary layer.

One commonly utilised method for boundary layer characterisation is known as the y+ method, this method defines the first cell height in proportion to characteristics of the fluid flow and the body which is being analysed. The following generic equation for y+ is as follows [19]:

$$y^+ = \frac{y u_\tau}{v}$$

Where:

y is the distance of the point in space to the subsequent wall

u_τ is the frictional velocity

v is the kinematic viscosity

This can be utilised to determine the first cell height from the body by the following passage [20]:

Calculation of the Reynolds number [20], which describes the fluid flow:

$$Re = \frac{\rho * u * L}{v}$$

Obtaining the Blasius boundary layer [8] which determines the generic skin friction coefficient:

$$Bcf = \frac{1}{\sqrt{Re}}$$

This can be subsequently used incorporated into determining shear stress from the wall

$$\tau_w = \mu \frac{du}{dy}$$

$$\mu f = \sqrt{\frac{\tau_w}{\rho}}$$

We then finally obtain the height of the first cell against the wall of the profile:

$$y_{height} = \frac{v * y_+}{\rho * \mu f}$$

Implementation of the y+ value in SolidWorks Flow simulation cannot directly defined due to the mesh settings being characterised by cell refinement ratios and not by absolute reference values. Therefore, it was necessary to define first the calculated 'ideal' y+ value and calculate the achievable y+ value based on cell refinement calculated by the meshing program inbuilt to the software.

Meshing was carried out using the base mesh density settings, refinements where controlled in the program to prevent the automated meshes that would modify the surrounding mesh (out with the desired modifications to the y+ cells) and thus affect the Figures 27 through 29 detail example variations in the y+ for the initial boundary cells, growth rates.

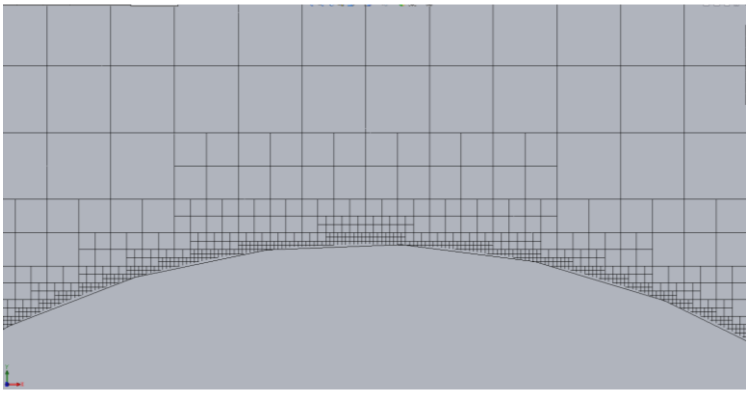

FIGURE 27, EXAMPLE OF SMALLER Y+ MESH

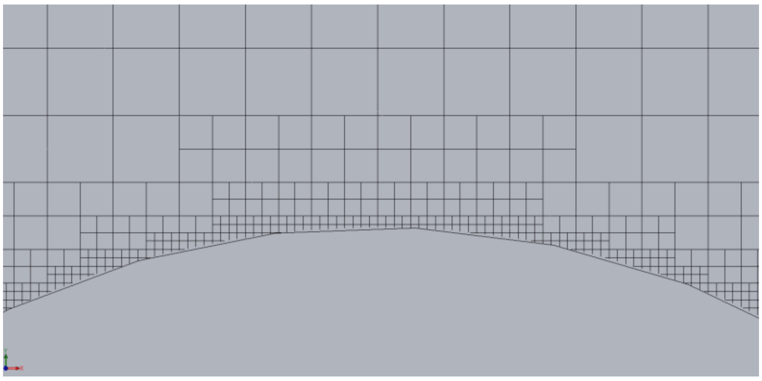

FIGURE 28, EXAMPLE OF INTERMEDIATE Y+ MESH

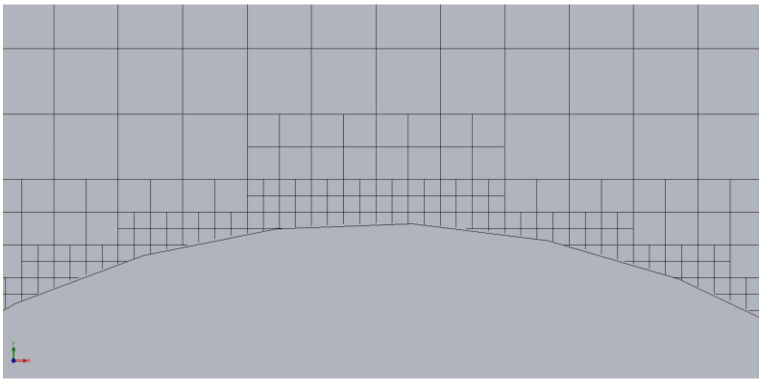

FIGURE 29, EXAMPLE OF LARGER Y+ MESH

The variation in the y+ values was determined by literature sources which provided estimated values of best practice for the given type of application. It is expected that by capturing sufficiently the boundary layer for the case using the bluffer body, the y+ value can be translated also into the more aerodynamically efficient profiles. This hypothesis will be proven in the validation and verification section.

Results

Below in chart 13 is the reported sensitivity study for the range of y+ values utilised according to the percentage error.

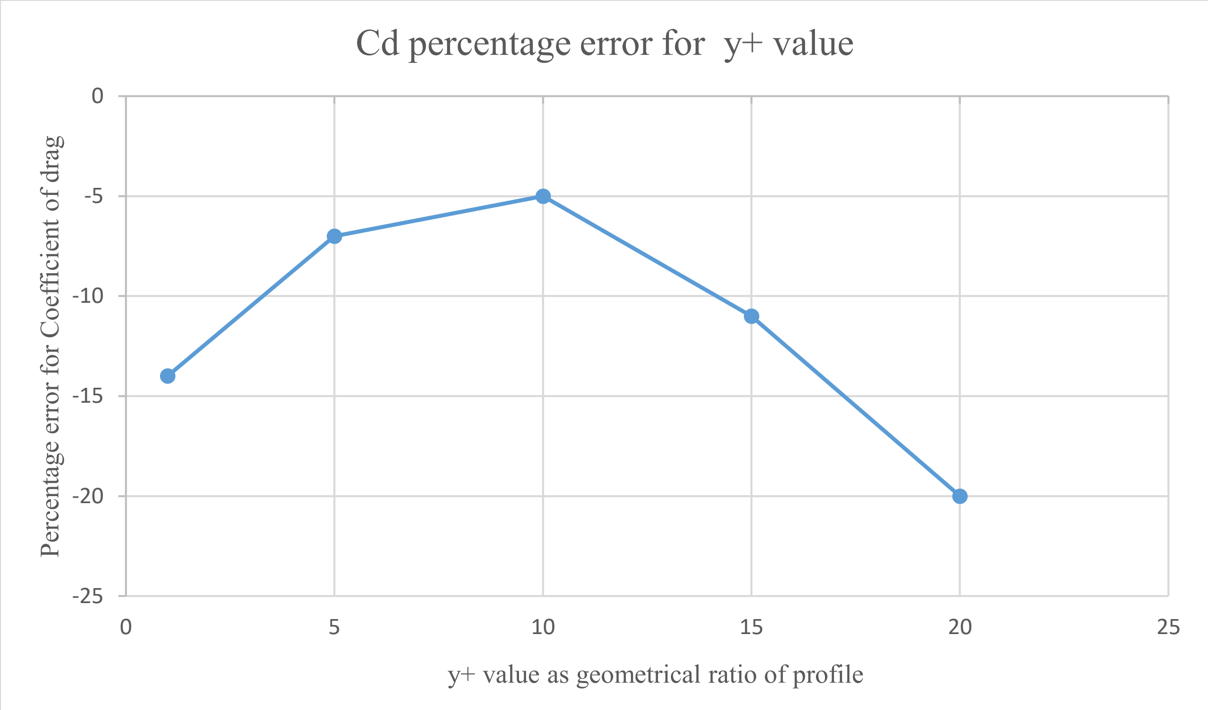

CHART 13, Y+ SENSITIVITY FOR DRAG COEFFICIENT ERROR

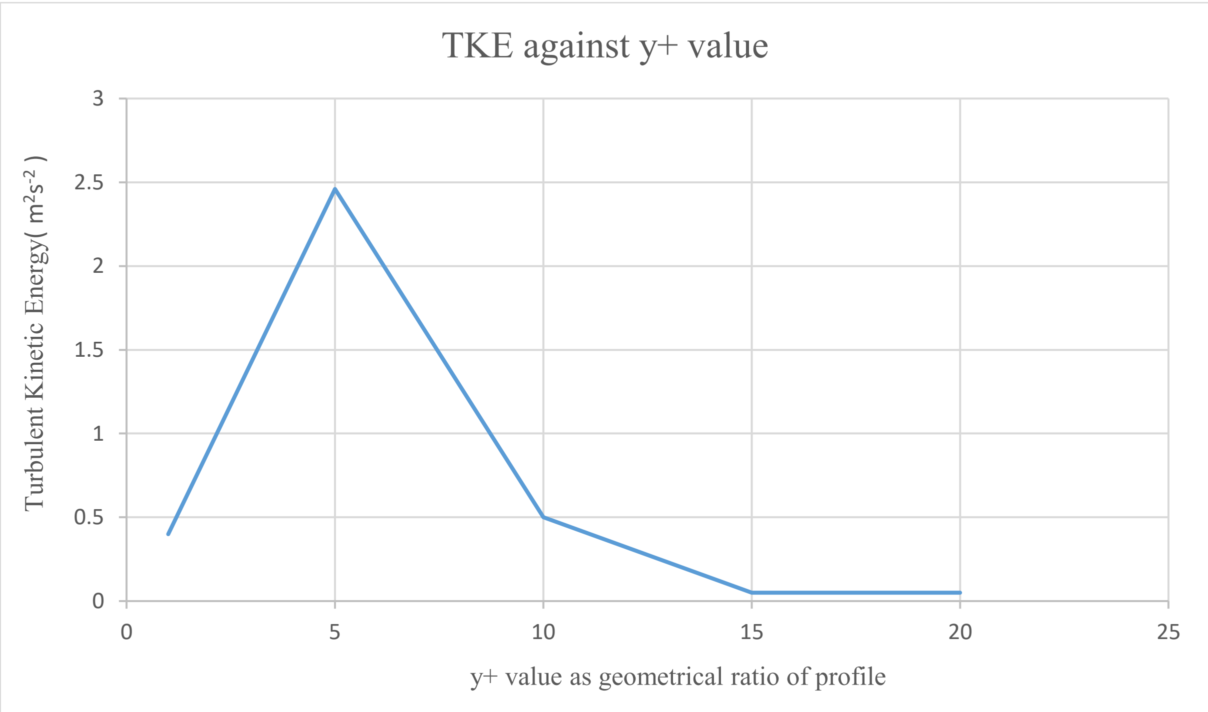

CHART 14, Y+ SENSITIVITY FOR TKE VALUE

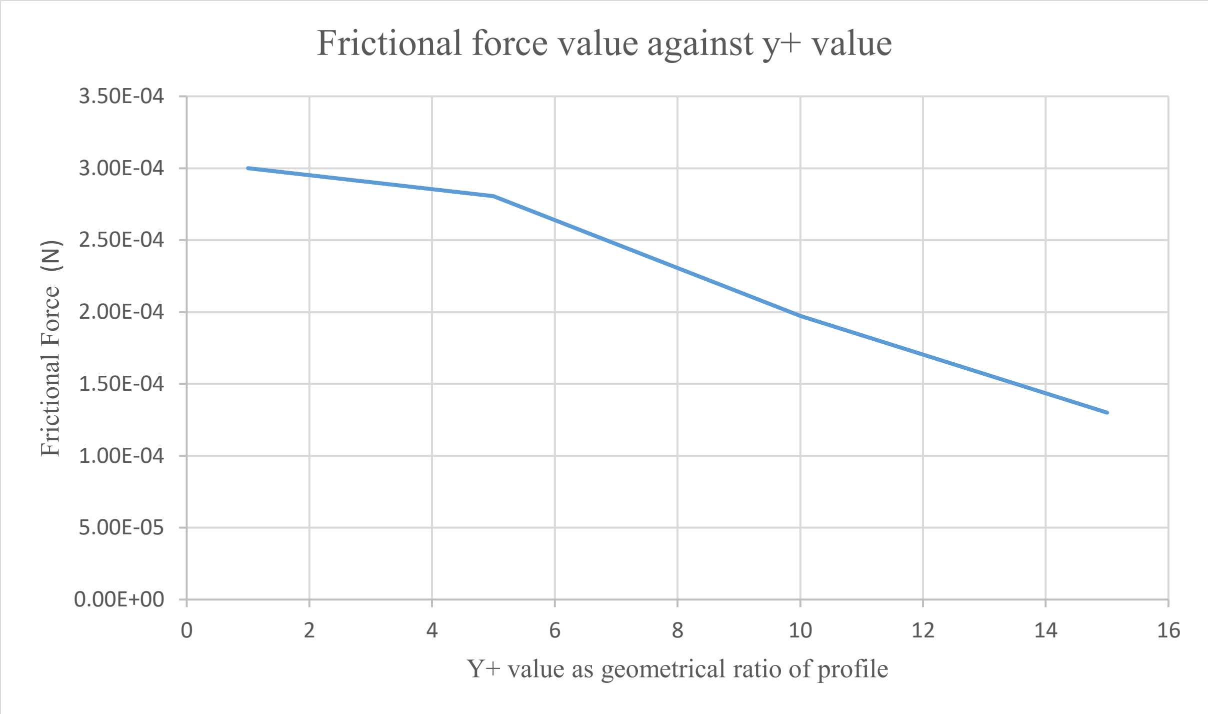

CHART 15, Y+ SENSITIVITY FOR FRICTIONAL FORCE

It is interesting to note that for the cylindrical profile there is an optimum in the y+ value for the drag coefficient. The following observations and hypotheses are made:

- The low y+ values cause the solution are significantly lower than the accepted Cd value:
    - Over estimation of the boundary layer energy through the frictional forces at the surface, thus keeping fluid flow attached to the cylindrical profile for longer than in physical reality, this can be seen in the underestimation in the TKE values
    - Lose of stability and convergence in the residual values for the steady state solution due to increased cell density around key areas
    - Induced drag underestimation again from the TKE values
    - Cell gradients are too drastic between the initial y+ cell and the generic grid providing false values for fluid dynamic properties
- The high y+ value results also underestimate the drag coefficient:
    - Inability to suitably capture the frictional forces present and thus drag force
    - Lower induced drag from a smaller modelled boundary layer
    - Lower turbulence effects due to reduction in fluid flow volume that has diverged from the atmospheric path

The previous results become more obvious when observing the absolute values and indicated behaviour of each case (Low Cd and High Cd) against the optimal y+ value.

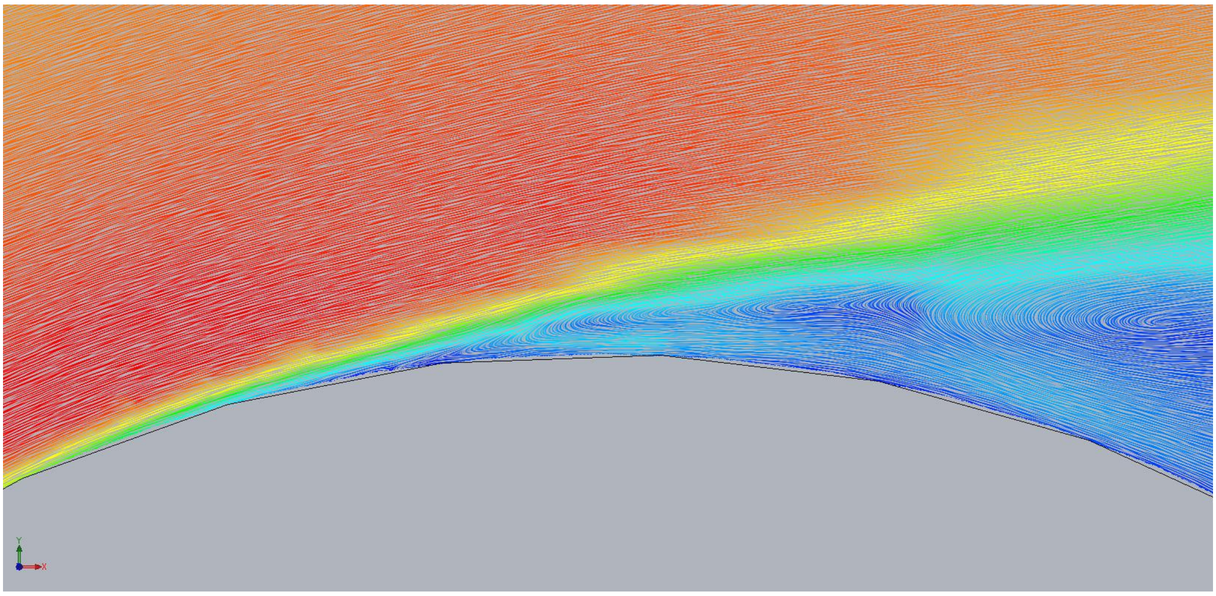

FIGURE 30, THE REPORTED VELOCITY PROFILE FOR A SMALL Y+ CELL SIZE

Here we can see the influence that a smaller y+ value has on the boundary layer present in the solution upstream from the widest part of the profile. A thick boundary layer is present across the solution which leads to a false separation point and incorrect values for both the Cd and frictional force across the profile. The solution appears to lose stability / resolution around part of the wake, leading to suggest that work should be undertaken to improve the flow field refinements.

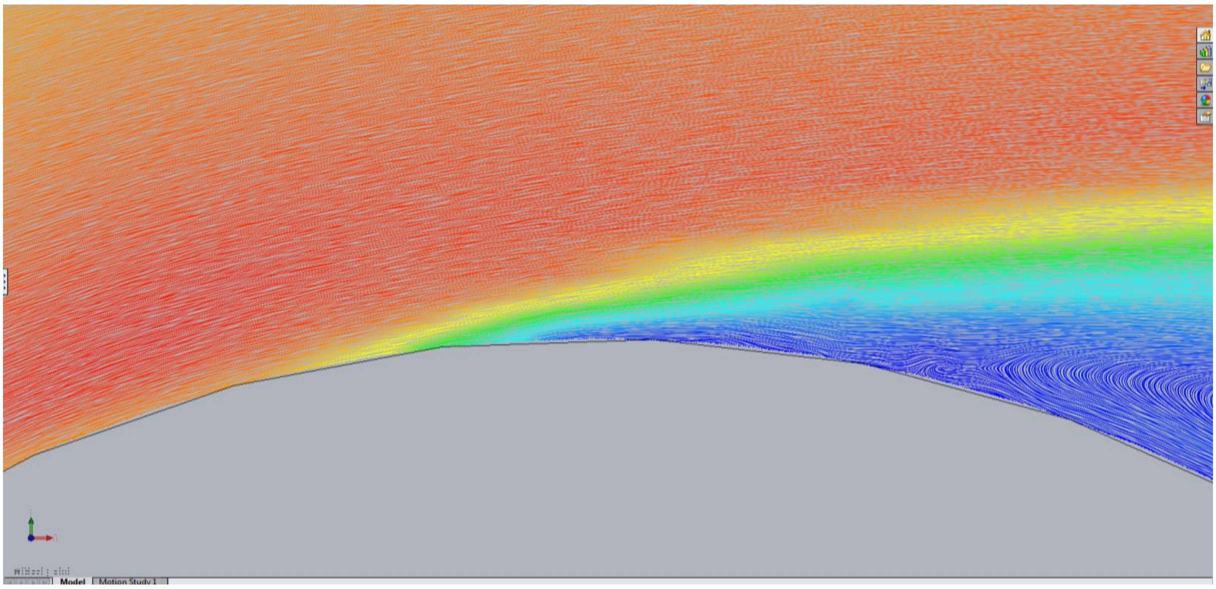

FIGURE 31, THE REPORT VELOCITY PROFILE FOR THE OPTIMUM Y+ CELL SIZE

Comparing the separation point and the extent of the wake for figure 31 compared to figure 32 we can see that a thicker boundary layer is present within the prior solution, with improved stability observed across the steady state solution leading to suggest that growth rate is important when considering the stability of the particular solution

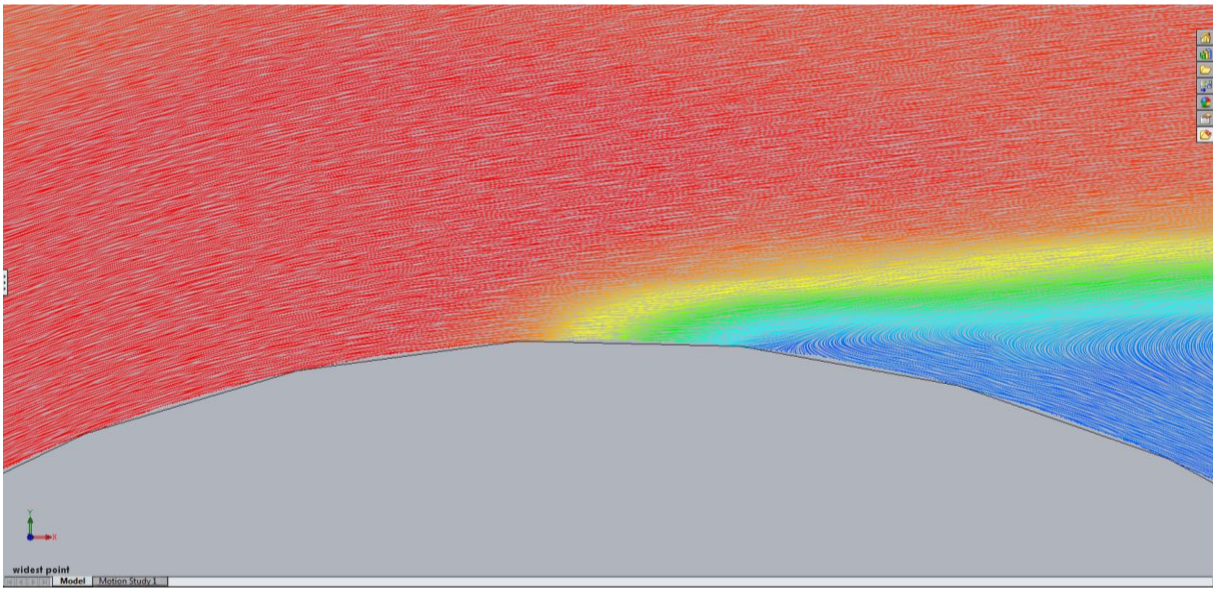

FIGURE 32, THE REPORTED VELOCITY PROFILE FOR A LARGER Y+ CELL SIZE

For the larger y+ value, it is visible that the boundary layer is not sufficiently captured leading to a false separation point and thus inaccurate modelling of the frictional layer along with the downstream wake, leading to a lower Cd value. The issue of stability however in the beginning of the wake appears to improve across this solution, leading to the hypotheses that a smaller y+ value increases instability within the solution and that more attention should be paid to the inflation rate between subsequent cells within the mesh.

For the specific profile an acceptable level of accuracy is determined for evaluating the coefficient of drag, however it remains to be observed if it is possible to apply these results for other expected profiles. Furthermore, the results of the simulation may not accurately capture the full details of the flow field and therefore more work is required to be able to fully resolve the solution to be rendered useful.

3.6.2. Meshing controls

The settings detailed are imposed through SolidWorks Flow Simulation and modified according to the specific cell / mesh settings, it was necessary to measure and quantify the cell settings as SolidWorks Flow Simulation generally works through qualitative numerical inputs rather than quantitative that is common in most CFD software.

This meant that specific 'target values' where defined according to literature, calculation and experimentation and implemented by assessing to what extent each mesh control would modify the computational geometry in SolidWorks Flow Simulation. From the specific findings it was possible to understand the influence that each control has on a cell and overall mesh and specific profile parameters could be input that would yield controllable and variable parameters within the meshing studies.

4. Application and validation of developed method

To investigate the utility of the method, it was necessary to assess the ability of the generic protocol that was developed using the cylinder profile to form appropriate meshes and computational solutions.

- Coefficient of drag
- Turbulent Kinetic Energy
- Frictional Force

Two profiles where selected to evaluate the method effectiveness and to investigate if further development is required to ensure that generic profiles can be estimated. The two profiles where selected on the following basis:

- The profiles could be accurately and effectively modelled in CAD to ensure transfer into the software for meshing and simulating
- The profiles replicated similar shapes found in cycling product design and are comparable in terms of aerodynamic behaviour to those found in industry
- The profiles had well established experimental values and phenomena that can be compared to the methodology developed
- These profiles are expected to display important characteristics for bicycle frame development that would be important to model:
    - Boundary layer attachment / separation across profile
    - Upstream pressure values
    - Downstream pressure values
    - Turbulence generation / reduction

Under these conditions, the following two profiles where selected:

- NACA 0024 profile figure 33
    - Commonly used in the cycling industry
    - Similar ratios for UCI ruling for bike tubing profiles
    - Well established and researched
    - An example of a well-designed air foil for a cycling application
    - Chopped adaptations of this profile are frequently utilised in the cycling industry

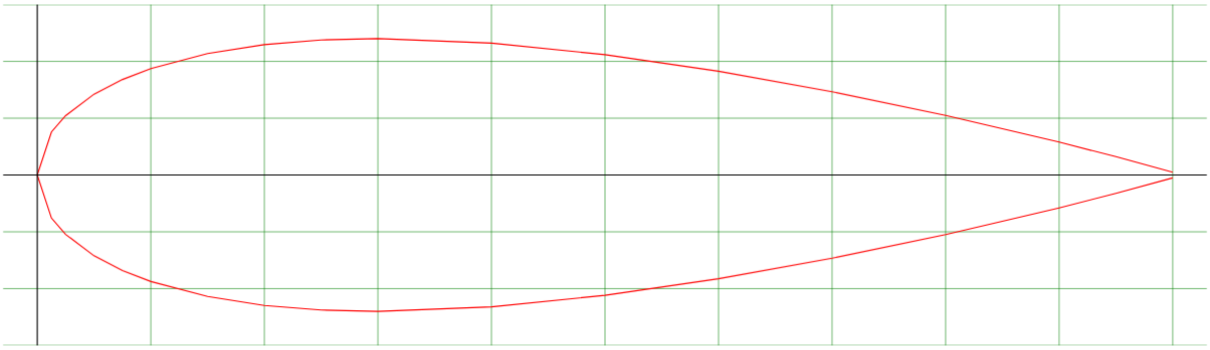

FIGURE 33, THE NACA 0024 PROFILE ADOPTED FOR THE METHODOLOGY EVALUATION

- Eppler 863 figure 34
    - Wider profile compared to the NACA0024
    - Example of an air foil with less suitability for cycling airflow cases
    - Expected separation and boundary layer specific behaviour
    - Used to assess if the method would be able to effectively predict a poorer designed air foil and if it could be used according to the research criteria in the future for product development

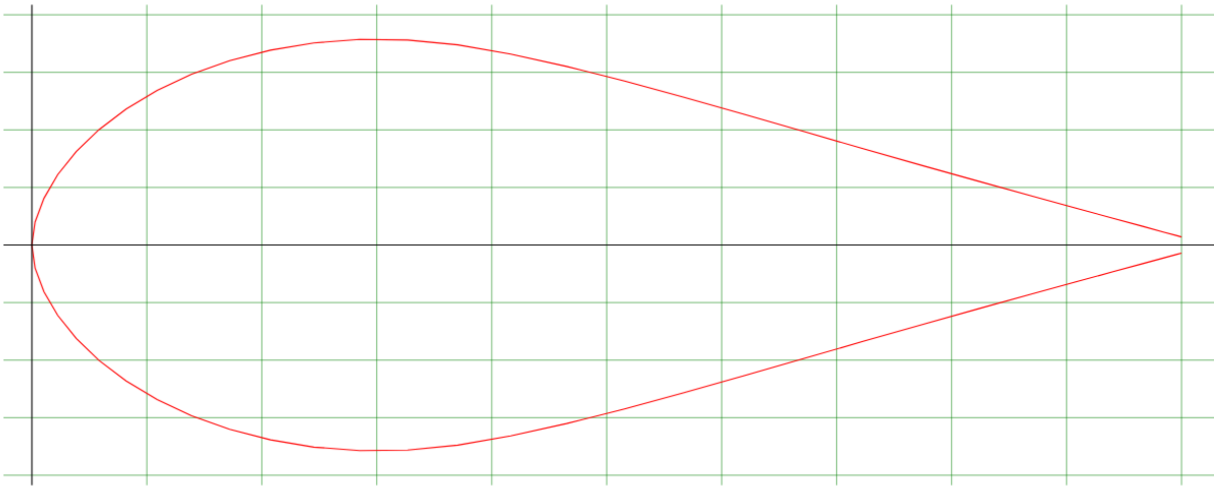

FIGURE 34, THE WIDER EPPLER863 PROFILE ADOPTED FOR THE METHODOLOGY EVALUATION

4.1. Assessment of methodology

The two profiles followed the same methodology as defined in the development activities with the cylindrical profile. Therefore, the main purpose of this section of the report is to detail the effectiveness and validity of such a methodology and any refinements that are required.

4.1.1. CAD import

The import for CAD was different compared to the cylindrical profile, this is due to the construction of mathematically defined profiles for both the NACA and Eppler profiles.

Therefore, rather than creating basic extrusions, the air foils are shaped in SolidWorks CAD program utilising the industrial standard coordinates for each of the respective profiles. By first creating the line profile of each shape, it was then possible to create a solid body and extrude to the given thickness, to be used for the computational grid.

4.1.2. Domain Generation

The domain was generated according to the sensitivity studies undertaken for the cylindrical profile; this means that the domain was defined generically from points on the profile as ratios against:

- The chord of the profile
- Thickness of the profile
- The characteristic length of the profile, if different to the cord

4.1.3. Mesh Generation

The meshes where generated according to the defined characteristics from the cylindrical study, the meshes where therefore generated according to:

- The characteristic length of the profile
- Boundary conditions
- The angle of separation of the profile
- Reynolds number associated with the fluid flow

Imposing the previously determined mesh conditions for the profile specific methodology, the following meshes where created in figure 35 and 36 for the NACA 0024 and Eppler 863 profiles respectively:

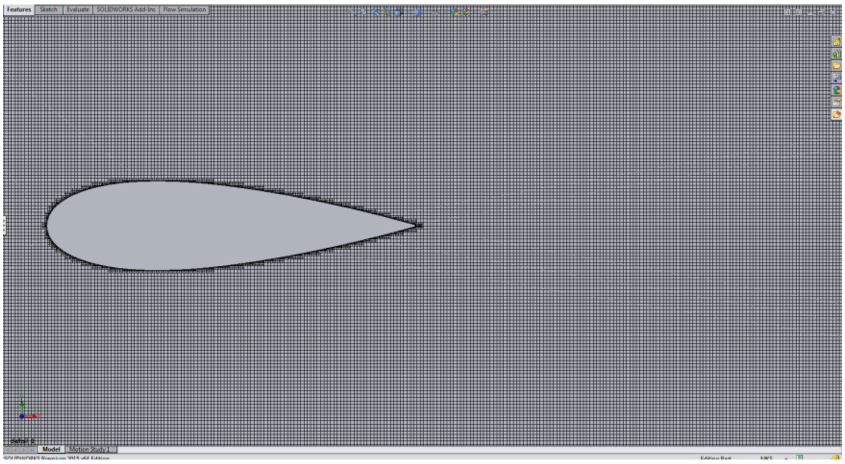

FIGURE 35, THE OVERALL COMPUTATIONAL GRID FOR THE NACA0024 GENERATED BY THE METHODOLOGY

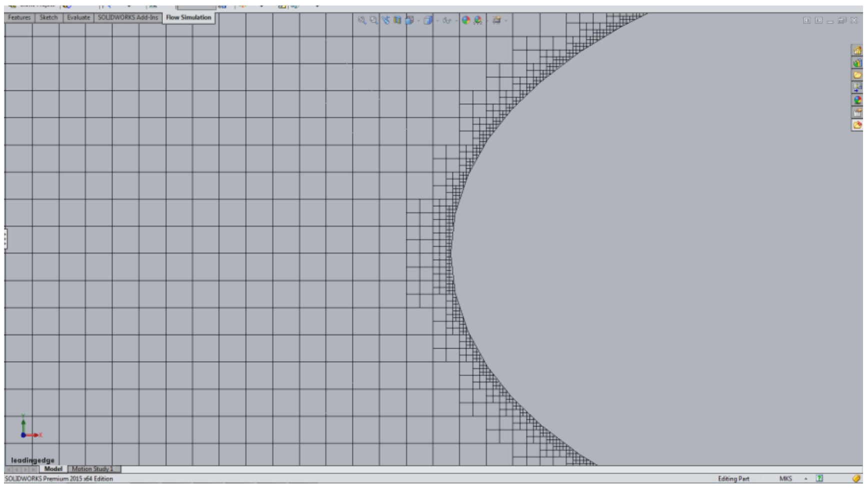

FIGURE 36, DETAILED VIEW OF THE COMPUTATIONAL GRID FOR THE NACA0024 GENERATED BY THE METHODOLOGY

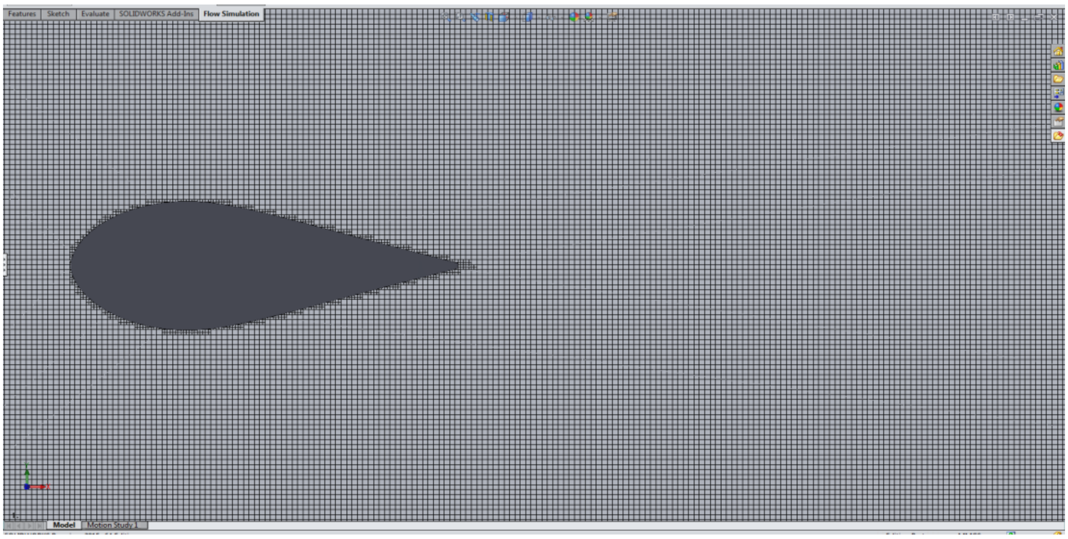

FIGURE 37, THE OVERALL COMPUTATIONAL GRID FOR THE EPPLER863 GENERATED BY THE METHODOLOGY

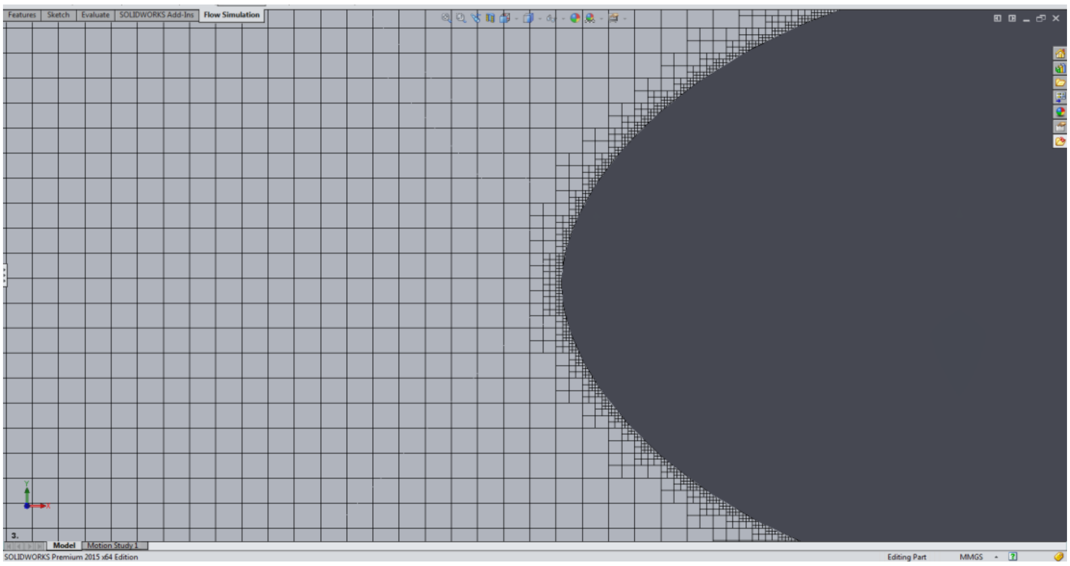

FIGURE 38, DETAILED VIEW OF THE COMPUTATIONAL GRID FOR THE EPPLER863 GENERATED BY THE METHODOLOGY

4.2. Initial study

Testing was carried out using the standard established test protocol for the profiles previously established in the literature review. The two profile tests were carried out using basic standard test set-ups meaning:

- Use of the nominal boundary conditions for the air flow properties
- Standard alignment, no yaw angle imposed

This was done to assess the methods ability in a basic way with the scope of future tests to be undertaken to assess the full capability of the mesh for assessing cycling product performance and analysis.

4.3. Results

The following section reports the results for each of the two profiles tested, with the scope to assess the suitability of the methodology.

4.3.1. NACA profile

The NACA profile was simulated using the defined standard boundary conditions, the following conclusion is made about the profile from the result:

- A stable and converged solution was obtained
- A valid solution was obtained in terms of aerodynamic behaviour
- The error in the coefficient of drag was +12% from the accepted value

Figure 40 reports the result for the velocity profile:

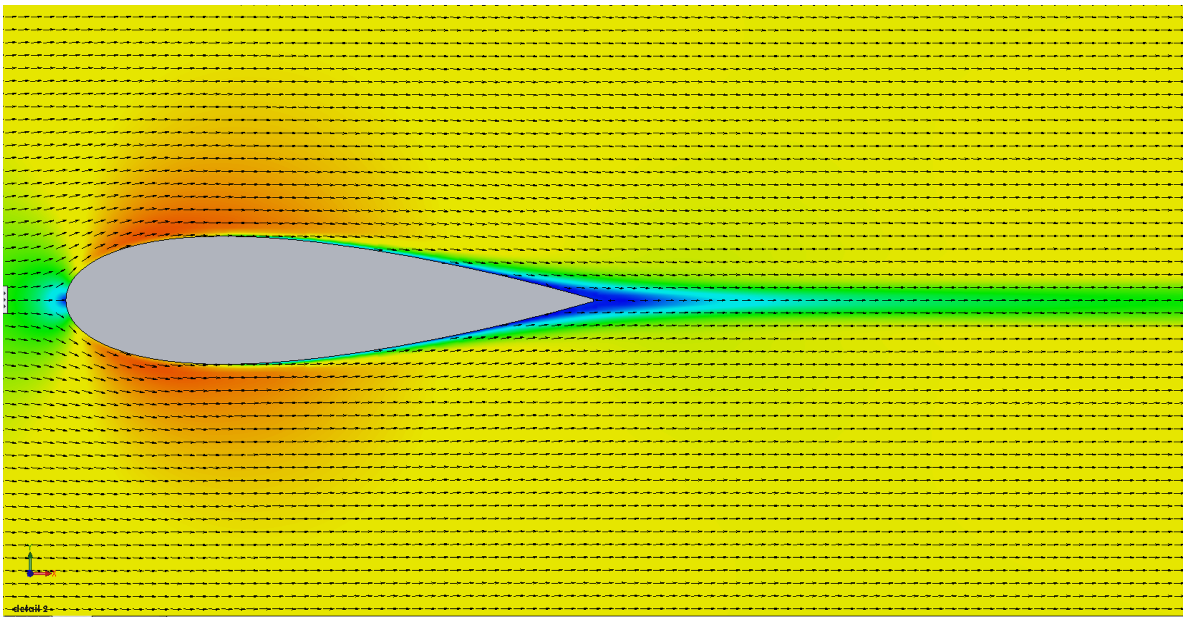

FIGURE 40, OUTPUT FOR THE VELOCITY PROFILE OF THE NACA0024 MESHED BY THE DEVELOPED METHODOLOGY

Figure 41 details the solution around the tail of the profile, a crucial area to evaluate for the air foil performance and thus solution accuracy :

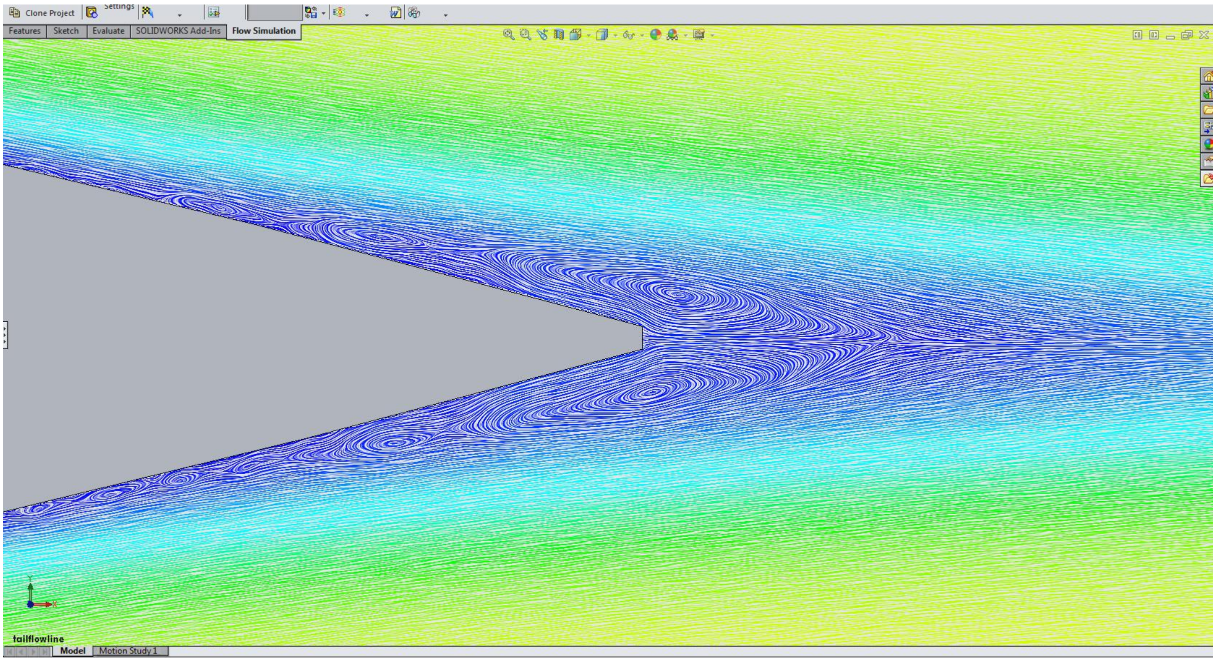

FIGURE 41, THE OUTPUT FOR THE PROFILE TAIL COMPLETE WITH FLOW SEPERATION AND TURBULENCE GENERATION

Whilst figure 42 reports the result for the overall domain

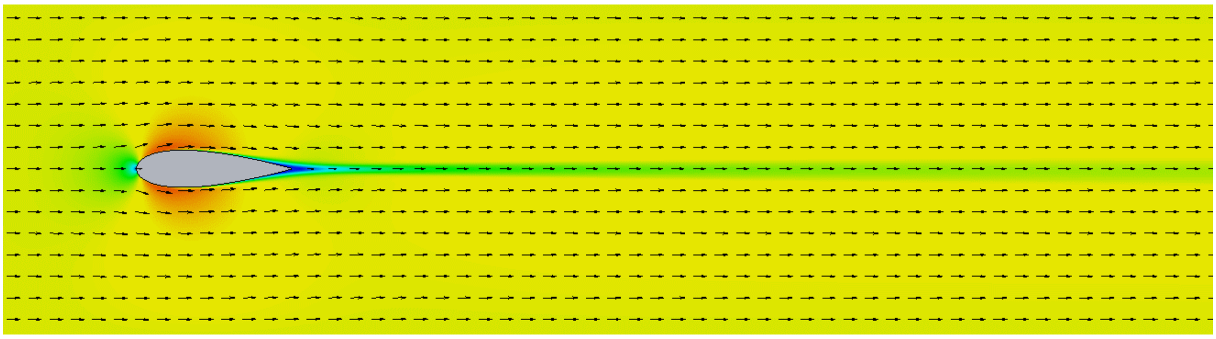

FIGURE 42, THE OVERALL FLOW FIELD VELOCITY REPORTED FOR THE NACA0024 PROFILE GENERATED BY THE METHODOLOGY

The following behaviour can be noted in the result:

- Laminar flow is achieved across the profile until the latter portion of the tail where separation begins
- Uniform deviation and compression of the fluid is visible as the fluid moves around the profile
- The fluid realigns with the ambient condition quickly after the end of the profile

The profile provides a result that would be defined just out with an acceptable level of accuracy of +/- 10% whilst providing a valid fluid behaviour as observed by the plots, this leads to the following conclusions:

- The solution can suitably capture the fluid dynamic phenomena
    - Boundary layer across the profile
    - Separation but stability at the rear portion

- The solution has a few issues:
    - Probable overestimation of the frictional forces, given that this is a significantly different type of solution compared to the cylindrical profile especially for the boundary layer, the following statements are considered:
        - The y+ value is too small to accurately capture the frictional forces adequately compared with the cylindrical profile
        - The y+ value is too large to accurately capture the frictional forces adequately compared to the cylindrical profile
    - Overestimation of the TKE in the downstream portion
        - The TKE may be overestimated leading to a false conclusion on the drag force present
        - The mesh shaped that effectively captures the higher turbulence values for the cylinder may overestimate the values for the more efficient NACA profile
    - The system is probably more sensitive when considering a NACA profile:
        - Significantly lower Cd value than a cylinder
            - Deviations in the solution are more pronounced as a % error
        - Significantly lower turbulence generation than a cylinder
        - Higher surface area and thus frictional force

4.3.2. Eppler profile

The Eppler profile was simulated using the defined standard boundary conditions, the following conclusion is made about the profile from the result:

- A converged solution was obtained
- Stability was not fully obtained
- The aerodynamic behaviour was not as expected from literature
- The error in the coefficient of drag was +83% from the accepted value

Figure 43 reports the overall profile result for velocity

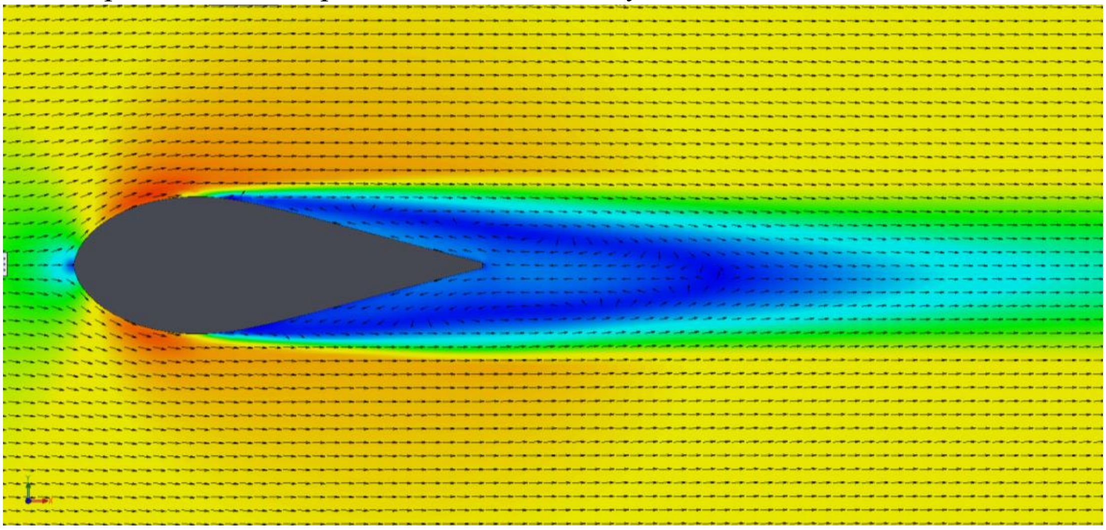

FIGURE 43, THE PROFILE VELOCITY PLOT FOR THE EPPLER863 PROFILE AND THE OVERSTATED BOUNDARY LAYER SEPARATION FROM THE WIDEST POINT OF THE PROFILE

Figure 44 reports the downstream tail velocity

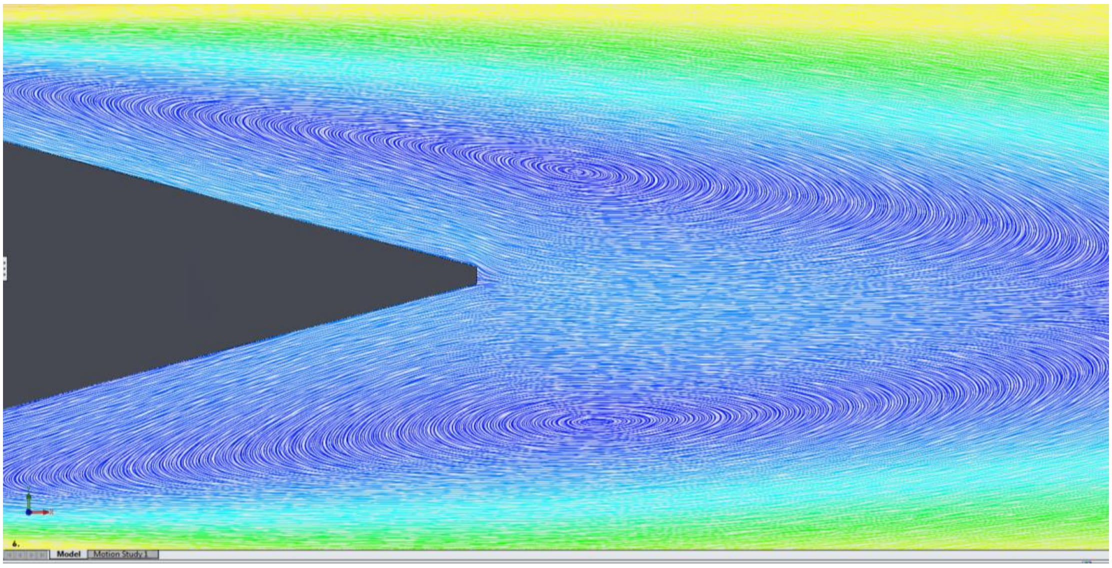

FIGURE 44, THE DOWNSTREAM TAIL VELOCITY PLOT AND SUBSEQUENT TURBULENCE GENERATION FROM FLOW SEPARATION

Figure 45 details the overall domain velocity profile

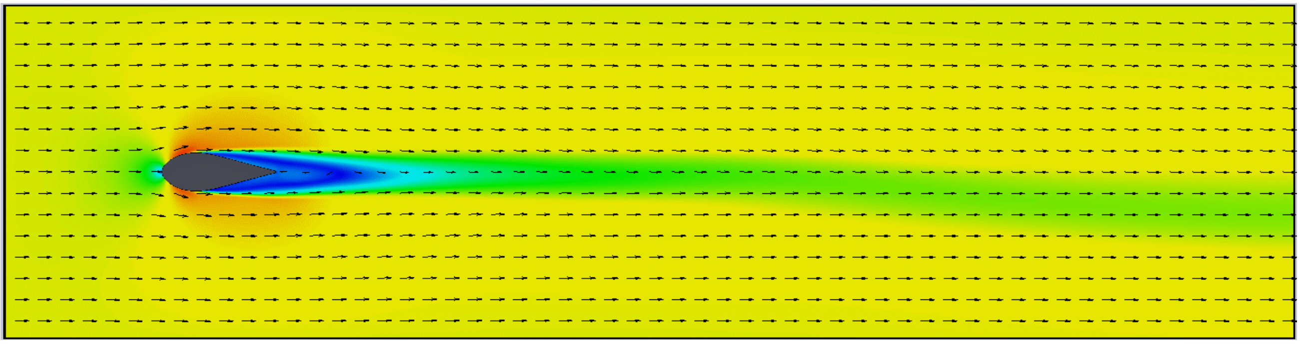

FIGURE 45, THE OVERALL DOMAIN FLUID FLOW VELOCITY REPORTED WITH THE SUBSEQUENT ANOMALOUS FLOW FIELD RESULT

The following comments can be made about the fluid behaviour reported:

- Flow separation occurs at the widest part of the profile, this may not be the case for an Eppler profile with the given characteristics in reality
- A large turbulent zone is created in the downstream of the widest point of the profile
- The size and scale of the separation zone leads to suggest anomalies within the values both for turbulence and for the dynamic pressures
- The flow field deviates in the latter portion of the solution

The methodology yields a Cd result that is +83% compared to the standard literature accepted value leading to several hypotheses and conclusions:

- Flow separation is over exaggerated because of the y+ value not being able to suitably capture the boundary layer energy

- The mesh is unable to suitably resolve the realignment of the flow downstream from the widest section of the profile
    - The mesh is not suitably defined around this zone of flow realignment
    - The general flow field grid is not sufficient to aid stability and resolution for a case with such fluid flow conditions, this could be from the cells growth and gradients being incompatible

- The geometry specific methodology has created inappropriate cell quality
    - The profile has a higher aspect ratio compared to the NACA profile which would also be the closest boundary layer dependent case
    - The profile has a higher separation angle than the NACA profile and thus more resolution is required downstream of the separation point
    - The general domain and cell sizing may be incompatible because of the shape of the profile

4.4. Investigation into the optimisation of the methodology

An optimisation was carried out for the studies undertaken with the two aerospace standard profiles; this was done to investigate the following:

- Assess if the mesh control parameters where also the most effective for these cases as with the cylinder profile study
- Improve on the profile specific protocol by adding extra mesh controls
- Reach a more accurate solution within the scope of 10% validity for Cd

Therefore, the following optimisation parameters where considered for optimisation:

- Y+ value
- Cell refinement strategies

With the final focus on developing profile specific rules for each of the two air foil cases.

The NACA0024 profile achieved a Cd value of +12% when compared to the literature accepted values, this value is near to the acceptable deviation of 10% and therefore the solution is expected to be sensitive to changes in the meshing conditions.

Conversely, the Eppler863 profile achieved a Cd value of +83% when compared to literature accepted values, this value is far from the acceptable deviation of 10%, it was expected that the optimisation would yield positive insights into the causation of such poor Cd values and an invalid velocity profile.

4.4.1. Y+ optimisation

A study was undertaken to vary the y+ value according to flow and profile conditions, chart 18 details the sensitivity study.

Results NACA0024

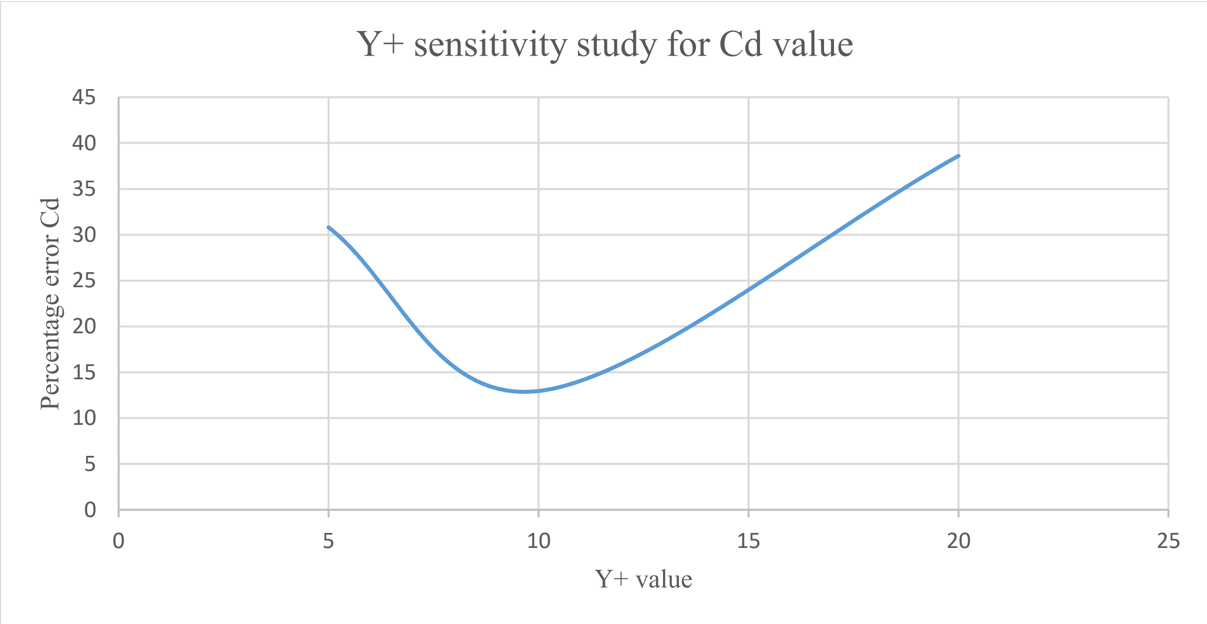

CHART 2, THE OUTPUT FOR THE Y+ SENSITIVITY STUDY FOR THE CD VALUE

The minimum value of percentage error corresponds to the y+ value defined originally by the methodology, meaning that the method for estimating the initial y+ value stands correct and is a valid approach for modelling the boundary layer of profiles equitable to the NACA0024.

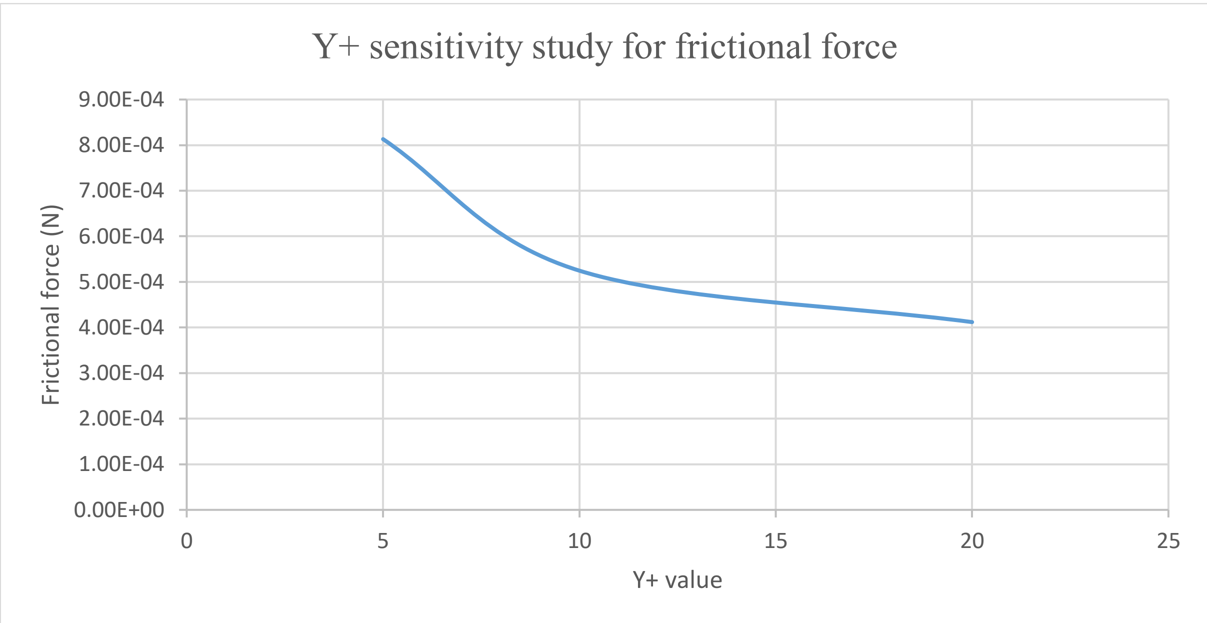

CHART 3

The results in chart 19 for frictional force follow a similar but not identical trend as that of the cylinder study. It is expected that the frictional force values reduce with an increase in the y+

value and therefore a trade-off between frictional force and turbulence generation is found in the grid around the defined y+ value.

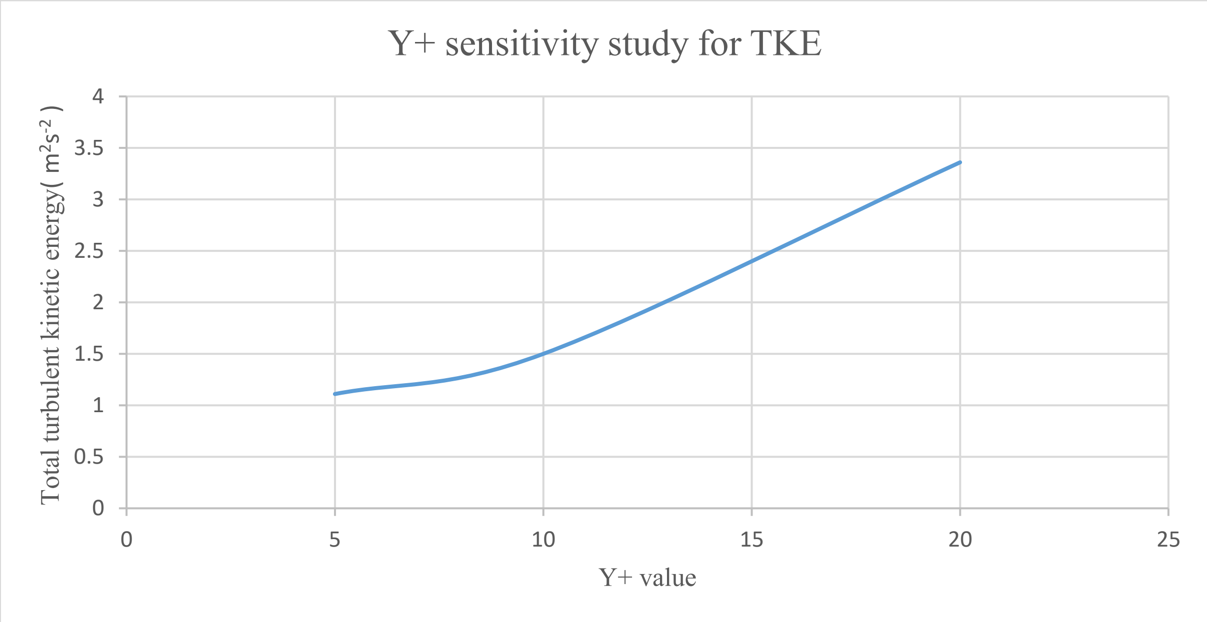

CHART 4, THE OUTPUT OF THE Y+ SENSITIVITY STUDY FOR TKE VALUES

Conversely in chart 20 the sensitivity to the y+ for the total kinetic energy is observed, therefore it can be deduced that there is a trade of for the y+ value between turbulent kinetic energy generation and the frictional force around the body, this trade-off is yields to be the same for both the cylindrical and NACA0024 profiles, meaning that no optimisation was achieved with the y+ value.

Results Eppler863

The same study was undertaken for the variation in the y+ for the Eppler863 profile, charts 21, 22 and 23 denote the results

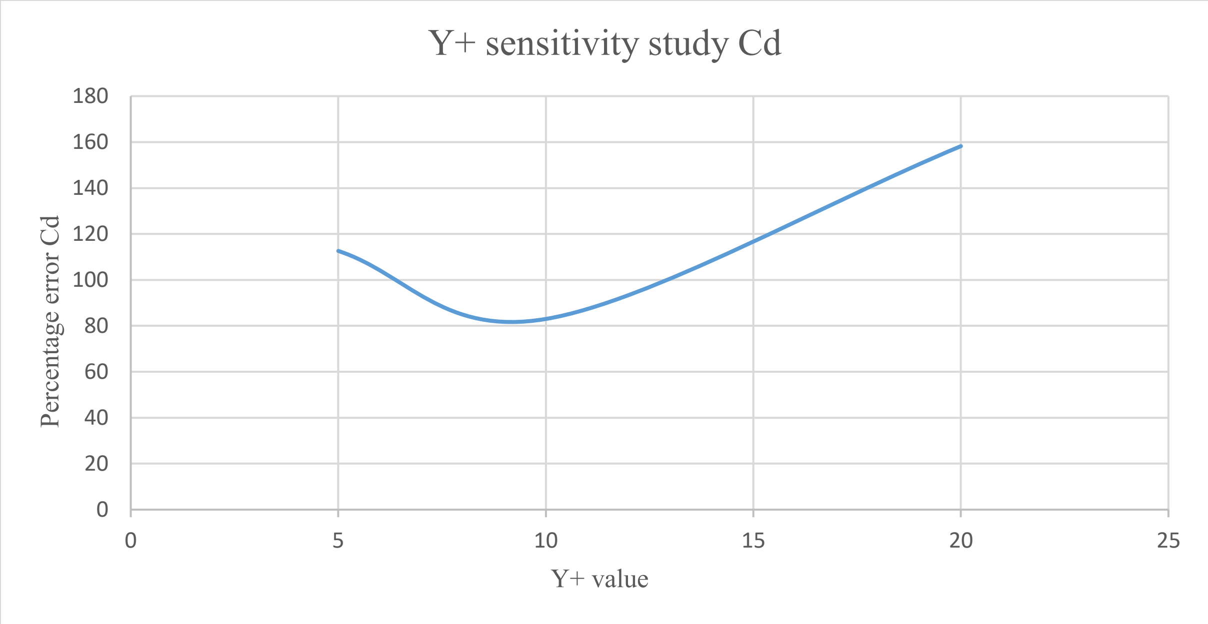

CHART 5, OUTPUT OF Y+ SENSITIVITY STUDY FOR CD FOR EPPLER 863 PROFILE

In chart 21 the y+ sensitivity is reported, the trend follows a similar trend for that of the NACA0024 profile and an optimum is obtained. This optimum s^{-2} also corresponds to the optimum y+ established through the generic profile methodology meaning that the best-case scenario for the y+ is of 83%. m^2

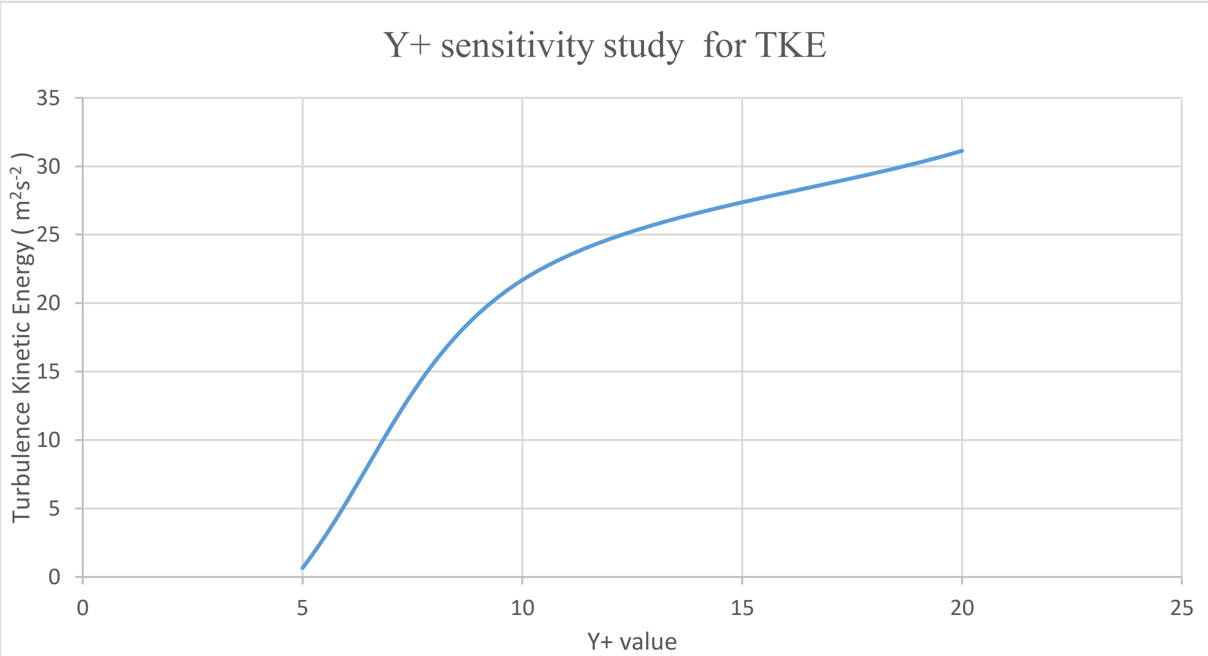

CHART 6, OUTPUT OF SENSIVITY STUDY FOR TKE FOR EPPLER 863 PROFILE

The TKE is reported in chart 22, this trend is different compared to the NACA0024 or the cylindrical profile. The reason for this different trend line is likely because of the nature of the separation of the flow and therefore the TKE does not increase in a linear fashion with an increase in the y+ value, this could be for a few reasons:

- The turbulence generated by the cells downstream is not influenced as much by the cells within the boundary layer
- The boundary layer is insufficiently modelled for this case
- The general condition for this type of profile is that the boundary layer does not influence turbulence generation as much as the other profiles

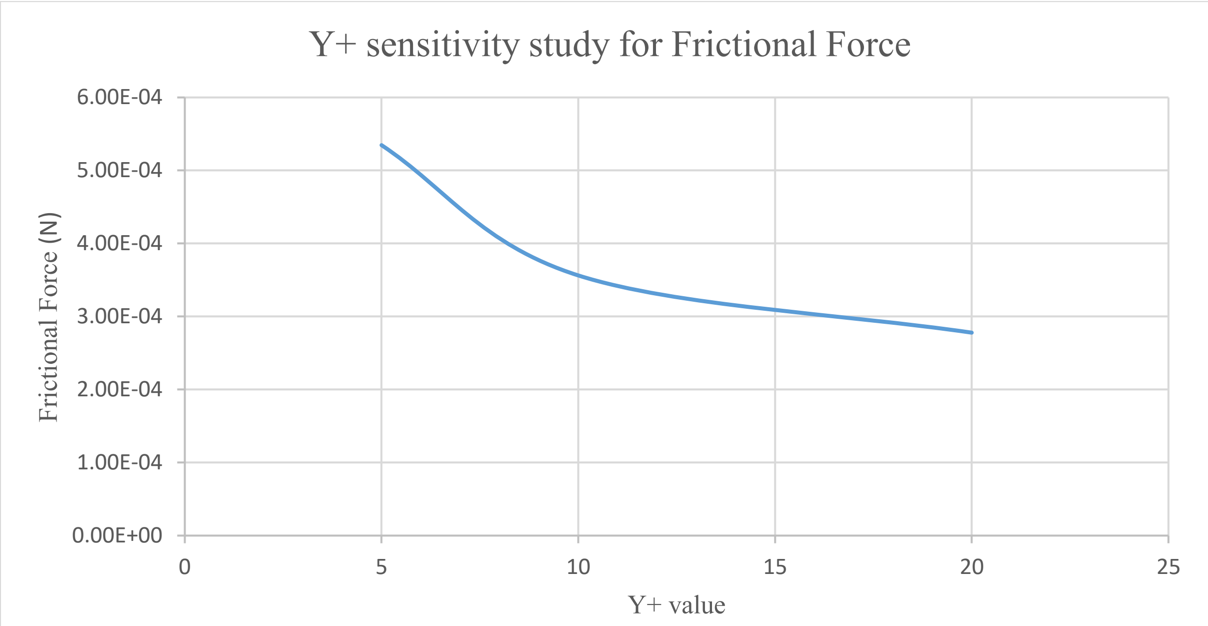

CHART 7, OUTPUT FOR Y+ SENSITIVITY STUDY FOR FRICTIONAL FORCE FOR THE EPPLER 863 PROFILE

Frictional force in chart 23 follows a similar trend to that of the NACA0024, again, the y+ selection yields true for the 'best case scenario' when dealing with the optimisation of the y+. it is conclusive that the problems surrounding the Eppler863 profile case are not due to the modelling of the boundary layer.

Cell refinement optimisation

A cell refinement study was undertaken to assess if modifying the surrounding cells from the y+ values could improve the solution. This was undertaken under the hypothesis that the resolution of cells around the modified flow field leads to better resolution for pressure and velocity gradients between the boundary layer and the ambient flow field.

Figures 46, 47 and 48 detail the difference in adding the controls to the meshing, this was done through the previously discussed mesh controls defined in SolidWorks Flow Simulation.

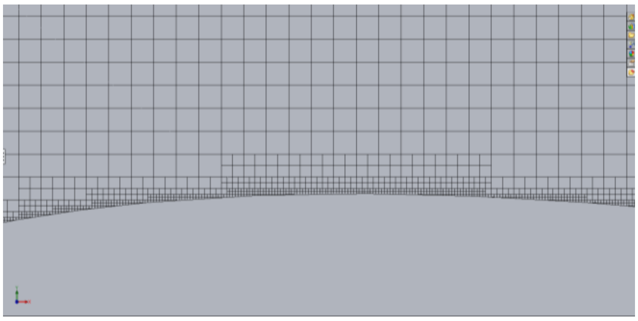

FIGURE 46, THE COMPUTATIONAL GRID WITHOUT THE REFINEMENT OF CELL OPTIMISATION

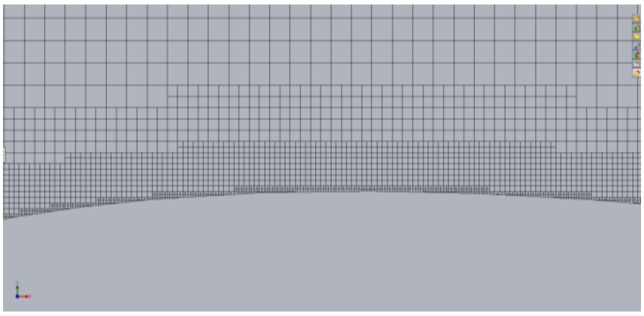

FIGURE 47, THE COMPUTATIONAL GRID WITH CELL REFINEMENT AROUND THE Y+ MINIMUM CELL SIZE

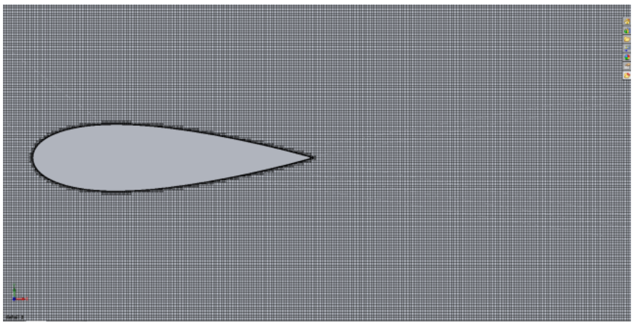

FIGURE 48, OVERALL COMPUTATIONAL GRID FOR SOLUTION WITHOUT INCREASED INFLATION LAYERS

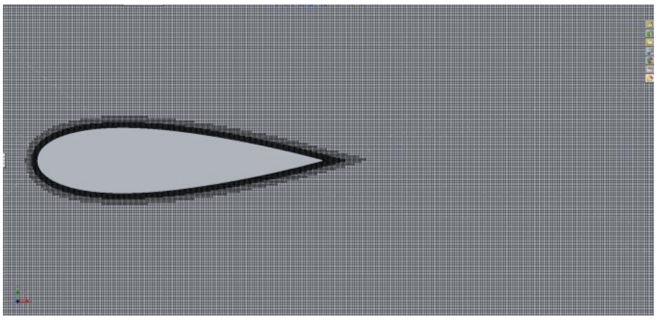

FIGURE 49, OVERALL COMPUTATIONAL GRID FOR SOLUTION WITH INCREASED INFLATION LAYERS

Results NACA0024

The study undertaking cell refinement yielded the following results in chart 24:

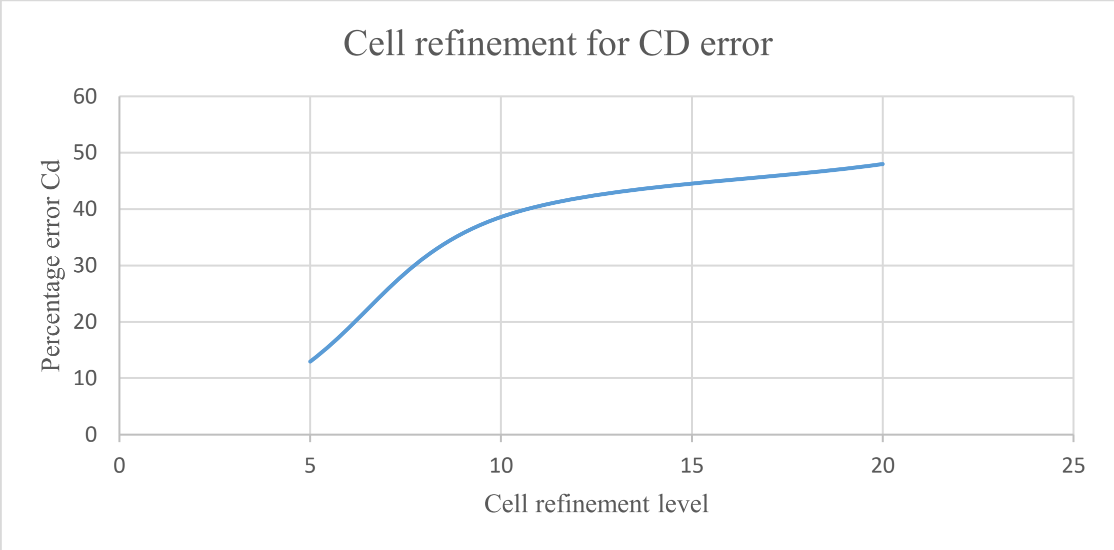

CHART 8, REPORTED RESULT FOR CELL REFINEMENT LEVEL AGAINST CD ERROR FOR THE NACA0024 PROFILE

The error in the Cd value increased with an increase in cell resolution for the NACA0024 profile, leading to the conclusion that increasing cell refinements was detrimental to the specific profile and flow case, this is further backed up when observing the velocity profile plots for incrementally worsening solutions.

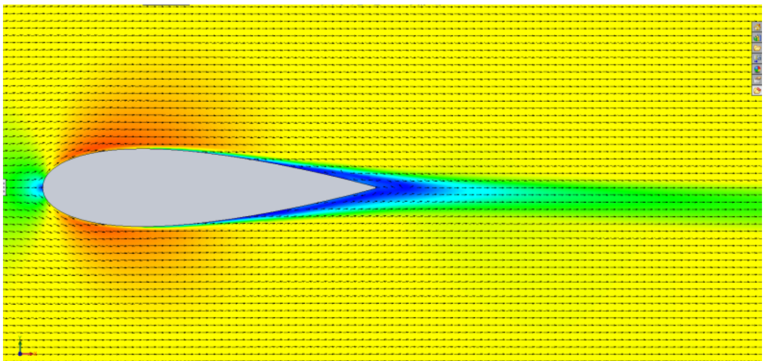

FIGURE 50, REPORTED VELOCITY PROFILE FOR MODIFIED CELL REFINEMENT

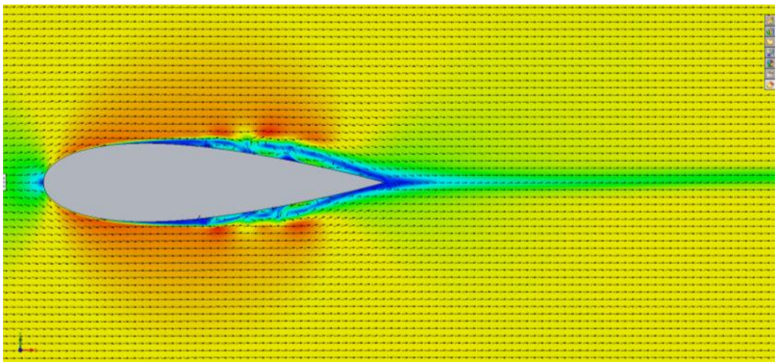

FIGURE 51, REPORTED VELOCITY PROFILE FOR HIGH DENSITY CELL REFINEMENT

As can be observed in figure 50 and figure 51, the increase in the density of the surrounding cells from the profile is detrimental to the stability and accuracy of the solution.

Results Eppler863

The result of the Eppler863 study are reported below in chart 25:

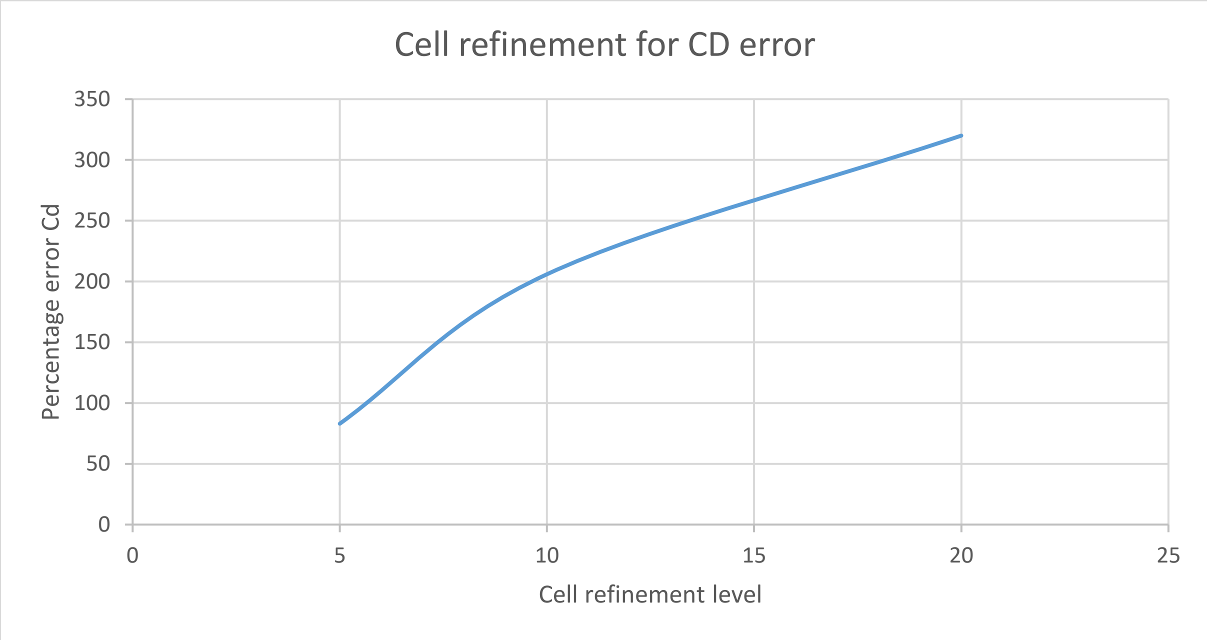

CHART 9, REPORTED OUTPUT FOR CELL REFINEMENT AND CD ERROR FOR THE EPPLER863 PROFILE

Again, for the increasing in cell refinement around the y+ cells a detrimental effect is observed. The optimisation of the solution through increasing the cells did not increase stability nor the Cd error within the solution

4.4.2. Conclusion of optimisation

To conclude for the activities involving an attempt to optimise the solution, no further progress was achieved on improving the solution accuracy by modifying the mesh. This is for several reasons:

- Both the profiles had obtained a y+ value that was sufficient for SolidWorks Flow to model the flow according to the specified conditions
- SolidWorks Flow Simulation runs utilising cell averaging and therefore not a great amount of improvement can be made on the solution through minor modifications to the computational grid

The following conclusion was also made:

- Cell refinements did not improve the accuracy of the surrounding flow field gradients for velocity and pressure as expected

To further improve the accuracy of the solution, the following methods should be considered:

- Modifying the solver
- Changing the meshing method, this was not available through SolidWorks Flow Simulation at the time of writing
- Changing the boundary conditions to values that may be easier for the solution to solve

5. Conclusion

To conclude, a methodology was developed that satisfies the criteria for the scope of this research, meaning that a utilisable method was determined that can be used in future to provide development insights at Canyon GmbH.

The following positive outcomes were achieved from this research:

- The construction of a geometrically specific CFD simulation for development insights for bicycle applications
- Accurate modelling for some expected profile cases in industry
- Indications and resolution significant enough to provide feedback for design insights of a specific product or idea
- An economically viable development which can be utilised for preliminary investigations and indications

There are short comings for the described methodology within this research, these are namely:

- The overall accuracy of the method may not be sufficient for effectively determining the Cd values of some profiles.
- The methodology is limited for what it can achieve with some profiles, namely profiles which have a higher separation angle and boundary layer dependency
- Deeper insights of the fluid dynamic behaviour induced by a respective air foil were not established, this is partly due to the limitation of SolidWorks Flow Simulation at the time of writing, but improvements can be made to the methodology to cover:
    - Transient flow cases
    - Wake behaviours
    - Profile surface pressures

To further develop this methodology in the future, the following steps could be taken:

- Investigate other available solver models in SolidWorks Flow Simulation to assess if they can better capture the fluid dynamic behaviours
- Classify specific cases for each profile in a great amount of detail:
    - Classification of profiles by estimating the boundary layer dependency of each problem
    - Classification of profiles by approximating the separation angle
    - Classification of solutions by expected turbulent energy generated
- Validate the findings of this methodology using a more advanced and expensive software such as StarCCM+ or ANSYS Fluent
- Assess more profiles and shapes with the methodology, specifically bicycle industry specific profiles or the ones developed at Canyon GmbH

6. Bibliography

[1] B. Blocken, T. v. Druenen, Y. Toparlar and T. Andrianne, “Aerodynamic analysis of different cyclist hill descent positions,” *Journal of Wind Engineering and Industrial Aerodynamics* , vol. 181, pp. 27-45, 2018.

[2] H. Robertshaw, “Cycling weekly,” June 2017. [Online]. Available: https://www.cyclingweekly.com/news/racing/pros-hit-astonishing-speeds-130kmh-tour-de-suisse-descent-335610. [Accessed August 2017].

[3] L. J. Clancy, Aerodynamics, New York : Wiley and Sons , 1975.

[4] N. Hall, “Bernoulli's Equation,” National Aeronautics and Space Administration , 05 May 2015. [Online]. Available: https://www.grc.nasa.gov/www/k-12/airplane/bern.html. [Accessed August 2017].

[5] S. F. Hoerner, Fluid- Dynamic Drag, Bakersfield CA: S: F. Horner, 1965.

[6] N. Hall, “Boundary Layer , National Aeronautics and Space Administration,” 05 May 2015 . [Online]. Available: https://www.grc.nasa.gov/www/k-12/airplane/boundlay.html.

[7] A. D. Young, Boundary Layers, Washington : American Institute of Aeronautics and Astronautics , 1989 .

[8] J. Vanier, “THE BOUNDARY LAYERS IN FLUIDS WITH LITTLE FRICTION (translated),” NATIONAL ADVISORY COMMITTEE FOR AERONAUTICS , Washington , 1950 .

[9] D. S. Pachpute, “CFD Modelling of Boundary Layer,” [Online]. Available: https://cfdflowengineering.com/cfd-modelling-of-boundary-layer/. [Accessed 2017].

[10] D. Balmer, Seperation of Boundary Layers, Edinburgh: School of Engineering and Electronics, University of Edinburgh, 2003 .

[11] J. D. Anderson, Fundamentals of Aerodynamics 5th Edition, New York: McGraw-Hill , 2011.

[12] P. Chang, Otryvnye techeniia vol.1, Moscow , 1972.

[13] G. Batchelor, An Introduction to Fluid Dynamics, Cambridge: Cambridge Mathematical Library , 2000.

[14] D. Wilson, Bicycling Science, MIT Press , 2004 .

[15] S. Bogos, A. Dumitrache and F. Frunzulica, “Turbulence Model in CFD Simulation of Low-Reynolds Number Airfoils Flow,” AIP Publishing , Bucharest , 2015 .

[16] S. Wang, Y.Zhou, M. Mahbub and H.Yang, “Turbulent Intensity and Reynolds number effects on an airfoil at low Reynolds numbers,” *Physics of Fluids ,* no. 26, 2014.

[17] W. K. George, “Lectures in Turbulence for the 21st Century,” 2013.

[18] Science Learning Hub , “Causes of aerodynamic drag,” Science Learning Hub , 2011. [Online]. Available: https://www.sciencelearn.org.nz/resources/1346-causes-of-aerodynamic-drag.

[19] B. Kerloch, “LE DIMENSIONNEMENT DES RESEAUX D’ASSAINISSEMENT DES AGGLOMERATIONS”.

[20] P. J. LaNasa and E. L. Upp, A Practical GUide to Accurate Flow Measurement, Waltham, MA : Butterworth-Heimann , 2014.

[21] I. Tani, “Annual Review of Fluid Mechanics,” *Vol. 9.87 ,* January 1977.

[22] Y. Cengel and J. Cimbala, Fluid Mechanics Fundamentals and Applcaitions (Third Edition), New York : McGraw-Hill , 2010.

Acknowledgements

My first thank you goes to the team at Canyon for giving me the privilege to undertake this research within the company, thank you to Michael Adomeit especially for his supervision, his guidance and his much appreciated mentoring on ‘all things aero’. More specifically I would like to thank the crew in the road bike department for their help, their openness, and their friendship during my 6 months in Koblenz. I would like to mention everyone else from Canyon that I had the pleasure of meeting during my time there, I had a lot of fun and some treasured memories of my time within the company, both professionally and socially.

I would like to thank my parents, Moira and Ian, and my sister Shona, for all the support that they gave me not only in completing this thesis, but the support, motivation and love that I received during the difficult times I faced during my academic career. Without them, I know that none of this would have been possible and so I am eternally grateful.

A mention goes to all the friends I made in my time in Torino, both through academic and other activities that shouldn’t be mentioned here! They know who they are, and I hope that I can express my gratitude and love to them and how much it means to be part of a diverse, open, hard working and hard partying network of people, who want to make the world a better place.

Grazie a tutti quelli dal CUS Torino Rugby e GipeTOUCH che mi hanno supportato, sfidato e aiutato sia nel campo di rugby che nella mia vita a Torino, è stato sempre un piacere giocare insieme e sicuramente vi ricorderò con tanti bei memori, la rugby italiana rimarrà sempre nei miei pensieri.

Lastly, I want to send my appreciation to my girlfriend Özgün, without her love I would have never remained sane throughout these tough times and for never giving up on me. Her resilience, empathy and determination continue to inspire me every day.