INVESTIGATION OF WIND LOADS ON BRIDGE DECKS – AN EXPERIMENTAL
AND NUMERICAL ANALYSIS

by

Anna Lena Assel

A project

presented to Ryerson University
and University of Applied Sciences Karlsruhe

in partial fulfillment of the
requirements for the degree of
Master of Engineering

in the Program of
Civil Engineering

Toronto, Ontario, Canada, 2017

© Anna Lena Assel 2017

AUTHOR'S DECLARATION

I hereby declare that I am the sole author of this project. This is a true copy of the project, including any required final revisions.

I authorize Ryerson University to lend this project to other institutions or individuals for the purpose of scholarly research.

I further authorize Ryerson University to reproduce this project by photocopying or by other means, in total or in part, at the request of other institutions or individuals for the purpose of scholarly research.

I understand that my project may be made electronically available to the public.

Abstract

Wind tunnel testing is very expensive, especially due to the handmade models and the cost of building and operation of wind tunnels. Therefore, numerical modelling such as computational fluid dynamics can be a more cost efficient solution and is seen to take the leading position in the future. The following research contains the validation of the Computational Fluid Dynamics tool VXFlow by comparing the numerical results with values in literature, the Eurocode and wind tunnel measurements. The study includes the analysis of static wind loads for different wind barrier geometries for the example of the Süderelbebrücke in Hamburg. VXFlow is found to give good results in a short computational time but tends to overpredict the drag coefficient. The application is aligned for investigations in the preliminary design of bridge decks and can be also used for wind tunnel validation.

Keywords: Bridge deck, CFD tools, VXFlow, Static wind loads, Wind barrier

Acknowledgements

The support of many people made the current study possible. My special thanks to Prof. Dr. Ramani Ramakrishnan, Prof. Dr. Götz Bramesfeld and Frank Bowen from Ryerson University, who supported the performance of the wind tunnel test in the laboratories of the Aerospace Engineering Department at Ryerson University in Toronto. Many thanks to my supervisors Prof. Dr. Markus Baumann from University of Applied Sciences Karlsruhe and Prof. Dr. Ahmed Shaker from Ryerson University Toronto, who guided me throughout the study. I want to thank Prof. Dr. Guido Morgenthal from Bauhaus-Universität Weimar for introducing me to the state of arts of wind tunnel investigations and providing me with the Software VXFlow, which allowed a great insight into the topic of Computational Fluid Dynamics. Finally I want to thank Dr. Abraham Sanchez and Dr. Burkhard Krenn, who provided valuable knowledge and advice for any arising issues during the work in the office schlaich bergermann partner in Stuttgart. Many thanks for your support and advice.

Contents

Author's Declaration	ii
Abstract	iii
Acknowledgements	iv
List of Tables	vii
List of Figures	viii
List of Appendices	x
1 Introduction	1
1.1 Evolution of Wind Engineering	1
1.2 Purpose of the Study	2
2 Wind Loads	3
2.1 Wind Climate	3
2.2 Surrounding Conditions	3
2.3 Fluid Structure Interaction	4
3 Fluid Dynamics	5
3.1 Reynolds Number	5
3.2 Velocity and Pressure	5
3.3 Pressure Coefficient	7
3.4 Acting Wind forces	9
3.5 Force Coefficient	10
4 Physical Equations Describing Fluid Flow	11
4.1 Continuity Equations	11
4.2 Material Equations	18
4.3 Navier-Stokes Equations	20
5 Numerical Approach	21
5.1 Overview of Numerical Models	21
5.2 Vortex Particle Method in VXFlow	23
6 The VXFlow Package	29
6.1 Input	29
6.2 Output and Post-processing	32
6.3 Pressure Distribution and Coefficients	32
6.4 Force Coefficients	37
7 Comparison of Numerical and Experimental Analysis	42
7.1 Numerical Analysis	43
7.2 Experimental Analysis	48
7.3 Comparison and Conclusion	49

8 Investigation of different Wind Barriers	54
8.1 Wind Barrier Design	56
8.2 Effects on the Bridge Deck	56
8.3 Effects on the Vehicles	60
8.4 Summary	66
9 Conclusion	67
Appendices	67
A VXFlow Software	70
B Investigation of different Wind Barriers	73
B.1 Force and Moment Coefficients for Bridge Deck	73
B.2 Force and Moment Coefficients for Windward Vehicle	73
B.3 Force and Moment Coefficients for Leeward Vehicle	73
Bibliography	86
Glossary	87

List of Tables

4.1	Mass flow	12
4.2	Conversion of non-conservation to conservation form	15
4.3	Unknown variables of Continuity Equations	18
6.1	Input parameters VXFlow after Morgenthal	29
6.2	Output files VXFlow after Morgenthal	32
6.3	Comparison - Pressure coefficients of cylinder 1	34
6.4	Pressure coefficients of cylinder 2	35
6.5	Drag coefficients of rectangular sections	39
7.1	Reference values for computing coefficients	43
7.2	Summary of deviations of force coefficients (numerical - experimental)	50
7.3	Blockage ratio of experimental study	52
8.1	Criteria and parameters for wind barrier design	54
8.2	Models for investigation of wind barriers	56
8.3	Summary of deviations of force and moment coefficients (bridge deck)	59
8.4	Reference values for computing coefficients of vehicle	61
8.5	Summary of deviations of moment coefficients (vehicles)	64
8.6	Summary of standard deviations of drag coefficients (vehicles)	65

List of Figures

2.1	Boundary layer; mean wind speed profile	3
2.2	Types of fluid structure interaction	4
3.1	Flow field: Velocity and pressure	5
3.2	Bernoulli on the example of the flow field around a sphere	6
3.3	Lift, drag and moment in the global coordinate system	9
4.1	Navier-Stokes Equations	11
4.2	Mass flow	12
4.3	Flowchart, forces on fluids	13
4.4	Normal and shear forces in x-direction	14
4.5	Heat flow and forces due to working done on the element in x-direction	16
5.1	Numerical models	21
5.2	Visualisation of Lagrangian description	23
5.3	Numerical model of VXFlow	25
5.4	Release of vortex particles	26
5.5	Computation of interaction by cell-to-cell scheme	26
5.6	Variation of merging distance	28
6.1	Computational Domain in VXFlow	31
6.2	Geometry of cylinder	33
6.3	Pressure coefficients of cylinder 1	34
6.4	Pressure coefficients of cylinder 2	35
6.5	Geometry and pressure coefficients of cylinder collocation	36
6.6	Pressure on closed body due to constant pressure	37
6.7	Comparison of Drag Coefficients	38
6.8	Rectangular sections - Comparison of Drag Coefficients (a)	40
6.9	Rectangular sections - Comparison of Drag Coefficients (b)	40
7.1	Visualisation of models T1, T2, T3	42
7.2	Coefficients for time steps 1 to 10000 (Model T1, 0°)	44
7.3	Lift, drag moment coefficients for time steps 1001 to 10000 (Model T1, 0°)	44
7.4	Lift, drag, moment coefficients for Model T1	45
7.5	VXFlow - Variation of lift and drag coefficients (Model T1, T2, T3)	46
7.6	Velocity field around Model T1 (0°)	47
7.7	Velocity field around Model T2 (0°)	47
7.8	Velocity field around Model T3 (0°)	47
7.9	Model T1	48
7.10	Locations of measurement points	49
7.11	Wind tunnel - Variation of lift and drag coefficients (Model T1, T2, T3)	49

7.12	Wind tunnel results: Variations of lift and drag coefficients (Model T1, T2, T3)	50
8.1	Loss of space due to original wind barrier design	55
8.2	Force and moment coefficients by comparison	57
8.3	Velocity field around Model B1 (0°)	58
8.4	Velocity field around Model B1 (10°)	58
8.5	Velocity field around Model B1 (-10°)	59
8.6	Overturning and sideslip of vehicles due to wind	60
8.7	Velocity field around Model B1 including vehicles (0°)	62
8.8	Velocity field around Model B1 (10°)	62
8.9	Velocity field around Model B1 including vehicles (-10°)	62
8.10	Moment coefficients by comparison of windward vehicle	63
8.11	Moment coefficients by comparison of leeward vehicle	63

List of Appendices

B.1	Lift coefficients of Model B1 to B5 for bridge deck	74
B.2	Drag coefficients of Model B1 to B5 for bridge deck	75
B.3	Moment coefficients of Model B1 to B5 for bridge deck	76
B.4	Deviations of lift coefficients in relation to Model B1 for bridge deck	77
B.5	Deviations of drag coefficients in relation to Model B1 for bridge deck	77
B.6	Deviations of moment coefficients in relation to Model B1 for bridge deck	77
B.7	Lift coefficients of Model B1 to B5 for windward vehicle	78
B.8	Drag coefficients of Model B1 to B5 for windward vehicle	79
B.9	Moment coefficients of Model B1 to B5 for windward vehicle	80
B.10	Deviations of lift coefficients in relation to Model B1 for windward vehicle	81
B.11	Deviations of drag coefficients in relation to Model B1 for windward vehicle	81
B.12	Deviations of moment coefficients in relation to Model B1 for windward vehicle	81
B.13	Lift coefficients of Model B1 to B5 for leeward vehicle	82
B.14	Drag coefficients of Model B1 to B5 for leeward vehicle	83
B.15	Moment coefficients of Model B1 to B5 for leeward vehicle	84
B.16	Deviations of lift coefficients in relation to Model B1 for leeward vehicle	85
B.17	Deviations of drag coefficients in relation to Model B1 for leeward vehicle	85
B.18	Deviations of moment coefficients in relation to Model B1 for leeward vehicle	85

1 Introduction

Dynamic and static wind loads can be very harmful for structures. For a safe design it is necessary to be aware of the wind loads, which are likely to occur during the lifetime of an asset. Static wind loads are dependent on the outer geometry and the site location of a structure. Dynamic wind loads depend on mass, stiffness and damping ratio additionally. Furthermore, wind is not a static phenomenon but its characteristics change with time, which increases the complexity of predicting wind loads. Hence, due to the high number of influences it is not possible to give standardised wind loads especially for special structures and geometries. Wind tunnel testing is the common approach in the latest two centuries to predict wind loads and the dynamic behaviour of structures. Nowadays, Computational Fluid Dynamics (CFD) is applied complementary, and for some cases it can replace wind tunnel testing. Wind Engineering is the discipline analysing structures in terms of wind and its history is described in more detail in the following section.

1.1 Evolution of Wind Engineering

The first wind tunnel studies for civil engineering purposes were conducted in the 18th century. At that time, drag forces were analysed by moving models through the air. The technique of using a fixed model, which allows the fluid to move around the body, was developed in the following century. In the beginning, only models in smooth flowing wind could be tested, until atmospheric turbulence could be simulated by boundary layer wind tunnels around 1950 [1]. Since then, the improvement of wind tunnel testing has led to more sophisticated and meaningful studies.

State of the art wind engineering uses wind tunnel tests to predict wind loads and wind response of structures, especially for unconventional geometries. The possibilities range from relatively simple force balance tests to much more elaborate investigations which, for example, include a small-scale aerodynamic model of an entire bridge. Due to the complexity of wind tunnel test, regarding the performance and the analysis of results, expertise is required. Moreover, the majority of tests are time-consuming and expensive. Additionally, if the bridge design were to be changed, testing would be increasingly time-consuming and costly, due to the difficulty of adaption.

The development of CFD codes, which provide computer-aided analysis for fluid-structure interaction, facilitates similar investigations on a computer than wind tunnel testing. The method had many limitations especially due to computational cost, and the accuracy was not always certain due to the simplifications introduced to the physical models in order to save computational time and cost. However, the development of computer technology and research in computer modelling of fluid flows lead to gradually improvement of CFD packages to become more useful and affordable for civil engineers in the design process of structures.

The advantage of CFD tools is that it enables engineers to investigate without the need of any facilities other than computer hardware and software. Changes in the structure can be made very easily, and the

possibilities expand to studies which are not feasible with conventional ones. For example, it is possible to include the optimization of structures for wind loads in the software calculations. Indeed, despite these possibilities, lack of familiarity with the software and some reservation in terms of accuracy need to be overcome. In particular, CFD tools are still in development for getting a good agreement between accuracy and computational cost.

1.2 Purpose of the Study

In the interest of becoming more familiar with CFD tools, well known examples of literature are investigated with VXFlow. As a first comparison the infinitely long cylinder in flow is examined. The flat plate serves as a second example for validating the software. Additionally, a bridge section model test was carried out in a wind tunnel. The acquired data of this wind tunnel study will be compared with results of VXFlow in order to analyse the differences between the two methods. The quality of the results is analysed so that conclusions can be drawn for future projects, and to determine in which cases a costly wind tunnel test can be replaced or complemented by a study with VXFlow.

Therefore, this document provides theoretical background of wind loads in general, Fluid Mechanics and Computational Fluid Dynamics. The physical equations, which describe the flow of a fluid perfectly are derived and numerical approaches to solve these physical equations are introduced subsequently. This includes the description of the Vortex Particle Method, on which VXFlow is based.

After result validation the last part provides an analysis of the forces acting on the bridge depending on different wind barrier geometries. The current study focuses on the analysis of static loads of the Süderelbe Bridge in Hamburg-Moorburg, whereas the influence of the different geometries on the wind loads of the bridge deck is investigated. Additionally, the effect on the vehicles on the bridge is considered by analysing the influence of the change in geometry on the overturning moment and fluctuation of forces due to cross wind.

Indeed, the analysis of the dynamic response of bridges to wind forces is of major interest. As civil engineers learned after the collapse of the Tacoma Narrows Bridge in Washington, USA, in 1940 due to aeroelastic flutter, the investigation of dynamic response is essential, especially for slender structures. However, wind tunnel testing and numerical analysis of the dynamic behaviour is beyond the scope of this study.

2 Wind Loads

Reproducing the properties of wind is a main goal in wind engineering to ensure the comparability of the results. This principle applies for wind tunnel investigations, as well as computational modelling with CFD tools. The current chapter introduces wind loads, their components, and their relations for a better understanding of the incoming wind.

2.1 Wind Climate

The wind climate mainly determines the movement of air and wind velocity. Global atmospheric circulation is caused by the different surface temperatures on earth and pressure differences in the atmosphere [2]. At the equator hot air rises and moves toward the poles, which leads to a pressure deficit. Additionally, air cools down next to the poles, contracts, and creates space for air flowing in. The result is high pressure at the poles and a constant flow of air is caused. Furthermore, local phenomena like strong upward winds, storms, or dust devils have to be considered when designing a structure for wind loads.

2.2 Surrounding Conditions

The surrounding conditions influence the properties of wind in terms of velocity and turbulence [2]. The surrounding conditions, which include natural conditions (flat terrain to mountains) as well as artificial components like other buildings and structures, determine the roughness of terrain. Turbulences depend on the roughness of the surface: The rougher the surface, the bigger the turbulence intensity. Furthermore, the roughness of the surface also leads to the decrease of velocity, as a higher turbulence intensity always leads to a reduction of wind speed. Their distribution of lower speeds and higher turbulences closer to the surface and vice versa for higher levels is known as the atmospheric boundary layer.

Figure 2.1: Boundary layer; mean wind speed profile after Xu [3]

The change of the mean velocity depending on height is illustrated in Figure 2.1. Due to the permanent changes of the movement of air and the turbulences, wind loads are permanently changing over space and time. Therefore, wind has to be described in statistical terms like the mean velocity and their fluctuation components.

2.3 Fluid Structure Interaction

As the wind comes into contact with the surface of the structure, it creates pressure, suction and shear forces over it. The distribution of velocity and turbulence which occurs on the surface of the body is similar to the one for the atmospheric boundary layer. The wind velocity converts to a force acting over the surface of the structure. Furthermore, the shape of the structure highly influences the pressure distribution, which in turn stresses the structure itself. This relation, that the geometry of the structure influences the wind load is the basic form of fluid structure interaction (Type I). The current study is based on this type of fluid structure interaction, which is the study of static bodies in the flow of a dynamic fluid. Type II includes the global movement of the body. Speaking in terms of wind tunnel testing of rigid models which are fixed on springs, this is an example for this kind of fluid structure interaction. Deformations of the body itself are included in Type III, which is important for flexible structures for example slender bridge decks or high-rise buildings. The three main types of fluid structure interaction are illustrated in Figure 2.2.

Figure 2.2: Types of fluid structure interaction

For typical structures and shapes, values for wind loads can be found in literature or codes. They depend on different factors, which take wind climate and the topography into account. If the shape is more complex or a sophisticated structure is chosen, wind tunnel tests are common to determine the static wind loads. Also supplementary CFD analysis becomes more important and practical. As wind loads might depend on the wind-induced dynamic response of the structure, dynamic analyses needs to be carried out in certain cases. Additionally dynamic analysis is a tool to investigate dynamic effects like flutter or Vortex Induced Vibration, that might affect the structural design.

3 Fluid Dynamics

Fluid Dynamics is the study of the interaction between fluids and objects. With respect to Civil Engineering, it describes the relation between the stationary structure and the air; specifically the characteristics of how the air moves around the structure [4]. In order to understand Wind Engineering in general, and the research carried out, the basics of Fluid Dynamics are introduced in this chapter.

3.1 Reynolds Number

An important quantity in Aerodynamics, and for similar requirements of wind tunnel testing and CFD calculations, is the Reynolds Number, Re . It is a dimensionless number which relates the inertia forces with the viscous forces, and is used for distinguishing between laminar and turbulent flow. The Reynolds number is given by Simiu and Miyata [5]:

$$Re = \frac{uL}{\mu/\rho} = \frac{uL}{\nu} \quad (3.1)$$

Smooth flow with streamlines, being parallel to one another, is called laminar flow and is described by a low Reynolds Numbers ($Re \lesssim 10^3$) [2]. Accordingly, turbulent flow is described by higher values of Reynolds Numbers. In this case, the flow is dominated by inertia forces, swirling and large fluctuations in velocity. While laminar flows are more predictable as they are more steady, the increase of the turbulence intensity leads to more fluctuations and therefore, unpredictable effects. As the Reynolds Number of the test set-up needs to match the prototype conditions in order to achieve a similar wind flow, the scale for geometry and velocity has to be adapted accordingly. This concept applies for the model as well as the surrounding conditions and reproduction of the boundary layer.

3.2 Velocity and Pressure

Figure 3.1: Flow field: Velocity and pressure

As explained in the previous chapter, apart from turbulence intensity, wind is mainly characterized by its velocity and pressure. Velocity and pressure change in an interactive way in the flow field due to disturbances. The description of flow fields is mainly based on these two parameters, velocity and pressure, which is also

illustrated in Figure 3.1. The variables, velocity components u, v, w and pressure p are dependent on the location and also change with time. This relationship between velocity and pressure is expressed in the Bernoulli Equation (3.2) in the theoretical case of steady flow without viscous effects [5]. The basis for this equation is the concept of the conservation of energy.

$$p_{\infty} + \frac{1}{2}\rho u_{\infty}^2 = p + \frac{1}{2}\rho u^2 \quad (3.2)$$

where

- p_{∞} = Free stream pressure
- u_{∞} = Free stream velocity
- p = Pressure in flow field
- u = Velocity in flow field
- ρ = Density

Moreover, the velocity u at the face of the body, which is perpendicular to the flow stream, changes to zero locally if a body is placed in the streamline. The location in the flow field with zero velocity is known as stagnation point. Rearranging the equation above, the pressure on the body surface can be calculated as:

$$p = p_{\infty} + \frac{1}{2}\rho u_{\infty}^2 \quad (3.3)$$

Figure 3.2: Bernoulli on the example of the flow field around a sphere

The explained relations are also depicted in Figure 3.2 on the example of the flow around a circular section. Furthermore, the portions of static and dynamic pressure, which add up to the energy head H are illustrated qualitatively assuming no losses. It can be seen that an increase in static pressure p results in a decrease in dynamic pressure $0.5\rho u^2$ and vice versa, due to the fact that total energy is constant. Pressures p greater than the initial pressure p_∞ in the free flow, e.g. p_0 reflect positive pressure, contrary pressures with lower values, e.g. p_1 , indicate negative pressure (suction).

By following particles in the stream over space and time in the Lagrangian manner the changes in the flow field due to the interruption of a body can be visualized, as done in Figure 3.2. The first particle, illustrated with solid filling, moves with the mean velocity of the undisturbed flow and the pressure equals the atmospheric pressure. The velocity changes to zero when impinging on the body. This decrease of velocity leads to an increase of pressure to its maximum value. Following the particle which is coloured in white, it is assumed that the velocity increases when passing the body. Therefore, the pressure decreases to a value lower than zero, which indicates suction on the body surface in this region.

As mentioned before, the validity of the Bernoulli Equation is restricted to the steady flow of an inviscid fluid. The incorrect application of the Bernoulli Equation is common, and therefore the limitations are described in more detail [6]. The Bernoulli Equation does not include shear stresses due to the assumption of an inviscid flow. Viscosity is given for every real fluid. Values for the kinematic viscosity of typical gases have a wide range, for example $\nu = 1.5111 \cdot 10^{-5} \text{ m}^2/\text{s}$ for air and $\nu = 9,9279 \cdot 10^{-3} \text{ m}^2/\text{s}$ for honey at 20°C [7]. If the influence of viscosity is negligible, the assumption of an inviscid fluid is permitted.

Moreover, fluid density is considered to be independent of pressure in the Bernoulli Equation. This limitation to constant density is related to the incompressibility of the fluid. For relatively low pressures, as it is the case for usual civil engineering projects, the assumption of incompressibility does not introduce a large error to the results because fluids are characterized by a very low compressibility. The influence is of more importance for higher pressures, for example for the field of Aerospace Engineering, where velocities are close or even exceed the speed of sound.

The remaining assumption which is of concern for typical applications according to Potter and Wiggert [6], is the presence of steady flow. Steady flow is defined by all flow properties, such as velocity and pressure, are constant for every position in the flow field over time. However, steady flow is not present if turbulence characterize the flow, for example because of the atmospheric boundary layer, gusts or in the wake downstream a body. The Navier-Stokes Equations, which are included in Chapter 4, describe the flow in every detail without the limitations of the Bernoulli Equations.

3.3 Pressure Coefficient

Coefficients are used in Aerodynamics to describe the distribution of pressures and forces on the surface of an object subjected to wind. The pressure coefficient refers to a reference point, usually located in the free stream, and describes the ratio of static pressure to dynamic pressure. In other words, the pressure coefficient is a nondimensional number, which determines the pressure on a reference area depending on the

free stream wind velocity, u_∞ [5]. It is defined by the equation:

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho u_\infty^2} \quad (3.4)$$

Assuming an ideal fluid, the pressure coefficient cannot be greater than 1.0, which can be derived by rearranging the Bernoulli Equation 3.2. The equation reaches its greatest value at stagnation points as this means $u = 0$ and p is at the maximum. Furthermore, the coefficient expresses suction by negative values and pressure with a positive number due to the direct influence of the pressure p on the sign.

$$\begin{aligned} p_\infty + \frac{1}{2}\rho u_\infty^2 &= p + \frac{1}{2}\rho u^2 \\ p - p_\infty &= \frac{1}{2}\rho u_\infty^2 - \frac{1}{2}\rho u^2 \\ \frac{p - p_\infty}{\frac{1}{2}\rho(u_\infty^2 - u^2)} &= 1 \\ \frac{p - p_\infty}{\frac{1}{2}\rho u_\infty^2} &= C_p \leq 1 \end{aligned}$$

Alternatively the derivation can be given as:

$$\begin{aligned} C_p &= \frac{p - p_\infty}{\frac{1}{2}\rho u_\infty^2} \\ C_p &= \frac{(H - \frac{1}{2}\rho u^2) - (H - \frac{1}{2}\rho u_\infty^2)}{\frac{1}{2}\rho u_\infty^2} \\ C_p &= \frac{\frac{1}{2}\rho u_\infty^2 - \frac{1}{2}\rho u^2}{\frac{1}{2}\rho u_\infty^2} \\ C_p &= 1 - \frac{u^2}{u_\infty^2} \leq 1 \end{aligned}$$

This constraint of pressure coefficients to a maximum value of 1.0 can also be explained in means of energy. As the total energy is constant or decreasing, the excess of pressure at a location over that at infinity ($p - p_\infty$) cannot be larger than the dynamic pressure in the free stream $0.5\rho u_\infty^2$. This relation is also illustrated in Figure 3.2. In certain cases, especially at stagnation points, C_p values greater than 1.0 are observed and it is discussed in literature whether it is result of a numerical inaccuracy or result of redistribution of energy in real flows of viscous substances. Batchelor [8] states that the total pressure, consisting of static and dynamic pressure, in the flow increases if the net viscous force accelerates the fluid. However, he also declares that the total pressure decreases generally in flow direction due to work done against viscous forces. In summary, that means that locally pressure coefficients can overcome the value of 1.0 due to that the energy head H is greater at certain locations than the free flow head it is related to. But this local increase of total energy needs to be accompanied by a local decrease of an adjoining streamline because the total energy in the flow field cannot increase. This emphasises the fact that greater values than 1.0 can occur very sporadic only. A similar opinion is shared by Issa [9], who says that the local increase of total pressure in frictional flow is caused by the redistribution of mechanical energy due to the action of viscous stresses, and the influence

of the effect is proportional to the Reynolds Number, which itself is influenced by the viscosity. However, Norris [10] explains that this rise is negligible for highly turbulent flows, which are of main interest for engineers, and indicates an artificial error due to the application of low Reynolds Number models in CFD calculations. Those simulate e.g. higher viscosity and therefore, lead to an overprediction of viscous stresses. This correlation is supported by Batchelor [8], who claims that inertia forces predominate over viscous effects for high Reynolds Numbers.

3.4 Acting Wind forces

Since pressure is a force per unit area, forces can be calculated by multiplying pressures with the area. Forces acting on an object immersed on the wind flow are described by three components, called drag force F_D , lift force F_L and moment M . The definition is given in the following equations:

$$\begin{aligned} F_L &= \oint p_y dA \\ F_D &= \oint p_x dA \\ M &= \oint p_i \cdot e dA \end{aligned}$$

The drag force acts in the direction of wind flow, while the lift force acts perpendicular to it. To put it in other words, F_D is parallel and F_L perpendicular to the angle of attack, which defines the local reference system. By taking the integral of pressures in the respective direction, the specific forces can be calculated [3]. The moment M describes the force multiplied by the eccentricity of it to the section center. The relationship between the angle of attack α and the forces is illustrated in Figure 3.3. Furthermore, the horizontal and vertical forces (F_H, F_V) which are F_D and F_L expressed in the global reference system of the section geometry, are shown in the figure.

Figure 3.3: Lift, drag and moment in the global coordinate system after Xu [3]

3.5 Force Coefficient

According to pressure coefficients, force coefficients can be defined for lift, drag and moments. The equations are given in dependence on the angle of attack α [3]. Common reference lengths are presented, which are also utilized in the current study. However, other reference lengths are conceivable. In terms of wind tunnel testing, the drag force $F_D(\alpha)$, lift force $F_L(\alpha)$ and moment $M(\alpha)$ can be determined either by a force balance or by integration of pressures.

$$C_D(\alpha) = \frac{F_D(\alpha)}{\frac{1}{2}\rho u_\infty^2 \cdot LH} \quad (3.5)$$

$$C_L(\alpha) = \frac{F_L(\alpha)}{\frac{1}{2}\rho u_\infty^2 \cdot LB} \quad (3.6)$$

$$C_M(\alpha) = \frac{M(\alpha)}{\frac{1}{2}\rho u_\infty^2 \cdot LB^2} \quad (3.7)$$

4 Physical Equations Describing Fluid Flow

By means of the so-called Continuity Equations and Material Equations the flow of fluids can be described completely and in every detail, including very small vortices and turbulences. The equations are known by the name Navier-Stokes Equations (see Figure 4.1).

Figure 4.1: Navier-Stokes Equations

4.1 Continuity Equations

The Continuity Equations include three governing physical laws[11] :

- I. Conservation of mass
- II. Conservation of momentum (Newton's second law)
- III. Conservation of energy (First law of Thermodynamic)

These laws have to be satisfied for each fluid element, which is part of the continuous media. Considering this requirement, the behaviour and properties of fluids can be determined with different numerical approaches for different stages of time ¹.

Conservation of Mass

To describe the first of the three mentioned laws, the law of conservation of mass, an infinitesimally small and moving fluid element of volume $V = dx \cdot dy \cdot dz$ is considered. The density ρ and the velocity $\vec{u}$ of the element is a function of the three space coordinates x, y, z and time t , which is expressed mathematically as [12]:

$$\rho = f(x, y, z, t)$$

$$\vec{u} = u, v, w = f(x, y, z, t)$$

Thus, the mass m , which is dependent on the density of the element, is a function of time as well [12]:

$$m = \rho \cdot V = \rho \cdot dx \cdot dy \cdot dz = f(x, y, z, t)$$

¹Further explanation of numerical approaches is given in Chapter 5

Table 4.1: Mass flow

Direction in coordinate system	Initial mass flow rate	Downstream stage mass flow rate
x-direction	$(\rho \cdot u) \cdot dy \cdot dz$	$\left[(\rho \cdot u) + \frac{\partial}{\partial x}(\rho \cdot u) \cdot dx \right] \cdot dy \cdot dz$
y-direction	$(\rho \cdot v) \cdot dx \cdot dz$	$\left[(\rho \cdot v) + \frac{\partial}{\partial y}(\rho \cdot v) \cdot dy \right] \cdot dx \cdot dz$
z-direction	$(\rho \cdot w) \cdot dx \cdot dy$	$\left[(\rho \cdot w) + \frac{\partial}{\partial z}(\rho \cdot w) \cdot dz \right] \cdot dx \cdot dy$

The mass flow rate of the fluid element can be described separately for each direction in the coordinate system. The separate portions are given in the following Table 4.1 according to Lecheler [12]. Additionally, the mass flow with its components in each direction is illustrated in Figure 4.2.

Figure 4.2: Mass flow after Lecheler [12]

Stating that the change of density ρ and the change of mass flow rate in the three different directions of the coordinate system over time is zero, the given formulas are combined together to a first version of the conservation of mass law.

$$\begin{aligned}
 & \frac{\partial}{\partial t}(\rho \cdot dx \cdot dy \cdot dz) \\
 & + \left[(\rho \cdot u) + \frac{\partial}{\partial x}(\rho \cdot u) \cdot dx \right] \cdot dy \cdot dz - (\rho \cdot u) \cdot dy \cdot dz \\
 & + \left[(\rho \cdot v) + \frac{\partial}{\partial y}(\rho \cdot v) \cdot dy \right] \cdot dx \cdot dz - (\rho \cdot v) \cdot dx \cdot dz \\
 & + \left[(\rho \cdot w) + \frac{\partial}{\partial z}(\rho \cdot w) \cdot dz \right] \cdot dx \cdot dy - (\rho \cdot w) \cdot dx \cdot dy \\
 & = 0
 \end{aligned}$$

By reducing the terms $(\rho \cdot u)$, $(\rho \cdot v)$, $(\rho \cdot w)$ and dividing by $dx \cdot dy \cdot dz$, the law of conservation of mass can finally be written in differential form as:

$$\frac{\partial}{\partial t}(\rho) + \frac{\partial}{\partial x}(\rho \cdot u) + \frac{\partial}{\partial y}(\rho \cdot v) + \frac{\partial}{\partial z}(\rho \cdot w) = 0 \quad (4.1)$$

Equation 4.1 can also be written in conservation form as:

$$\frac{\partial}{\partial t}(\rho) + \vec{\nabla} \cdot (\rho \cdot \vec{u}) = 0 \quad (4.2)$$

Conservation of Momentum

For the consideration of 3D-flows the law of conservation of momentum is based on three equations, one for each direction in space:

$$F_x = m \cdot a_x \quad F_y = m \cdot a_y \quad F_z = m \cdot a_z$$

Figure 4.3: Flowchart, forces on fluids after Lecheler [12]

The force consist of many components for fluids, which can be divided into the two main groups of body forces and surface forces. The components are given in Figure 4.3 in general and for an infinitesimal element in Figure 4.4 according to Lecheler [12].

Again an infinitesimal fluid element with the Volume $V = dx \cdot dy \cdot dz$ is considered in order to describe the law of conservation of momentum. The derivation of the law is given for the example of the x-component in the following paragraphs. Resume that the basic equation of Newton's second law for the x-direction is $F_x = m \cdot a_x$. The left hand side of the equation F_x is expressed in more detail as [12]:

$$F_x = \left[\frac{\partial}{\partial x}(\tau_{xx}) + \frac{\partial}{\partial y}(\tau_{yx}) + \frac{\partial}{\partial z}(\tau_{zx}) - \frac{\partial p}{\partial x} + \rho \cdot g_x \right] \cdot dx \cdot dy \cdot dz$$

The right hand side of the equation consists of the mass m and acceleration a , which is the derivative of velocity over time. The equations for mass m and acceleration a are given below [12].

$$m = \rho \cdot V = \rho \cdot dx \cdot dy \cdot dz$$

Figure 4.4: Normal and shear forces in x-direction after Lecheler [12]

$$a_x = \frac{du}{dt} = \frac{\partial u}{\partial t} + u \cdot \frac{\partial u}{\partial x} + v \cdot \frac{\partial u}{\partial y} + w \cdot \frac{\partial u}{\partial z}$$

Putting the given formulas into one equation, and reducing the volume $V = dx \cdot dy \cdot dz$ on both sides of the equation, we get the non-conservative formulation for momentum in x-direction (Equation 4.3) [12]. The equations for the momentum in y-direction (Equation 4.4) and z-direction (Equation 4.5) can be derived accordingly.

$$\frac{\partial}{\partial x}(\tau_{xx}) + \frac{\partial}{\partial y}(\tau_{yx}) + \frac{\partial}{\partial z}(\tau_{zx}) - \frac{\partial p}{\partial x} + \rho \cdot g_x = \rho \cdot \frac{\partial u}{\partial t} + \rho \cdot u \cdot \frac{\partial u}{\partial x} + \rho \cdot v \cdot \frac{\partial u}{\partial y} + \rho \cdot w \cdot \frac{\partial u}{\partial z} \quad (4.3)$$

$$\frac{\partial}{\partial x}(\tau_{xy}) + \frac{\partial}{\partial y}(\tau_{yy}) + \frac{\partial}{\partial z}(\tau_{zy}) - \frac{\partial p}{\partial y} + \rho \cdot g_y = \rho \cdot \frac{\partial v}{\partial t} + \rho \cdot u \cdot \frac{\partial v}{\partial x} + \rho \cdot v \cdot \frac{\partial v}{\partial y} + \rho \cdot w \cdot \frac{\partial v}{\partial z} \quad (4.4)$$

$$\frac{\partial}{\partial x}(\tau_{xz}) + \frac{\partial}{\partial y}(\tau_{yz}) + \frac{\partial}{\partial z}(\tau_{zz}) - \frac{\partial p}{\partial z} + \rho \cdot g_z = \rho \cdot \frac{\partial w}{\partial t} + \rho \cdot u \cdot \frac{\partial w}{\partial x} + \rho \cdot v \cdot \frac{\partial w}{\partial y} + \rho \cdot w \cdot \frac{\partial w}{\partial z} \quad (4.5)$$

In the subsequent paragraphs the mathematical procedure for gaining the conservation form is explained for the example of the conservation of momentum in x-direction. The background of this change from the so called non-conservation form to the conservation form is that the conservative form includes the total momentum even after derivation, whereas portions of the momentum get lost when deriving the non-conservation form [12]. Therefore, it is more accurate to use the conservative form of the formula when solving fluid dynamic problems.

The right hand side of the non-conservative equation 4.3 is converted to the conservative form following the mathematical laws for deriving products (explanation is given in Table 4.2). The term in brackets represents the Continuity Equation 4.1 and thus, is zero due to the law of conservation of mass.

$$\begin{aligned}
 & \rho \cdot \frac{\partial u}{\partial t} + \rho \cdot u \cdot \frac{\partial u}{\partial x} + \rho \cdot v \cdot \frac{\partial u}{\partial y} + \rho \cdot w \cdot \frac{\partial u}{\partial z} \\
 &= \frac{\partial(\rho \cdot u)}{\partial t} - u \cdot \frac{\partial \rho}{\partial t} + \frac{\partial(\rho \cdot u^2)}{\partial x} - u \cdot \frac{\partial(\rho \cdot u)}{\partial x} + \frac{\partial(\rho \cdot u \cdot v)}{\partial y} - u \cdot \frac{\partial(\rho \cdot v)}{\partial y} + \frac{\partial(\rho \cdot u \cdot w)}{\partial z} - u \cdot \frac{\partial(\rho \cdot w)}{\partial z} \\
 &= \frac{\partial(\rho \cdot u)}{\partial t} + \frac{\partial(\rho \cdot u^2)}{\partial x} + \frac{\partial(\rho \cdot u \cdot v)}{\partial y} + \frac{\partial(\rho \cdot u \cdot w)}{\partial z} - u \cdot \left[\frac{\partial \rho}{\partial t} + \frac{\partial(\rho \cdot u)}{\partial x} + \frac{\partial(\rho \cdot v)}{\partial y} + \frac{\partial(\rho \cdot w)}{\partial z} \right] \\
 &= \frac{\partial(\rho \cdot u)}{\partial t} + \frac{\partial(\rho \cdot u^2)}{\partial x} + \frac{\partial(\rho \cdot u \cdot v)}{\partial y} + \frac{\partial(\rho \cdot u \cdot w)}{\partial z}
 \end{aligned}$$

Table 4.2: Conversion of non-conservation to conservation form

Solution for differential equation	Solved for non-conservative terms
$\frac{\partial(\rho \cdot u)}{\partial t} = \rho \cdot \frac{\partial u}{\partial t} + u \cdot \frac{\partial \rho}{\partial t}$	$\rightarrow \rho \cdot \frac{\partial u}{\partial t} = \frac{\partial(\rho \cdot u)}{\partial t} - u \cdot \frac{\partial \rho}{\partial t}$
$\frac{\partial(\rho \cdot u^2)}{\partial x} = \rho \cdot u \cdot \frac{\partial u}{\partial x} + u \cdot \frac{\partial(\rho \cdot u)}{\partial x}$	$\rightarrow \rho \cdot u \cdot \frac{\partial u}{\partial x} = \frac{\partial(\rho \cdot u^2)}{\partial x} - u \cdot \frac{\partial(\rho \cdot u)}{\partial x}$
$\frac{\partial(\rho \cdot u \cdot v)}{\partial y} = \rho \cdot v \cdot \frac{\partial u}{\partial y} + u \cdot \frac{\partial(\rho \cdot v)}{\partial y}$	$\rightarrow \rho \cdot v \cdot \frac{\partial u}{\partial y} = \frac{\partial(\rho \cdot u \cdot v)}{\partial y} - u \cdot \frac{\partial(\rho \cdot v)}{\partial y}$
$\frac{\partial(\rho \cdot u \cdot w)}{\partial z} = \rho \cdot w \cdot \frac{\partial u}{\partial z} + u \cdot \frac{\partial(\rho \cdot w)}{\partial z}$	$\rightarrow \rho \cdot w \cdot \frac{\partial u}{\partial z} = \frac{\partial(\rho \cdot u \cdot w)}{\partial z} - u \cdot \frac{\partial(\rho \cdot w)}{\partial z}$

Reinsert the derived term in the former formula (non-conservation form) and rearranging the equation furthermore, the final law of conservation of momentum can be written for the x-direction in the spatial system in conservation form [13]

$$\begin{aligned}
 & \frac{\partial}{\partial t}(\rho \cdot u) + \frac{\partial}{\partial x}(\rho \cdot u^2 + p - \tau_{xx}) + \frac{\partial}{\partial y}(\rho \cdot u \cdot v - \tau_{yx}) + \frac{\partial}{\partial z}(\rho \cdot u \cdot w - \tau_{zx}) - \rho \cdot g_x = \\
 & \frac{\partial}{\partial t}(\rho \cdot u) + \nabla \cdot (\rho \cdot u \cdot \vec{u}) + \frac{\partial p}{\partial x} - \frac{\partial \tau_{xx}}{\partial x} - \frac{\partial \tau_{yx}}{\partial y} - \frac{\partial \tau_{zx}}{\partial z} - \rho \cdot g_x = 0
 \end{aligned} \tag{4.6}$$

By applying the same procedure for the y- and z- direction, the equations for the conservation of momentum can be completed.

$$\begin{aligned}
 & \frac{\partial}{\partial t}(\rho \cdot v) + \frac{\partial}{\partial x}(\rho \cdot v \cdot u - \tau_{xy}) + \frac{\partial}{\partial y}(\rho \cdot v^2 + p - \tau_{yy}) + \frac{\partial}{\partial z}(\rho \cdot v \cdot w - \tau_{zy}) - \rho \cdot g_y = \\
 & \frac{\partial}{\partial t}(\rho \cdot v) + \nabla \cdot (\rho \cdot v \cdot \vec{u}) + \frac{\partial p}{\partial y} - \frac{\partial \tau_{xy}}{\partial x} - \frac{\partial \tau_{yy}}{\partial y} - \frac{\partial \tau_{zy}}{\partial z} - \rho \cdot g_y = 0
 \end{aligned} \tag{4.7}$$

$$\begin{aligned}
 & \frac{\partial}{\partial t}(\rho \cdot w) + \frac{\partial}{\partial x}(\rho \cdot w \cdot u - \tau_{xz}) + \frac{\partial}{\partial y}(\rho \cdot w \cdot v - \tau_{yz}) + \frac{\partial}{\partial z}(\rho \cdot w^2 + p - \tau_{zz}) - \rho \cdot g_z = \\
 & \frac{\partial}{\partial t}(\rho \cdot w) + \nabla \cdot (\rho \cdot w \cdot \vec{u}) + \frac{\partial p}{\partial z} - \frac{\partial \tau_{xz}}{\partial x} - \frac{\partial \tau_{yz}}{\partial y} - \frac{\partial \tau_{zz}}{\partial z} - \rho \cdot g_z = 0
 \end{aligned} \tag{4.8}$$

Conservation of Energy

The law of conservation of energy describes the relationship of the rate of change of energy dE_{tot}/dt to the heat flux $\dot{Q}$ and the rate of working done on the volume element $\dot{W}$ [12]. For closed systems the rate of change of energy dE_{tot}/dt equals the heat flux $\dot{Q}$ plus working done on the volume element $\dot{W}$. This relationship is also known as the first law of Thermodynamics and is given in the following. Furthermore, the three terms, included in this law, will be explained in more detail in the following paragraphs.

$$\frac{dE_{\text{tot}}}{dt} = \dot{Q} + \dot{W}$$

The total energy E_{tot} includes three components: internal, kinetic and potential energy. The internal energy is calculated by $E_{\text{in}} = m \cdot e$, whereas e is the specific internal energy [12]. Kinetic energy can be determined for each spatial direction individually. The three equations are listed below. A fourth equation is given, which summarizes the first three equations by expressing u^2, v^2, w^2 as vector $\vec{u}^2$.

$$E_{\text{kin}} = \frac{1}{2} \cdot m \cdot u^2; E_{\text{kin}} = \frac{1}{2} \cdot m \cdot v^2; E_{\text{kin}} = \frac{1}{2} \cdot m \cdot w^2$$

$$E_{\text{kin}} = \frac{1}{2} \cdot m \cdot \vec{u}^2$$

The third component, potential energy $E_{\text{pot}} = m \cdot g \cdot h$ can be neglected for the case of gas [12]. Consequently, the total energy can be described as:

$$E_{\text{tot}} = E_{\text{in}} + E_{\text{kin}} = m \cdot e + \frac{1}{2} \cdot m \cdot \vec{u}^2 = \rho \cdot dx \cdot dy \cdot dz \cdot \left(e + \frac{1}{2} \cdot \vec{u}^2 \right)$$

Figure 4.5 summarizes the acting forces of heat flow and the forces of working done on the infinitesimal fluid element in x-direction graphically. Afterwards the components $\dot{Q}$ and $\dot{W}$ are explained in more detail.

Figure 4.5: Heat flow and forces due to working done on the element in x-direction after Lecheler [12]

The origin of heat transfer $\dot{Q}$ is a difference in temperature within the fluid and the effort of the fluid to achieve thermal equilibrium [12]. Heat is transferred within a fluid by collision of infinitesimal elements with each other, known as thermal conduction $\dot{q}_L$. A second phenomenon, known as thermal radiation $\dot{q}_S$, leads to a more uniform distribution of temperature by waves emitted and absorbed by the volume element. The heat flux $\dot{Q}$ is influenced by the density and volume and the thermal conductivity λ of the element. Heat transfer $\dot{Q}$ is calculated as [12]:

$$\dot{Q} = \left[\rho \cdot \dot{q}_S + \frac{\partial}{\partial x} \left(\lambda \cdot \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(\lambda \cdot \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(\lambda \cdot \frac{\partial T}{\partial z} \right) \right] \cdot dx \cdot dy \cdot dz$$

$$\text{where} \quad \dot{q}_{L,x} = -\lambda \cdot \frac{\partial T}{\partial x} \quad \dot{q}_{L,y} = -\lambda \cdot \frac{\partial T}{\partial y} \quad \dot{q}_{L,z} = -\lambda \cdot \frac{\partial T}{\partial z}$$

The rate of working done on the volume element $\dot{W}$ occurs due to body and surface forces and is calculated by multiplying the acting forces by the velocity in each spatial direction [12]. It consist out of four portions: gravitation g , pressure p , normal and shear stresses τ . The final equation for calculating the rate of working is given as:

$$\begin{aligned} \dot{W} = & \left[\rho \cdot (u \cdot g_x + v \cdot g_y + w \cdot g_z) - \frac{\partial}{\partial x} (u \cdot p) - \frac{\partial}{\partial y} (v \cdot p) - \frac{\partial}{\partial z} (w \cdot p) - \frac{\partial}{\partial x} (u \cdot \tau_{xx} + v \cdot \tau_{xy} + w \cdot \tau_{xz}) \right. \\ & \left. - \frac{\partial}{\partial y} (u \cdot \tau_{yx} + v \cdot \tau_{yy} + w \cdot \tau_{yz}) - \frac{\partial}{\partial z} (u \cdot \tau_{zx} + v \cdot \tau_{zy} + w \cdot \tau_{zz}) \right] \cdot dx \cdot dy \cdot dz \end{aligned}$$

The equations of heat transfer and rate of working are plugged into the first law of Thermodynamic and hence, the energy equation 4.9 [12] can be developed in non-conservation form after further simplifications and rearrangements.

$$\begin{aligned} & \frac{\partial}{\partial t} \left[\rho \cdot \left(e + \frac{1}{2} \cdot \vec{u}^2 \right) \right] \\ & + \frac{\partial}{\partial x} \left[\rho \cdot u \cdot \left(h + \frac{1}{2} \cdot \vec{u}^2 \right) - (u \cdot \tau_{xx} + v \cdot \tau_{xy} + w \cdot \tau_{xz}) - \lambda \cdot \frac{\partial T}{\partial x} \right] \\ & + \frac{\partial}{\partial y} \left[\rho \cdot v \cdot \left(h + \frac{1}{2} \cdot \vec{u}^2 \right) - (u \cdot \tau_{yx} + v \cdot \tau_{yy} + w \cdot \tau_{yz}) - \lambda \cdot \frac{\partial T}{\partial y} \right] \\ & + \frac{\partial}{\partial z} \left[\rho \cdot w \cdot \left(h + \frac{1}{2} \cdot \vec{u}^2 \right) - (u \cdot \tau_{zx} + v \cdot \tau_{zy} + w \cdot \tau_{zz}) - \lambda \cdot \frac{\partial T}{\partial z} \right] \\ & - \rho \cdot (u \cdot g_x + v \cdot g_y + w \cdot g_z) - \rho \cdot \dot{q}_S = 0 \end{aligned} \tag{4.9}$$

The law of conservation of energy is also given in conservative form [12].

$$\frac{\partial}{\partial t} \left[\rho \left(e + \frac{1}{2} \cdot \vec{u}^2 \right) \right] + \vec{\nabla} \cdot \left[\rho \cdot \vec{u} \cdot \left(h + \frac{1}{2} \cdot \vec{u}^2 \right) - \tau \cdot \vec{u} - \lambda \cdot \vec{\nabla} T \right] = \rho \cdot \vec{g} \cdot \vec{u} + \rho \cdot \dot{q}_S \tag{4.10}$$

4.2 Material Equations

The Material Equations consist of three physical laws[11] :

- I. Equation of state: Gas law
- II. Caloric Equation of states
- III. Relations for Newtonian fluid

Those equations, describing the fluid and the relations of normal and shear forces are used as an aid to solve the Continuity Equations. The latter include 17 unknowns but only 5 equations and therefore, additional equations are needed in order to solve the nonlinear differential equations [12]. The 17 unknown values of the Continuity Equations are listed in Table 4.3.

Table 4.3: Unknown variables of Continuity Equations

Unknown variable	Description
ρ	Fluids density
u, v, w	Velocity in three spatial coordinates
p	Pressure
e	Specific internal energy
h	Specific enthalpy
T	Temperature
τ_{xx}	Normal stresses in x-direction perpendicular to the x-constant area $dy \cdot dz$
τ_{yy}	Normal stresses in y-direction perpendicular to the y-constant area $dx \cdot dz$
τ_{zz}	Normal stresses in z-direction perpendicular to the z-constant area $dx \cdot dy$
τ_{xy}	Shear stresses in y-direction along the x-constant area $dy \cdot dz$
τ_{yx}	Shear stresses in x-direction along the y-constant area $dx \cdot dz$
τ_{xz}	Shear stresses in z-direction along the x-constant area $dy \cdot dz$
τ_{zx}	Shear stresses in x-direction along the z-constant area $dx \cdot dy$
τ_{yz}	Shear stresses in z-direction along the y-constant area $dx \cdot dz$
τ_{zy}	Shear stresses in y-direction along the z-constant area $dx \cdot dy$

The Material Equations introduce the first assumptions into the physical model of fluid flow. In particular, the assumption of an ideal gas and an Newtonian fluid. However the influence of those assumptions is negligible, or for the case of the assumption of a Newtonian fluid valid for all practical aerodynamic tasks.

The first equation of state relates pressure, density and temperature to each other. As mentioned before, the given equation is valid for an perfect gas, which is a sufficiently accurate assumption and widely applied in common software packages [11].

Gas law:

$$p = \rho \cdot R \cdot T \quad (4.11)$$

Caloric Equation of states:

$$e = c_v \cdot T \quad \text{and} \quad h = c_p \cdot T \quad (4.12)$$

Knowing the velocities in the spatial directions and the viscosity coefficient of the fluid, normal and shear stresses can be calculated for Newtonian fluids. As stress is proportional to velocity gradients for those fluids, the following equations obtained by Stokes apply [13].

Relations for Newtonian fluid:

$$\tau_{xx} = -\frac{2}{3}\mu \cdot \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) + 2 \cdot \mu \cdot \frac{\partial u}{\partial x} \quad (4.13)$$

$$\tau_{yy} = -\frac{2}{3}\mu \cdot \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) + 2 \cdot \mu \cdot \frac{\partial v}{\partial y} \quad (4.14)$$

$$\tau_{zz} = -\frac{2}{3}\mu \cdot \left(\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} \right) + 2 \cdot \mu \cdot \frac{\partial w}{\partial z} \quad (4.15)$$

$$\tau_{xy} = \mu \cdot \left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y} \right) \quad (4.16)$$

$$\tau_{xz} = \mu \cdot \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right) \quad (4.17)$$

$$\tau_{yz} = \mu \cdot \left(\frac{\partial w}{\partial y} + \frac{\partial v}{\partial z} \right) \quad (4.18)$$

$$\tau_{yx} = \tau_{xy} \quad (4.19)$$

$$\tau_{zy} = \tau_{yz} \quad (4.20)$$

$$\tau_{zy} = \tau_{yz} \quad (4.21)$$

Furthermore, physical equations like c_v , λ , and μ are not constant but vary with temperature. In different approaches, some values are assumed to be constant or calculated as functions of temperature [11].

4.3 Navier-Stokes Equations

The five main equations of conservation, introduced in chapter 4.1, are summarized in conservation form for clarity.

Conservation of Mass:

$$\frac{\partial}{\partial t}(\rho) + \vec{\nabla} \cdot (\rho \cdot \vec{u}) = 0$$

Conservation of Momentum (x-, y-, z-direction):

$$\frac{\partial}{\partial t}(\rho \cdot u) + \nabla \cdot (\rho \cdot u \cdot \vec{u}) + \frac{\partial p}{\partial x} - \frac{\partial \tau_{xx}}{\partial x} - \frac{\partial \tau_{yx}}{\partial y} - \frac{\partial \tau_{zx}}{\partial z} - \rho \cdot g_x = 0$$

$$\frac{\partial}{\partial t}(\rho \cdot v) + \nabla \cdot (\rho \cdot v \cdot \vec{u}) + \frac{\partial p}{\partial y} - \frac{\partial \tau_{xy}}{\partial x} - \frac{\partial \tau_{yy}}{\partial y} - \frac{\partial \tau_{zy}}{\partial z} - \rho \cdot g_y = 0$$

$$\frac{\partial}{\partial t}(\rho \cdot w) + \nabla \cdot (\rho \cdot w \cdot \vec{u}) + \frac{\partial p}{\partial z} - \frac{\partial \tau_{xz}}{\partial x} - \frac{\partial \tau_{yz}}{\partial y} - \frac{\partial \tau_{zz}}{\partial z} - \rho \cdot g_z = 0$$

Conservation of Energy:

$$\frac{\partial}{\partial t} \left[\rho \left(e + \frac{1}{2} \cdot \vec{u}^2 \right) \right] + \vec{\nabla} \cdot \left[\rho \cdot \vec{u} \cdot \left(h + \frac{1}{2} \cdot \vec{u}^2 \right) - \tau \cdot \vec{u} - \lambda \cdot \vec{\nabla} T \right] = \rho \cdot \vec{g} \cdot \vec{u} + \rho \cdot \dot{q}_S$$

As mentioned before, the number of unknown variables is reduced by introducing the Material Equations. The entirety of equations is known as Navier-Stokes Equations. The equations are non linear differential equations, which can be solved analytically for special cases only. Generally speaking, the system is solved by numerical analysis, consequently the solution is an approximation. Mathematical models for the numerical approach are presented in the following Chapter 5

5 Numerical Approach

As mentioned in Chapter 4, the Navier-Stokes Equations describe the flow of fluids completely, including very small vortices and fluctuations. However, this high accuracy and the requirement of very fine grid spacing lead to long computing times and high computational costs. Therefore, different numerical approaches are commonly used, which introduce simplifications to the Navier-Stokes Equation, so that the solution can be found with reduced operating costs. In order to achieve a sufficient amount of accuracy without squandering computational resources, the mathematical model needs to be chosen in dependence on the requirements of the particular problem.

Many software packages use the Reynolds-Averaged Navier-Stokes Equations (RANS). Furthermore, the Euler Equations and Potential Equations lead to shorter computing time due to the increasing number of assumptions in the order they are listed. These three common numerical models are introduced in the following section.

5.1 Overview of Numerical Models

While the computational time for flows modelled by Reynolds-Averaged Navier-Stokes Equations usually range from hours to days, the application of Euler or Potential Equations reduce the time to seconds or minutes [12]. Therefore, those equations are often used if the physical characteristics are modelled with a sufficient amount of accuracy in the investigated case. Figure 5.1 gives a summary of three common numerical models and the simplifications, which are included. Further explanation is given in the following.

Figure 5.1: Numerical models after Lecheler [12]

Averaged Navier Stokes Equations

Reynolds-Averaged Navier-Stokes Equations (RANS) are gained by time averaging the Navier-Stokes Equations. By this procedure statistically steady flow variables are calculated and therefore, the results of RANS describe a perfectly laminar flow [12]. In order to capture the fluctuations in case of a turbulent flow, turbulence models are used, which re-introduce the effect of turbulence. Essentially, these turbulence models are additional terms or equations, which are added to the momentum equations. In the words of Franke [14] "[t]he task of a turbulence model is to prescribe the turbulent fluxes or Reynolds stresses as a function of the mean flow variables." Different turbulence models are known with varying accuracy and resulting computational cost in dependence on the complexity of the model. Most of the commercial software packages like Start CCM+ or ANSYS include the numerical approach of RANS and offer different choices for modelling the fluctuations.

Euler Equations

If the Euler Equations are applied, viscosity is assumed to be zero [7] and therefore, all the terms τ given in Equations 4.13 to 4.21 for Newtonian fluids vanish. Additionally, the thermal conductivity λ is neglected and the density ρ is assumed to be constant. Due to the neglect of friction in the region of the boundary layer, small vortices are omitted and grids with larger spacing can be applied, which is the main factor in order to save computational time and costs. The Euler Equations contain 5 second order and non-linear differential equations of time and space. It includes the continuity of mass, momentum and energy. Furthermore, convection is described, whereas friction, which interferes the convection, is neglected. The model is good in the stadium of design, when a lot different geometries need to be analysed and optimized for wind loads. Sufficient accuracy is also achieved for turbulent flows, high Reynolds Numbers respectively, without flow separation [12]. The software VXFlo, which is used for this work, is based on the Euler Equations. More details about the code are given in Section 5.2.

Laplace's Equation

The Laplace's Equations describe the potential flow and therefore are a further simplification of the Euler Equations. Thus the fluid is also inviscid, the thermal conductivity λ is neglected and the density ρ is assumed to be constant. Additionally, the potential flow is characterized as irrotational, which distinguishes between Euler and Laplace's Equation. Hence, the number of non-linear differential equations is reduced to 1 second order equation of time and space. The continuity of mass and energy is included, but in contrast the continuity of momentum is neglected. Therefore, an essential requirement for the application of the Laplace's Equation is that shock waves are not expected in the flow, because they cannot be represented by the Laplace's Equation. A shock wave can be explained as a sudden and discontinuous change of flow conditions like pressure, velocity, density and temperature, which can occur if the velocity exceeds the speed of sound, known as supersonic flow, only. Therefore, this numerical model is applied for transonic flow and subsonic flow only and the additional constraint is not of concern for civil engineering projects.

5.2 Vortex Particle Method in VXFlow

The software VXFlow, which is used for this work, is based on the Vortex Particle Method. The method gained wider recognition, especially for 2-dimensional cases, in the recent years due to the reduced computational costs in comparison to other methods. One of the advantages of the classical Vortex Particle Method is that the computational domain is grid free, which excludes problems of body fitting, when it comes to generating a grid [15]. Furthermore, the computational domain does not have boundaries defined as boundary conditions as it is imposed in other methods like the popular Finite Volume Method. Therefore, the Vortex Particle Method represents the free-space boundary condition, which is likely to cause problems when implemented in grid-based methods. The software VXFlow is particularly developed for the computation of two-dimensional problems in the field of Bridge Engineering. Static and dynamic investigations are supported by the software.

Method of Description

The Vortex Particle Method is based on the Lagrangian description of fluid flows [15]. In this case the observation is fixed to a specific fluid particle, whose movement and therefore, its velocity and acceleration is tracked. Figure 5.2 visualises the Lagrangian idea. Two particles P and Q and their paths are illustrated. It can be observed that the location x_i, y_i, z_i of a particle P at the time t_i depends upon the initial position x_0, y_0, z_0 of a particle at the time t_0 . For completeness Euler is mentioned as the alternative description of fluid flows. In this case, the location is fixed and it is observed what happens at that specific spatial location [7]. Euler as a method of description of the flows is not to be confused with the Euler Equations explained in Section 5.1, which is a way of expressing the conservation laws.

Figure 5.2: Visualisation of Lagrangian description after Granger [7]

Laws of Conservation and Simplifications

The Navier-Stokes Equations, derived in Chapter 4 are the base for the Vortex Particle Method, as it is common for CFD packages.

The simplifications, which lead to a faster solution of the equations, are listed in the following paragraph as stated by Morgenthal [15]. According to the simplifications, the third of the Navier-Stokes Equations 4.9, conservation of energy is not applied and the equations of the conservation of mass 4.1 and of the conservation of momentum 4.6 to 4.8 remain as the governing equations.

- I. Constant density ρ of fluid (incompressibility of fluid)
- II. Constant viscosity ν
- III. Single-phase, homogeneous fluid
- IV. High heat capacity (the change of temperature is considered to have a negligible impact)

Instead of solving for the pressure and the velocity, it has been proofed to be of value to introduce the vorticity w . Adapting the equation for the conservation of momentum in this manner, a transport equation can be derived. It is given for the case of a 2D study in Equation 5.1. For inviscid cases the equation can be simplified to Equation 5.2, which states that the change of vorticity over time is zero.

$$\frac{\partial w}{\partial t} + (u \cdot \nabla)w = \nu \nabla^2 w \quad (5.1)$$

$$\frac{Dw}{Dt} = 0 \quad (5.2)$$

Velocity Field

The velocity vector is described by the two components stream function ψ and the velocity potential ϕ [15] given in Equation 5.3. The later one usually represents the free stream velocity u_∞ , which basically leads to the movement of particles in the direction of the flow.

$$u = \nabla \times \psi + \nabla \phi = \nabla \times \psi + u_\infty \quad (5.3)$$

The boundary condition stipulate that the component of the stream function ψ_B is zero, as well as the gradient of the stream function $\nabla \psi_B$ times the normal vector n_0 is zero at the surface of objects. This is expressed in the following equations.

$$\psi_B = 0 \quad (5.4)$$

$$\nabla \psi_B \cdot n_0 = 0 \quad (5.5)$$

Diffusion and Vorticity

Originally the assumptions which are applied on the Vortex Particle Method simplify the Navier-Stokes Equations to the Euler Equations for inviscid flows. To include diffusive effects of viscous flows, the Random Walk Method by Chorin has been applied in VXFlow [15]. This approach is also illustrated in the flow chart of Figure 5.3.

Basically, the Random Walk Method, based on the Brownian Motion, changes the position of the particle by a random distance, which is normally distributed with mean value of zero ($\mu = 0$) and standard deviation of $\sigma = 2\nu t$. The vorticity on a solid boundary is diffused in the same manner applying a random value in the normal direction of half a Gaussian distribution [15]. Convergence to the Navier-Stokes Equations has been found after applying the diffusive models and manipulating the vorticity field in domains without boundaries.

Figure 5.3: Numerical model of VXFlow

Release of Vortex Particles

The bodies surface is implemented as a closed polygonal, divided into panels of given lengths Δs_i . In the middle of each panel a so-called control and sampling point is placed. At this point particles, carrying vorticity, are released, pressure and suction values are given out and the velocity boundary condition is applied [15]. The velocity boundary condition imposes that the velocity at the boundary is zero: $u_B = 0$. The induced velocity on the surface is converted to the surface vorticity, which is released in form of vortex particles. The particles of one panel, characterized by linear varying vorticity strength along the panel, are released evenly distributed along the panel from the bodies surface and in a release distance. This distance between the surface of the body and the release point is obtained by the Random Walk Method [15]. If the release point is not feasible because it is located inside the body, the procedure is repeated until a feasible solution is found. The number of particles is determined by limiting the vorticity, which can be carried by a single vortex.

Figure 5.4 gives an illustration of the described mechanism, which is implemented in VXFlow. Whereas γ_0 is the initial surface vorticity induced by the velocity on the surface and variable N_s gives the number of vortex particles, which is released from one panel. Γ_j is the vortex strength. Additionally, the vortex sheet is illustrated, showing the applied linear discretisation of the surface vorticity along the panel.

Figure 5.4: Release of vortex particles after Morgenthal [15]

Vortex Interaction

Although originally the Vortex Particle Method is grid free, a regular grid is established in the background of the domain in order to reduce the number of components when considering the interaction between vortices. Instead of computing the influence of single vortex particles on each other, the interaction between cells is calculated. The vortex strengths of all particles located in an observed cell are summarized and replaced by a single compensational vortex in the center of the cell [15]. This kind of implementation reduces the operation count and therefore, the computational time and cost. Figure 5.5 illustrates the described cell-to-cell scheme.

Figure 5.5: Computation of interaction by cell-to-cell scheme after Morgenthal [15]

The interaction of vortices, more precisely the interaction of panels, on each other is described by a influence matrix M . The matrix is given for a single body interfering the flow according to Morgenthal [15], whereas $m_{i,j}$ is the normal component of the velocity induced at the i -th panel by the j -th panel. The computation of the individual components $m_{i,j}$ and the extensions for multiple bodies can be read in Section 4.3.1 of the Ph.D. Thesis of Morgenthal [15].

$$M = \begin{bmatrix} m_{1,1} & \cdots & m_{1,j} & \cdots & m_{1,N_l} \\ \vdots & \ddots & \vdots & \ddots & \vdots \\ m_{i,1} & \cdots & m_{i,j} & \cdots & m_{i,N_l} \\ \vdots & \ddots & \vdots & \ddots & \vdots \\ m_{N_l,1} & \cdots & m_{N_l,j} & \cdots & m_{N_l,N_l} \\ \Delta s_1 & \cdots & \Delta s_i & \cdots & \Delta s_{N_l} \end{bmatrix} \quad (5.6)$$

Using this influence matrix and considering the free stream velocity and free vortices (vector b), the surface velocity is solved. This is expressed in Equation 5.7:

$$M\gamma_0 = b \quad (5.7)$$

The influence between vortices and the surface is described similarly by a influence matrix. The influence of a panel on the velocity of certain vortex V_i is computed. For details of the calculation please refer to Section 4.3.3 of the Ph.D. Thesis by Morgenthal [15]. Calculating the velocity interaction of a vortex in the mentioned way is computationally expensive and thus, the method is only applied to a distance of 15 panel lengths from the surface. Beyond this distance a less complex approximation is used by summarizing the mutual vortex interactions directly.

Vortex Deletion

The number of vortices influences the computational time directly and therefore, reducing the number of vortices without influencing the accuracy of the solution significantly, is a goal to be aimed for. Vortex particles are removed for three different cases, which will be explained in the following.

First, boundary conditions are implemented to prevent that vortices penetrate the inside of the solid body or cross the section [15]. This condition is called no-penetration boundary condition and leads to the deletion of the particles in case of non fulfilment of the constraint. Secondly, vortex particles are deleted if they travel outside the domain [15]. At a certain distance, the influence of a vortex on the flow around the body gets negligible. Therefore, the dimension of the domain has to be chosen in a sufficient size, so that influencing vortices are not removed but computational time is saved due to the exclusion of unnecessary vortex particles. However, due to the conservation of circulation the loss of circulation would be recreated erroneously. In order to prevent the recreation at the surface of the body, the vorticity loss is saved to a global variable, which compensates the loss in the equation of conservation of circulation.

And third, vortex deletion is executed if two or more vortices are merged due to a small distance between each other [15]. The strength of the vortices is assigned to a new vortex, while the original ones are deleted. If the merging distance Δ_m is chosen carefully and the position of the new vortex is derived by weighting the strength of the original vortices, enough accuracy can be obtained. Especially close to the surface the

merging distance should be chosen smaller, due to the need of a certain vortex density, but it can increase with increasing distance. Thus, in VXFFlow a polynomial function is introduced to calculate the merging distance. It is calculated by $\Delta_m = a_0 r^2 + a_1 r + a_2$, where r is the distance between the closest surface panel and the vortex [15]. Figure 5.6 gives an illustration for the calculation of the merging distance in case of two bodies. As the merging distance is chosen to be the minimum of all computed merging distances $\Delta_{m,i}$ it is $\Delta_{m,2}$ for the example of the two rounded sections.

Figure 5.6: Variation of merging distance after Morgenthal [15]

6 The VXFlow Package

6.1 Input

An input file is generated for running the program VXFlow. Essential parameters are the fluid properties, flow direction, definition of the bodies geometry and the properties of the domain. Furthermore, settings are provided, which determine the extend of the output. The most important parameters are given and explained in Table 6.1.

Table 6.1: Input parameters VXFlow after Morgenthal [15]

INPUT	PARAMETER (DEFAULT)	DESCRIPTION
U0X	(1.0)	Onset flow velocity in x-direction [m/s]
DENSTY	(1.2)	Density of the fluid [kg/m^3]
VISCTY	(0.000015)	Kinematic viscosity of the fluid [m^2/s]
DIFFU	(1)	Diffusion simulation by means of random walks is only performed if DIFFU=1 (otherwise viscosity value is neglected)
FVFL	()	Absolute velocities on the isoline grid
FVSTEP	()	Timesteps for output of absolute velocities on the isoline grid
VXFL	()	Vortex velocity output (vortex position and velocity)
VXSTEP	()	Timesteps for output of vortex velocity
NSTEPS	(10000)	number of timesteps to be performed; if NSTEPS=0 then calculation is performed until user interruption
NPANELFACT	(1.0)	Factor applied to the number of panels given in the section geometry part. Use this to scale the panel numbers to achieve higher resolution.
WINMINX	(-1.0)	Graphics output window: Coordinate space to be covered by the particle output window - usually of ratio 2:1
WINMAXX	(1.0)	
WINMINY	(-1.0)	
WINMAXY	(1.0)	
ISOMINX	(-1.0)	The dimensions of the isogrid - on the nodes of this grid the properties of the flow are computed and dumped to the respective file if desired.
ISOMAXX	(1.0)	
ISOMINY	(-1.0)	
ISOMAXY	(1.0)	
ISODIMX	(100)	The number of grid lines in the isogrid. The grid spacing is determined by this variables.
ISODIMY	(100)	

GRIDMINX	(-1.0)	Minimum x-coordinate
GRIDMAXX	(1.0)	Maximum x-coordinate
GRIDMINY	(-1.0)	Minimum y-coordinate
GRIDMAXY	(1)	Maximum y-coordinate
GRIDNX	(255)	Number of cells in x-direction (note, that for the use of the P3M algorithm FFTs are used to solve the Poisson equation on this grid and these are fastest for dimensions of powers of 2. Therefore, the number of cells (being one less than the number of nodes) are recommended to be "powers of 2 minus 1", e.g. 31, 63, 127 etc.
GRIDNY	(255)	Number of cells in y-direction; the same as for GRIDNX applies
NUMSEC	(1)	Number of individual cross sections (separate bodies)
ASECn	()	Area of individual cross section n+1
GEOMROT	(0.0)	Rotates the section by the given angle [deg] around the origin of the coordinate system. Positive angles are clockwise.

Here follow num.cornerpoints lines with the following format, providing information on how to discretise the surface of the section, running in an anti-clockwise manner around the section. The first point given is the leading 'edge' (corner) and the number of panels given is inserted on the edge running from this point to the one in the next line. The last line contains the point on the circumference and panel information for the edge which closes the section. Rows look like: ID CX CY N MX MY

ID	Edge shape identifier: (1)=linear edge; (2)=arc edge				
CX	X-coordinate of corner point				
CY	Y-coordinate of corner point				
N	Number of panels to be created on this edge				
MX	X-coordinate of the centre of the circle part of which this edge is. MX and MY are only provided for arc edges, i.e. ID=2. The edge is constructed through the first point and around the center given.				
MY	Y-coordinate of centre, cf. also MX				

Example:

would be achieved by the following sequence:

```

1 p1x p1y N1
1 p2x p2y N2
2 p3x p3y N3 Mx3 My3
1 p4x p4y N4

```

The default values for density and kinematic viscosity of air is assumed for the regular calculations. The values are defined similarly by Potter and Wiggert [7]: $\rho = 1.204 \text{ kg/m}^3$ and $\nu = 1.51 \cdot 10^{-5} \text{ m}^2/\text{s}$ as properties of air at atmospheric pressure and 20°C .

Instead of changing the direction of the incoming wind, the body is turned around its center point. Thus, the flow is horizontal ($U_{0Y} = 0$) for all angles, equal to the common test setup in a wind tunnel. The parameter to rotate the section is called GEOMROT. Furthermore, it is emphasized that the choice of the dimensions of the computational domain is essential for the reliability of the results. If the domain size is chosen too generously, the computing time and size of the output files is increased needlessly. But, the results can be falsified if the limits of the domain are too close to the section due to the interaction between adjacent vortex particles. The most disturbance of the flow, meaning the change of mean velocities, mean pressures and fluctuations, are obtained downstream of the section but an impact is also perceptible upstream of the body. The three different parameters WIND, ISO and GRID, which determine the domain size, are introduced in Table 6.1 and explained in the following.

WIN determines the size of the graphic output window. The size of the output files can be reduced by choosing a smaller output window but the results are not affected due to the disregard of the influence of outer regions. The ISO commands determine the domain in which the properties of the flow are computed and saved in the output files. GRID gives the dimensions of the largest region. Vortex particles which travel out of this region are deleted as described in Section 5.2. The choice of the GRID size is essential in order to prevent the distortion of the results, which is similar to the blockage effect in a wind tunnel. Recommendation for the choice of the domain size GRID are given in Figure 6.1. The domain size is chosen based on the largest dimensions B of the structure. In case of a bridge deck the width is typically decisive, shown in Figure 6.1. Depending on the aspect ratio the section height can be the controlling parameter for other sections (e.g. vertical rectangle).

Figure 6.1: Computational Domain in VXFlow

A template input file is given in Appendix A including a bridge deck and vehicles on it. In the example the dimensions for the WIN, ISO and GRID domain are chosen to be the same, which ensures the accessibility of the data for the entire domain, although it increases the computational cost.

6.2 Output and Post-processing

The output of VXFlow is given in form of text files. The most important files are given in Table 6.2 with a short description.

Table 6.2: Output files VXFlow after Morgenthal [15]

EXTENSION	DESCRIPTION
.in13	Input file (see Section 6.1)
.o3	Forces output file
.pr1	Pressures output file
.vx2	Vortex velocity output file (vortex position and velocity for every time step)
.fv2	Absolute velocities output file on the isoline grid
.hdr	Geometry output file

The post-processing of the data can be done with a Matlab script, which is provided by Prof. Morgenthal. The script was used to double check the results for pressure and force coefficients calculated with a separate Excel sheet. Furthermore, the code was applied to generate the plots to illustrate the geometries of the investigated sections and the velocity fields. The calculations for pressures and forces were done with Excel due to the additional flexibility for the calculations. For example, in the case of traffic on the bridge it was possible to calculate the forces on the bridge and on the vehicles separately, whereas VXPost is designed to give the force coefficients for the sum of all sections.

6.3 Pressure Distribution and Coefficients

The distribution of pressures on the surface of the body is calculated by VXFlow. The pressures are given for every panel and time step in form of a text file (filename.pr1). Other than usual, the calculated pressures are not absolute values but related to the pressure on the first generated panel of each section. Therefore, the qualitative pressures values are dependent on the definition of the geometry. The actual pressure for panel Nr.1 is set to zero and thus, is unknown. As a result, it is not possible to calculate the absolute values of the pressures on every panel of the contour of the investigated cross section. Moreover, the given pressure coefficients of VXFlow $C_{p,VX}$ do not follow the common definition of C_p values (Equation 3.4), for which the pressures are related to the pressure in the free flow instead of an arbitrary one. As a result, and especially for the case of a negative reference pressure, $C_{p,VX}$ values higher than 1.0 are obtained to a great extent. Furthermore, the index of the coefficient $C_{p,VX}$ does not indicate pressure or suction.

The difference between the two approaches is expressed in the following equations:

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho u_\infty^2} \quad \neq \quad C_{p,VX} = \frac{p - p_{ref_i}}{\frac{1}{2}\rho u_\infty^2}$$

where

- p = Pressure on body surface
- p_{ref_i} = Pressure on the first generated panel
- p_∞ = Free stream pressure
- u_∞ = Free stream velocity

Using the example of a cylinder with diameter of 2 m, the correlation between C_p and $C_{p,VX}$ was investigated. Velocity was chosen to be $u = 15\text{m/s}$ in x-direction and the density and viscosity of air was set by default ($\rho = 1.2\text{kg/m}^3$, $\nu = 1.5 \cdot 10^{-5}\text{m}^2/\text{s}$). The center point is located at the origin of ordinates $P(0|0)$. The geometry is given in Figure 6.2.

Figure 6.2: Geometry of cylinder

The first calculation was run for the cylinder with the first coordinate chosen to be $P(1|0)$. Therefore, the pressure and the pressure coefficient at this location, which is defined as panel 1, is zero, which can be obtained in the left plot of Figure 6.3. Negative pressures (suction) are plotted outside of the section geometry and positive pressures are drawn inwards of the cylinder vice versa. The results of VXFlow are compared with the pressure distribution stipulated by the Eurocode EN 1991-1-4 for circular cylinders of infinite length, which is illustrated as a grey line in Figure 6.3. It can be seen, that the pressure coefficients C_p given by the Eurocode differ from the coefficients $C_{p,VX}$, which are obtained by VXFlow.

Most of the coefficients $C_{p,VX}$ are positive, while due to the dominance of suction C_p values are mainly negative. Because the reference pressure is unknown, the assumption that the maximum value of the pressure coefficient is 1.0 (see Section 3.3) is used to scale the coefficients. A particular addend is determined for each time step individually and applied against every coefficient $C_{p,VX}$ of that time step with the aim

that the maximum of the pressure coefficients is 1.0. The scaled coefficients of every time step are averaged to mean values, which are illustrated on the right side of Figure 6.3. The distribution of pressures is more similar to the coefficients given by the Eurocode and the prediction of pressure and suction matches after the adjustment. Equal values are recognized on the windward side, however deviations are still observed on the leeward side of the cylinder. These deviations are explained by the shape of the section - for geometries characterized by sharp edges, which are common for bridge decks, the computation of static wind loads is predicted to be more robust. Another factor is the choice of the panel length, which limits the possibility of resolving turbulent boundary layers in a higher detail. The issue is discussed more deeply in Section 6.4. Table 6.3 gives the pressure coefficients in steps of 30° , where 0° is defined to be location $P(1|0)$ with the angle increasing counter-clockwise.

Table 6.3: Comparison - Pressure coefficients of cylinder 1

Angle	0°	30°	60°	90°	120°	150°	180°
VXFlow	0.02	0.49	0.51	-0.36	0.43	2.09	2.85
VXFlow scaled	-1.82	-1.40	-1.32	-2.19	-1.58	0.13	0.98
CODE EN 1991-1-4	-0.70	-0.70	-0.70	-1.85	-1.4	0.1	1

Figure 6.3: Pressure coefficients of cylinder 1

If a location is known where the pressure coefficient is zero, this location can be chosen as starting location, which makes scaling the coefficients unnecessary. This is solved for the circle presented of the previous example. Table 6.4 and Figure 6.4 show that the direct output does not vary a lot from the scaled coefficients for this choice of starting panel. The maximum averaged coefficient observed is 1.03. The coefficient is slightly higher than 1, which is result of the variation of the location of the zero point over the computational time.

Figure 6.4: Pressure distribution of cylinder 2

Table 6.4: Pressure coefficients of cylinder 2

Angle	0°	30°	60°	90°	120°	150°	180°
VXFlow	-1.76	-1.21	-1.15	-2.08	-1.38	0.26	1.03
VXFlow scaled	-1.86	-1.31	-1.26	-2.18	-1.49	0.15	0.92
CODE EN 1991-1-4	-0.70	-0.70	-0.70	-1.85	-1.4	0.1	1

Another test is carried out to verify that each section in the computational domain has its individual reference pressure $p_{ref,i}$. The chosen set up, which contains 2 circles is illustrated in Figure 6.5. The distance of the two cylinders is chosen to the length of twice the diameter so that the interaction is expected to be of minor influence. The geometry and dimensions are given in Figure 6.5. For the upper circle panel 1 starts at coordinate $P(0.25|0.75)$ and for the lower circle it is $P(-0.25|-0.75)$. The diameter changed to 0.5 m but the remaining variables are kept the same as in the previous example ($v = 15m/s$, $\rho = 1.2kg/m^3$, $\nu = 1.5 \cdot 10^{-5}m^2/s$).

Figure 6.5: Geometry and pressure coefficients of cylinder collocation

The resulting coefficients are illustrated on the right side of Figure 6.5, whereas the solid line represents the pressure coefficients given by VXFlow $C_{p,VX}$ and the dashed line shows the coefficients, following the criteria that the maximum coefficient is 1.0. From the figures it can be obtained that quantitatively the original coefficients $C_{p,VX}$ are not similar for the two circles but the distribution is the same, which can be deduced from the similarity of the scaled pressure coefficients. Hence, the different starting points lead to different reference pressures. In the present case e.g. the average addend is negative for the top circle (-1.83) and positive for the circle at the bottom (+0.99).

In conclusion it can be said, that the pressure output includes an unknown reference pressure, which also varies for every time step. The pressure coefficients $C_{p,VX}$ calculated with these pressures, which is automatically done by VXPost, are dependent on the definition of the geometry, in particular on the definition of the first panel. In order to get meaningful coefficients and achieve comparability with the standardized coefficients C_p , the pressure at one location needs to be known at least. Most likely this is the case for a stagnation point, where $C_p = 1.0$. However, the results are an approach to the coefficients and need to be handled with care. Additionally, for more specific geometries it is increasingly difficult to locate stagnation

points and for sections which are located in the wake of another body it is almost impossible to forecast the maximum pressure coefficient. As an example a bridge deck is named, which consists of two separate, parallel girders.

6.4 Force Coefficients

Analogous to the pressures and pressure coefficients, VXFlow gives a force output (filename.o3) and force coefficients can be estimated with VXPost, Excel or similar tools. The calculations of forces is based on the pressures, whereas the choice of reference pressure does not have an influence, which is specified in the following paragraphs.

Figure 6.6: Pressure on closed body due to constant pressure

A force is calculated by the pressure multiplied by its area of influence. Therefore, the total force on a body is determined by the integral of pressure over the surface area of the object under consideration of the orientation of pressure, which is perpendicular to the surface. The relation is expressed in the next equation, which is further simplified for constant pressure:

$$F = \sum_{i=0}^k p_i \cdot \vec{n}_i \cdot A_i = \oint p \cdot \vec{n} dA = p \cdot \oint \vec{n} dA \quad (6.1)$$

The vector $\vec{n}$ describes the orientation of the pressure and is the normal to the surface. By definition the sum of all normals around a closed surface is zero [16]. In other words every infinitesimal element of the surface has an opponent element, whose normal vector points in the opposite direction:

$$\sum_{i=0}^k n_i \cdot A_i = \oint \vec{n} dA = 0 \quad (6.2)$$

The equations can be applied on two-dimensional or three-dimensional bodies, which have a closed surface and are exposed to constant pressure. Considering that the force on a body is zero for constant pressures, regardless of the value, it is obvious that the choice of reference pressure p_{ref} does not have any influence on the calculation of the forces and force coefficients.

For validating VXFlow in terms of the force coefficients the circular cylinder is investigated and the results of VXFlow are compared with values given in literature by Tritton [17]. Similar values are given by Tietjens [18] for the same range of Reynolds Numbers or in simplified form in the Eurocode EN 1991-1-4 [19] for Reynolds Numbers between 10^5 and 10^7 . Mean values of the drag coefficient are compared for different Reynolds Numbers ranging from 10^3 to 10^7 . The results are illustrated in Figure 6.7.

Figure 6.7: Comparison of Drag Coefficients

In general, it can be said that the drag coefficient is overestimated by VXFlow. This result is supported by the comparison of the pressure distribution in Section 6.3, where increased suction was observed on the leeward side of the circular cylinder. This overestimation of suction leads to increased drag coefficients calculated by integration of the pressures. Higher deviations occur for Reynolds Numbers above $3 \cdot 10^5$, as the effect of decreasing drag forces for increasing velocities is not simulated by the calculations carried out with the VXFlow code. The reason for the drop of the drag force for Reynolds Numbers around $3 \cdot 10^5$ is the change from a laminar boundary layer to a turbulent one. As a consequence, the separation point is located further downstream on the surface of the cylinder, the wake is narrower and the drag force drops [17]. The choice of the number of panels, which is controlled by the input of the geometry directly or by the parameter NPANELFACT (see Table 6.1) influences the resolution of turbulent characteristics of the boundary layer. With the choice of a very small panel length it is expected that the turbulences of the boundary layer can be resolved in more detail so that the value in literature can be reproduced by VXFlow. The necessary decrease of panel length leads to an increase in computational time and therefore, the number of panels is restricted due to applicability. In the case of the investigated cylinder, the panel length is chosen to 0.03m for a diameter of 2m. Respectively, the perimeter is approximately 12.6m subdivided into 400 panels.

The dependence of the drag coefficient on Reynolds Numbers is determining for rounded and streamlined sections. In contrast, the drag force is more stable for bodies characterized by sharp edges [20]. For rounded shapes the location of the separation point and therefore, the width of the wake is dependent on the characteristics of the boundary layer. Unlike, for bodies with defined edges the separation point does not change depending on a laminar or turbulent boundary layer, but occurs at the protruding edge. Thus, the drag force whose quantity is directly influenced by the width of the wake, is similar for a wide range of Reynolds numbers. Therefore, the prediction of static force coefficients by VXFlow is more accurate for geometries with sharp edges than for streamlined and rounded shapes.

To confirm the assumption that higher accuracy is gained for geometries with sharp edges, the drag coefficient for rectangular sections of different aspect ratios B/H is studied. The Eurocode EN 1991-1-4 [19]

gives reference values for the drag coefficient of rectangular sections with different aspect ratios. Further reference values are obtained experimentally and numerically by Sadraey [21], Yu and Kareem [22] and other authors. A number of results is summarized in the Ph.D. Thesis of Liaw [23]. Good agreement is found between the Eurocode and the experimental and numerical studies found in literature for aspect ratios of $B/H \leq 0.5$ and $B/H \geq 1$. However, the peak of drag coefficients C_D up to a value of 3.0, which is obtained by studies for aspect ratios around $B/H = 0.7$, is not represented in the Eurocode. For the current study experimental or numerical data are not available for rectangular sections characterized by high aspect ratios thus both, the data of the Eurocode and the results of studies is used for comparison with the data of VXFFlow.

Figure 6.8 ² gives an illustration of the comparison between the results of VXFFlow and the given C_D -values of the Eurocode [19] for aspect ratios $0.2 \leq B/H \leq 40$. Additionally the values are given in Table 6.5. Good agreement to the Eurocode is obtained for high aspect ratios of 5, 10 and 40. Higher deviations to the Eurocode are observed for sections with ratios B/H smaller than 1.0 and the square section (aspect ratio $B/H = 1.0$). However, as mentioned before, a large number of studies is available regarding the drag coefficient of rectangular sections in this range of aspect ratios and differences between these studies and the Eurocode are observed. Because the Eurocode is developed for designing structures, it frequently includes simplifications and generalisations. Therefore, the results of the experimental and numerical studies are seen to give more accurate reference for the comparison.

The comparison of drag coefficients for aspect ratios $0 \leq B/H \leq 5$ is illustrated in Figure 6.9 ³. Additional to the data from experimental and numerical works, the values stipulated by the Eurocode and the results of VXFFlow are included in the figure. Thus, the deviation between the studies and the Eurocode can be observed also in the figure. The difference between the C_D values of the Eurocode and experimental data is considerably variable for aspect ratios around $B/H = 0.7$ ($C_D = 2.4$ stipulated by the Eurocode, $C_D = 3.0$ experimental results). Hence, the coefficient of $C_D = 3.0$, which is gained by the CFD analysis with VXFFlow, is similar to the results of experimental studies e.g. of Bearman and Trueman, Courchesne and Lancville, whose data is reviewed in the Ph.D. Thesis of Liaw [23]. Higher differences are obtained for the aspect ratios of $B/H = 0.2$ and $B/H = 1.0$.

Table 6.5: Drag coefficients of rectangular sections

Aspect ratio	0.2	0.7	1	5	10	40
VXFFlow	2.87	3.00	2.69	1.10	0.96	1.14
CODE EN 1991-1-4	2.00	2.40	2.10	1.00	0.90	0.90

²Note that the axis for the aspect ratio B/H is set to a logarithmic scale

³Note that the axis for the aspect ratio B/H is set to a linear scale

Figure 6.8: Rectangular sections - Comparison of Drag Coefficients (a)

Figure 6.9: Rectangular sections - Comparison of Drag Coefficients (b)

Depending on the aspect ratio, the drag force of bodies consists of form drag and skin friction drag in varying portions [21]. For flat plates with high aspect ratios which are placed parallel to the flow direction, the skin friction drag is determining. It is the result of the viscous shear stresses between the body surface and the air flowing by and acts parallel to the surface of the section. By contrast, the form drag is result of unbalanced pressures along the body and depends on the projected frontal area of the body blocking the flow. The force which acts on the body in case of form drag is normal to the body surface. An example for which form

drag is predominate, is a flat plate perpendicular to the flow direction. As the results of VXFFlow seem to be more accurate for bodies with high aspect ratios, it is assumed that the effect of skin friction is reproduced more precise than form drag. The analysis of the circular cylinder which mostly creates form drag showed discrepancies in the pressure distributions in Section 6.3 as well. The investigation of further similar aspect ratios is recommended in further studies to give a more profound statement regarding the deviations for different aspect ratios.

Aspect ratios of bridge decks are usually higher than $B/H = 2$ for which good agreement could be found for the rectangular sections. Exemplary, the section of the Süderelbe Bridge in Hamburg-Moorburg which is investigated in more detail in the current study is characterized by an aspect ratio of approximately $B/H = 7.5$ with wind barriers, and $B/H = 14$ without considering the wind barrier height. The investigation of the rectangular section showed good agreement with the Eurocode in this range of aspect ratios.

In summary, the results of VXFFlow for bodies with sharp edges are reasonable and have a higher accuracy as the coefficients obtained for the circular cylinder because the problem of localizing the separation point computationally vanishes for these geometries. In usual applications in civil engineering sections include sharp edges and hence, resolution of turbulences in the boundary layer is a subordinate issue. Problems can occur though for bridge sections, which include rounded parts, like rounded box girders as installed in cable-stayed bridges frequently. Generally the drag coefficient is overestimated as observed for the circle and the rectangular sections. For civil engineering purposes the general overestimation of the drag force won't cause any structural disasters but can lead to uneconomical design if the differences increase a certain threshold.

7 Comparison of Numerical and Experimental Analysis

Figure 7.1: Visualisation of models T1, T2, T3

A rigid sectional model is tested in the wind tunnel of Ryerson University in Toronto. The design for the models is chosen on the basis of the design of the Süderelbe Bridge. Withal, three different cross sections are analysed to compare the results of the wind tunnel tests with the results of VXFlo. The discrepancy between the data depending on the simplicity of the object is studied. The first Model T1 is most similar to the actual bridge design, as the two hollow steel boxes are connected by cross beams every 12m, and the wind barrier is also represented. Simplifications are applied on this model, for example the open cross beams at the edges are not represented due to limitations of assembling. The simplifications included in the models for the wind tunnel are rendered in VXFlo as well to ensure comparability. The wind barrier is removed for Model T2, while the light gap is still represented. Model T3 distinguishes between the second cross section due to the closed light gap. A three-dimensional illustration of the models for the wind tunnel and two-dimensional one for VXFlo is given in Figure 7.1.

The analysis includes the comparison of the drag and the lift coefficient. Table 7.1 gives an overview of the values, which are used to calculate the force coefficients. In order to ensure the comparability between the different models, the coefficients need to be related to the same width B , height H and length L . Thus, the height is chosen to $H = 2.9m$ (dimension from bottom of hollow steel boxes to top of it) for every model, even though the height is different for Model T1. While the density, viscosity and velocity for calculation of the VXFlo coefficients are stipulated by the settings in the input file, the parameters vary slightly for every test running in the wind tunnel and are measured in the laboratory.

Table 7.1: Reference values for computing coefficients

PARAMETER		VALUE (Wind tunnel)		VALUE (VXFlow)	
B	Section width	40.2015	m	40.2015	m
H	Section height	2.9	m	2.9	m
L	Section length	1.0	m	1.0	m
ρ	Density of air	- varies -		1.2	kg/m^3
ν	Viscosity of air	- varies -		$1.5 \cdot 10^{-5}$	m^2/s
u_∞	Free stream velocity	- varies -		15	m/s

$$C_L = \frac{F_L}{\frac{1}{2} \rho u_\infty^2 \cdot L B} \quad C_D = \frac{F_D}{\frac{1}{2} \rho u_\infty^2 \cdot L H} \quad C_M = \frac{M}{\frac{1}{2} \rho u_\infty^2 \cdot L B^2}$$

7.1 Numerical Analysis

The numerical analysis is carried out using the tool VXFlow. The dimensions are chosen to full size and test speed is $u = 15m/s$, density and viscosity are set by default. The values have been given in Table 7.1 before. Nine different angles of attack, from -10° to 10° in steps of 2.5° , are studied in separate runnings for each model. Each running includes 10,000 time steps to provide a adequate sample time and to capture the effects of the altering wind flow. The number of vortex particles is not sufficient to present the flow characteristics reliably in the first time steps thus, the first 1,000 time steps of each running for the current study are excluded from the analysis of the data. By checking the number of vortex particles in the computational domain throughout the time history it is assured that after 1,000 time steps the transient phase is overcome and a steady phase is reached. Furthermore, the effect of iteration to a more stable flow field can be observed by analysing the force and moment coefficients. In particular, the drag coefficient shows high variation in the first time steps. For the example of Model T1 at an angle of attack of 0° the drag coefficient is calculated to $C_D = 25.8$ erroneously for the first time step and settles down to an average of $C_D = 2.6$ for the time steps 1,001 to 10,000. Figure 7.2 shows the lift, drag and moment coefficient for the time steps 1 to 10,000 for the named example.

Figure 7.2: Coefficients for time steps 1 to 10000 (Model T1, 0°)

Figure 7.3: Lift, drag moment coefficients for time steps 1001 to 10000 (Model T1, 0°)

The variation of lift, drag and moment coefficient is illustrated in more detail in Figure 7.3, including the results of the time steps 1,001 to 10,000. Additional to the series of data the minimum, maximum and the mean value are presented in the plots. A recurring variation of the coefficients can be observed with a certain regularity. The lift force, drag force and moment are colligated values and thus, the variations of the coefficients occur simultaneously. Lift and drag force are related to each other as both values are calculated from the pressure, which is normal to the body surface. If the panel, which is exposed to pressure, is not parallel or perpendicular to the flow both components, lift and drag, are influenced. As the moment is calculated by multiplying the forces with the lever arm to the center of the section, it is directly influenced by a change of the forces. The visualisation of the data series in Figure 7.3 shows for example, that four pronounced peaks appear for every of the three coefficients approximately at the same time.

For the design of structures data time histories of the coefficients have to be considered. Neither mean values nor maximum and minimum values might be of interest for the design of a structure. While mean values can compensate to a value of zero if negative and positive coefficients balance, maximum and minimum values might express outlier data, which leads to the overprediction of wind loads and uneconomic design. However, for the comparison between experimental and numerical analysis or the comparison of different geometries (see Chapter 7), mean values can be utilized as comprehensive values to allow an overall statement. The results of VXFlo for the lift, drag and moment coefficient is analysed on the basis of mean values for the different angles of attack in the following.

Figure 7.4 gives the lift, drag and moment coefficient for Model T1. The lift force is negative for the majority of the angles of attack, which indicates a down force acting on the section. The change of sign is expected to occur at an angle of attack between 2.5° and 5° . Therefore, uplift forces are obtained for the angles of attack of 5° , 7.5° and 10° only. For horizontal wind flow (angle of attack 0°) the average lift coefficient is $C_L = -0.72$, which equals an uniform distributed load of -3.9kN/m , applied on the total width of the bridge deck. In general, the absolute lift force is lower for flat angles of attack and increases with increasing

inclination. The same relationship is valid for the drag coefficient, however it is positive regardless whether the section is turned clockwise or counter-clockwise, meaning that the force always acts in the direction of the wind flow. $C_D = 2.57$ is found as average drag coefficient for the angle of attack of 0° , which is equal to 1kN/m applied on the frontal area with the height of 2.9m . As the variation of the moment coefficient is of insignificant extend, the coefficient is not analysed in more detail in the current study. Neither the comparison of the three different geometries T1 to T3, nor the comparison between the numerical and experimental results provide new findings due to the negligible variation.

Figure 7.4: Lift, drag, moment coefficients for Model T1

The variations of lift and drag coefficients for the different geometries are illustrated in Figure 7.5. The function of lift coefficient is more regular for Models T2 and T3, when the lift coefficient is zero for the angle of attack of approximately 0° . This behaviour is affiliated with the decrease of capturing turbulent wind between the wind barriers, which is also depicted in Figures 7.6 to 7.8, showing the increased fluctuation of velocities on top of the bridge deck due to the wind barriers. Removing the wind barriers leads to decreased lift forces for the angles of -10° to 0° but increased forces for the remaining angles. For every angle the lift force for Model T3, characterized by the closed light gap, is higher than for Model T2 due to the increase of horizontal surface area. It can be concluded, that the general effect of the wind barrier on the lift force is an addition of a down force. For positive angles basically characterized by positive lift coefficients, this leads to a decrease of the load. In the same manner an increase of loads for negative angles of attack takes place due to negative coefficients. The general effect of a closed light gap is negative regarding the lift force due to the increase of the absolute forces.

With respect to the angle of attack better symmetry is observed for the drag coefficient of Models T2 and T3. For both models the smallest coefficients are observed for 0° . By removing the wind barriers, the projection area is reduced and therefore, the drag coefficient decreases for both models. It can be observed that the light gap of Model T2 leads to higher drag coefficients for the angles of attack between -7.5° and 2.5° than Model T3 with the closed light gap. At lower angles, the light gap creates additional projective area as the air flows through the center and flows against the face of the downstream section. For high inclinations

though, the opening serves as a discharge, where air is able to flow through without adding forces on the section. In this case the closed light gap leads to a higher projection area. Thus, higher drag coefficients are obtained for the remaining angles of -10° and 5° to 10° of Model T3. Furthermore, for 10° the drag coefficient of Model T3 is close to the one of Model T1. For this angle the effect of the wind barriers is similar to the effect of a closed light gap. As the most relevant angles of attack in bridge design are generally the angles between -2.5° and 2.5° , drag forces on the bridge can be reduced by closing the light gap. If the dynamic phenomena flutter is a concern, larger angles of attack gain importance due to high displacements and need to be considered. Also, designing decks with light gaps is a strategy to avoid or reduce the risk of flutter.

Figure 7.5: VXFlo - Variation of lift and drag coefficients (Model T1, T2, T3)

Figures 7.6 to 7.8 give an illustration of the velocity field around the three investigated models. The color scale is given on top of Figure 7.6 and is valid for all figures due to the identical free flow velocity with values between 0m/s (deep blue) and 25m/s (yellow). The mean velocities are given on the left hand side and fluctuations are expressed by the standard deviation on the right hand side. For mean velocities blue color represents a reduction and yellow or orange colouring an increase of wind velocity. The lighter the color for the standard deviation, the higher the fluctuations are, which occur in this area of the computational domain. The highest disturbances are created by Model T1, which can be obtained by the mean velocity and the standard deviation. The intensity of the velocity changes and the areal extend is most distinct for this model. The turbulent region comprises a wide area around the bridge deck. Additionally, the width of the wake is expanded compared to Model T2 and T3. The velocity field for Model T3 is similar to the one of a streamlined section. The velocity is decreased close to the body surface and slightly increased above and below the boundary layer of the surface. But, compared to Model T1 the influence is small. Downstream of the section, the wake is narrow and does not exceed the height of the body in a influential way. The velocity field of Model T2 shows characteristics, which lie in between Model T1 and T3. The boundary layer on the body surface is of greater extend compared to Model T3, which can be observed at the bottom of the downstream section exemplary. But, the influenced area close to section and the width of the wake compared to Model T1 is reduced.

In order to support the statement that turbulence intensity decreases from Model T1 to Model T2 and T3, the ratio W/H (width of the wake/section height) is analysed in more detail. The approximate width of the wake W for the models is illustrated in Figures 7.6(a), 7.7(a), and 7.8(a). The highest value is gained for Model T1 with $W/H \approx 6.0$ and reduces to $W/H \approx 4.3$ for Model T2. Further reduction to the ratio

of $W/H \approx 2.5$ is observed for Model T3. With decrease of the ratio W/H the disturbance of the flow with respect to the projective area decreases and the turbulence intensity is reduced.

Figure 7.6: Velocity field around Model T1 (0°)

Figure 7.7: Velocity field around Model T2 (0°)

Figure 7.8: Velocity field around Model T3 (0°)

7.2 Experimental Analysis

The test is arranged in the wind tunnel in the Aerospace Engineering Department at Ryerson University. It is a closed-circuit wind tunnel with smooth flow with a longitudinal turbulence intensity of less than 0.4%. Wind speeds up to 70m/s are possible and the mean velocity variations are within $\pm 0.5\%$. Thus, the boundary layer is not simulated in this test, which matches the modelled conditions in VXFlow. As a result, the model can be positioned vertically in the wind tunnel so it can be attached to the turntable. By turning it around the length axis from -10° to $+10^\circ$ in steps of 2.5° the pressure coefficients are estimated for the same angles of attack as for the CFD analysis in dependence on the wind direction. Furthermore, positive angles are clockwise according to the definition of VXFlow (see parameter GEOMROT in Table 6.1).

The models are built with the chosen length scale of 1:80. The sample time is 90 seconds with a sample frequency of 500Hz and a velocity of 25m/s. Hence, 45,000 data points are collected for each pressure tap and each test running. A second test is run with 30m/s to verify the results. Good agreement is found. Figure 7.9 shows the interior construction and the setup in the wind tunnel for Model 1, which will be explained in more detail in the following paragraphs.

Figure 7.9: Model T1

For a total of 32 measurement points, pressures are measured in the center line of the model. Thus, end effects are not likely to influence the quality of the results. Twelve points are equally distributed on each face, including two locations at each side of the cross beam in the center. Two measurement points are located at each wind barrier and due to the restriction of the number of measurement taps pressures are only measured on the outward facing surfaces. Further taps are located on the surfaces facing towards the light gap. The locations of sensors are shown in Figure 7.10 which corresponds to Model T1.

Figure 7.10: Locations of measurement points

The distribution of pressure coefficients over the cross section, which is gained from the wind tunnel test, is as expected. It is reasonable and also the increase or decrease of values due to different angles of attack can be explained by theoretical background of bluff bodies in wind flow. In order to enable the comparison between the results of the wind tunnel test with VXFlo, force coefficients are computed by integration of pressures over the surface area. For this purpose, the measured pressure is assumed for the tributary area as a constant value. The tributary area A_T which varies depending on the spacing of the measurement points, is illustrated in Figure 7.10. The results for lift and drag coefficients of the investigated models are given in Figure 7.11. Similar progression of the coefficients is gained for the different angles of attack as by the analysis with VXFlo. In detail the quantities of the coefficients and also the influence of the different geometries distinguishes from the results of the CFD analysis. The deviations and reasons are further explained in Section 7.3 by comparing the data of VXFlo with the results of the wind tunnel test.

Figure 7.11: Wind tunnel - Variation of lift and drag coefficients (Model T1, T2, T3)

7.3 Comparison and Conclusion

The deviations between the results of VXFlo and the wind tunnel test are given in Table 7.2 and are visualized in Figure 7.12. Positive values in Table 7.2 indicate that VXFlo gives higher values than estimated by the measurements in the wind tunnel. As a consequence negative values denote lower coefficients of the computation with VXFlo. The last column of the table gives the average of the absolute values of the deviations.

Table 7.2: Summary of deviations of force coefficients (numerical - experimental)

Angle of attack		-10°	-7.5°	-5°	-2.5°	0°	2.5°	5°	7.5°	10°	$\varnothing \Delta C$
Model T1	Lift	0.16	0.12	0.04	0.40	0.71	-0.11	-0.20	-0.24	-0.19	0.24
	Drag	-0.91	-1.03	-0.92	-0.28	-0.96	-1.70	-1.46	-1.07	-0.54	0.99
Model T2	Lift	0.17	0.20	0.29	0.21	0.01	-0.01	-0.04	-0.11	-0.11	0.13
	Drag	1.44	1.04	0.30	0.00	-0.22	0.06	0.66	0.96	1.17	0.65
Model T3	Lift	0.36	0.11	0.17	0.17	0.00	0.10	0.16	0.08	0.05	0.13
	Drag	2.02	1.04	0.15	-0.10	-0.27	-0.16	0.05	0.47	1.23	0.61

Figure 7.12: Wind tunnel results: Variations of lift and drag coefficients (Model T1, T2, T3)

In general, the coefficients of the two approaches are qualitatively similar to each other. Higher conformance is found for the lift coefficients, which can be obtained graphically but also by the average deviation, which

is given in Table 7.2. The average deviation is $\emptyset\Delta C_D = 0.99$ (T1), $\emptyset\Delta C_D = 0.65$ (T2) and $\emptyset\Delta C_D = 0.61$ (T3) for the drag coefficient, which is higher than the deviation of the lift coefficients with $\emptyset\Delta C_L = 0.24$ (T1), $\emptyset\Delta C_L = 0.13$ (T2) and $\emptyset\Delta C_L = 0.13$ (T3). The average deviations also indicate that the conformance is increasing with increasing simplicity of the model as the deviation is constantly reduced from Model T1 to Model T2 and Model T3.

The higher difference between the numerical and experimental data sets of T1 compared to the Models T2 and T3 can be a result of the arrangement of the pressure taps. In the current study inaccuracy is introduced generally due to the low number of measurement points in the experimental test setup. 32 measurement taps are available in the wind tunnel laboratory at Ryerson University. For comparison the number of pressure taps, which are usually installed to give precise information about the pressure distribution, ranges between 500 to 800. Large spacing of the pressure taps and the lack of sample points at certain areas (the inner side of the wind barrier is cited as an example) introduces additional uncertainties for the calculated force coefficients of the wind tunnel test. Hence, higher deviations are observed for Model T1 due to the lack of information about the pressure on the inner side of the wind barrier. Model T2 and T3 do not include wind barriers and therefore this uncertainty vanishes. Additional assumptions are made for all of the models as forces are calculated in this study by integration of pressures applying the measured pressure on the tributary area without taking the variation over this area into account. Due to the small number of measurement points and the simplified method of integration, known as rectangle method, the approach needs to be rated as an estimation and the high deviations between the experimental and numerical results of approximately 33% in average can be explained.

Moreover, the experimental and numerical results for Model T3 are expected to be closer than for the other two models because the closed light gap leads to two-dimensional properties of the model in the wind tunnel. Basically, the wind tunnel test is three-dimensional, which is characterizing the tests of Model T1 and T2 because these have an altering geometry in longitudinal direction due to the light gap and recurring cross beams. However, the influence of three-dimensional objects is not taken into account in VXFlow. Instead, the cross section with gap is represented and the influence of the cross beams is neglected. For Model T3, whose geometry does not change in length direction, the influence of three-dimensional effects vanishes. Therefore, the modelling as a two-dimensional section of infinite length in VXFlow is more accurate for this model. The analysis of the average divergences show that the drag forces converge (reduced difference of $\Delta C_D = 0.61$ for Model T3 compared to $\Delta C_D = 0.65$ for Model T2) but the divergence for the lift coefficient remains unchanged at $\Delta C_L = 0.13$.

Another difference in the model of the wind tunnel test and the one of VXFlow are the boundary conditions. Contrary to the considered domain in VXFlow which is free of boundaries, the utilised wind tunnel has walls on each side of the section. In closed flow fields as for the wind tunnel test, blockage effects occur when bluff bodies disturb the wind flow. The extend depends on the ratio of projected area of the object to the area of the test section. Blockage of the wind tunnel leads to an accelerated flow around the model and distorts the measurement results. According to You-Lin [3], blockage effects have to be considered if the model takes up more space than 5% of the cross section of the wind tunnel. Table 7.3 shows the blockage ratio for the three models depending on the angle of attack. The maximum blockage effect is calculated to 14% (Model

T1) while the minimum is 4%.

Table 7.3: Blockage ratio of experimental study

BLOCKAGE RATIO			
Angle of attack ($\pm$)	Model T1	Model T2	Model T3
0 °	7%	4%	4%
2.5 °	8%	5%	5%
5 °	10%	7%	7%
7.5 °	11%	8%	8%
10 °	14%	11%	11%

Higher deviations are expected with increasing blockage effect in the wind tunnel and thus, the significance increases with the enlargement of the angle of attack. Furthermore, the total influence is reduced for Models T2 and T3 due to the removal of the wind barrier. Thus, the accuracy increases for these two models. Both effects can be confirmed by the analysis of the deviations given in Table 7.2. In general, the deviation becomes minimum for the angle of attack of 0°, and increases with increasing inclination of the model. Also as mentioned, before the average deviation $\bar{\Delta}C$ shows that the best conformance is found for Model T3 and the worst for Model T1. It should be noted that small vibrations were observed for higher blockage effects at the leeward top edge of the section model, which can lead to further inaccuracies of the data.

Additionally the results of wind tunnel tests need to be scrutinized as they vary with the setup and surrounding conditions. Carrying out the same wind tunnel tests in different renowned wind tunnel laboratories often show discrepancies in their results. As an example, the same study was conducted for two industrial buildings for different terrains in multiple laboratories [24]. Their reports were compared for the moment at the clamped support of the portal frame and the pressure coefficients measured by a tap installed in the roof corner. The results varied significantly from the smallest to highest estimate of the six different laboratories, up to a factor of 3.6. The comparison of the reports showed that values have a higher range for lower structures. Furthermore, the variability occurred greater for higher surface roughness. The results have been compared for open terrain and suburban terrain, the latter of which contributes to the increase of discrepancy. Simiu and Miyata [5] explain this phenomena by the difficulty of defining the mean flow for lower levels, which gets even more difficult with higher surface roughness and indicated fluctuations. For the current study, it means that variations are expected to increase from Model T3 to Model T2 and Model T1 because turbulence intensity increases which can be observed qualitatively by the visualisation of the standard deviation in Figures 7.6 to 7.8. This leads to the conclusion that wind tunnel studies and their results have to be evaluated critically.

Morgenthal [15] also mentioned the difficulty of replicating results of the wind tunnel with CFD tools due to different restrictions of both methods. On the one hand, the experimental method mirrors the physical conditions of the flow perfectly. But inaccuracies can occur due to scaling factors e.g. for geometry, wind speed and viscosity. Furthermore, measurements can only be taken at sample points and it is not possible to get the properties of the entire flow field. On the other hand, CFD tools provide complete data of the flow

field and characteristics can be extracted at almost every sample point needed. However, the restriction of numerical models is the simulation of the physics. Whereas the physical equations introduced in Chapter 4 describe the flow perfectly, the numerical methods (see Chapter 5) involve simplifications in order to shorten the computational process. As a consequence of these simplifications and depending on the extend of the simplifications, the numerical models do not mirror the physical flow in every detail.

8 Investigation of different Wind Barriers

Wind barriers are part of most bridge architecture as they improve the safety for vehicles. However, they lead to an enlargement of the windward area of the bridge and therefore, increase the static wind loads on the bridge. In literature a lot of studies can be found regarding the optimization of wind barriers. Due to the big amount of parameters which can be changed in the design of the barrier and the different optimization criteria, it is not possible to give a universal design guide, which will satisfy the demands of every bridge structure. Criteria and parameters are listed in Table 8.1, which is not intended to be an exhaustive list. The analysis in the current study refers to the static loads and the effect of the different geometries on the dynamic properties of the bridge is not considered.

Table 8.1: Criteria and parameters for wind barrier design

CRITERIA (to be optimized)	PARAMETERS (to be changed)
Bridge Vehicle ↳ Lift Force ↳ Drag Force ↳ Moment Force ↳ Constant forces ↳ Fluctuation of forces ↳ Angles of attack	Distance Height Porosity Global geometry Shape of single elements

For optimizing the wind barrier design the two general objectives are to limit both, the loads on the vehicles and on the bridge deck. However, these two criteria have an adverse influence on each other. A balance has to be found so that sufficient safety is gained for the traffic participants and the increase of loads on the bridge is acceptable. As an example, no barriers lead to higher forces on the vehicles but minimize the forces on the bridge deck. With the increasing height of the wind barrier the forces on the vehicle decrease generally, but the forces on the bridge deck increase. For both the bridge and the vehicles the lift, drag and moment can be optimized. Furthermore, the constant forces or the fluctuations can be of primary interest and every criteria can be analysed for the different angles of attack. In sum, there are 12 different criteria for every angle of attack, which are of varying importance depending on the structure and environmental conditions. For vehicles the fluctuation of drag forces and the loading due to constant moments, as a measure for sideslip and overturning of vehicles, are named as significant criteria. For bridges the importance of the factors depends on the stiffness of the bridge deck against lateral, horizontal and torsional displacement. Regarding static loads, the constant forces and moments are decisive while fluctuations determine the dynamic behaviour like flutter or vortex induced vibrations [25].

Parameters are to be changed to adapt the design of the bridge deck in order to satisfy the requirements for the particular asset. The distance of the wind barrier to the vehicle affects the extend of protection, which correlates directly to the height h of the wind barrier. Measurable reductions are observed up to 50h,

but at a distance of $25h$ to the wind barrier the decrease is expected to be approximately 20% only [26]. Recommendations for the height of at least 2-3m are given by James [27] depending on the mainspan of the bridge and its susceptibility to aerodynamic phenomena. Furthermore, the porosity ratio of wind barriers contributes to the reduction of wind loads on the bridge deck but as adverse effect, the constant loads on the vehicles are found to be higher than for solid barriers. Porosity can be introduced by openings in the barrier, whereas the size of the openings influences the characteristics of the wake [28]. Alternatively, single elements in longitudinal direction can be arranged with gaps to create porosity. 30% porosity is considered to be optimal for wind barrier design stated by Kozmar et.al. [29] and 50% is accepted as reasonable compromise by James [27]. For single elements, the geometry and distance have an influence on the flow characteristics [30]. As examples for the shape of these single elements a flat plate with adjustable inclination, a 'V'-profile or an airfoil can be mentioned. Furthermore, the global shape of the wind barrier, meaning the inclination, curvature or the edge layout has an impact on the forces and moments.

The original design of the investigated bridge is provided with solid wind barriers with an inclination towards the center of the bridge. The designated angle of inclination is 20° . The client is interested in changing the design to perpendicular wind barriers or to an outward facing variant in order to eliminate the loss of space on the bridge deck below the barrier. Changing the angle to 0° or negative angles, the width of the bridge deck can be reduced while having the same usable deck area and reducing the project costs. The loss of space for Model B1 due to the inclination of the wind barrier towards the center is illustrated in Figure 8.1. The geometric profit which can be gained by other designs respectively is shown in the same figure for the example of B3 (see Section 8.1 for the different wind barrier designs). However, the change of the geometry will influence the flow properties and forces on the bridge deck. The expected increase of wind loads leads to the need of a stiffer girder and increases the costs in turn. Another important factor of wind barriers is the effectiveness of the protection of the vehicles in order to prevent road accidents due to sudden steering movements. Different designs of the wind barriers are analysed, considering the effect on the structure and the vehicles.

Figure 8.1: Loss of space due to original wind barrier design

8.1 Wind Barrier Design

Five different geometries (B1, B2, B3, B4, B5) are analysed and there are two configurations for every section geometry, one without and one including two vehicles.⁴ The height of the wind barrier is 2.50m in the original design and is kept at this value for all the geometries which are analysed. Table 8.2 gives an overview of the different geometries. The original model is shown with the investigated trucks exemplary.

Table 8.2: Models for investigation of wind barriers

NAME	DESCRIPTION	CROSS SECTION GEOMETRY
B1	Original design Inclination towards center $\alpha = 20^\circ$
B2	Perpendicular $\alpha = 0^\circ$
B3	Inclination outward $\alpha = -10^\circ$
B4	Recurved geometry $\alpha = -10^\circ$
B5	Rounded shape Adapting the geometry of Model B4

8.2 Effects on the Bridge Deck

The force and moment coefficients are analysed for the Models B1 to B5. The focus is put on the deviations between the different wind barrier designs in the following paragraphs. The coefficients are derived according to the principles introduced for the analysis in Section 7.1. The applied parameters and definition of positive and negative forces are given in Table 7.1. The mean values of the force and moment coefficients are

⁴Despite Model B5 includes rounded geometries, the separation point occurs at the protruding edge of the wind barriers and hence, the issues which occurred for the circular cylinder in Section 6.4 are not of influence for the bridge design of Model B5.

illustrated in Figure 8.2. Note that the positive effect of decreased coefficients compared to the coefficients of Model B1, is marked by a plus sign and the negative effect by a minus sign accordingly. Table B.1, Table B.2 and Table B.3 of Appendix B give the coefficients including the statistic analysis in more detail. Additional to the mean values, the minimum, maximum, and standard deviation is given.

Figure 8.2: Force and moment coefficients by comparison

The general change of forces in dependence on the angle of attack is similar for the investigated models. The lift coefficient is negative for the angles of less or equal 0° , which represents a downforce. On the contrary, the bridge is exposed to an upward force for the angles between 2.5° and 10° . The drag coefficient is positive for all angles, which expresses a horizontal force in flow direction. Windage is gained as the windward area of the bridge deck enlarges with increasing angle and therefore, the drag force increases with increasing angle, too. The total drag force is higher for negative angles than for positive ones because the surface opens up towards the incoming wind and the flow is captured between the wind barriers. In comparison, the bottom face of the bridge deck is more similar to a streamlined shape, which is significant for positive angles. The relation can be observed in Figures 8.3 to 8.5, in which the velocity field of Model B1 is illustrated as an example. The color bar which is valid for the three figures due to identical free stream velocity, is given on top of Figure 8.3

Figure 8.3: Velocity field around Model B1 (0°)

Figure 8.4: Velocity field around Model B1 (10°)

Figure 8.5: Velocity field around Model B1 (-10°)

As the most important angles of attack usually range from -2.5° to 2.5° regarding the analysis of static wind loads, the deviations for these angles are analysed in more detail in the following paragraph. Table 8.3 gives the deviations of the coefficients for the different designs of wind barriers. The coefficients of the original design are set as the reference value and therefore, the absolute divergence of the coefficients is zero for Model B1. Negative values denote a negative effect which means an increase of the coefficients, resulting in higher loads on the bridge deck. On the contrary, positive values express the decrease of the coefficient. Additionally, the deviations for all angles are given in Table B.4 to B.6.

Table 8.3: Summary of deviations of force and moment coefficients (bridge deck)

INFLUENCE OF WIND BARRIER DESIGN ON BRIDGE DECK						
COEFFICIENT		B1 (Reference)	B2	B3	B4	B5
LIFT	-2.5 °	0	-0.06	0.02	0.07	-0.15
	0°	0	0.07	0.01	0.01	-0.02
	2.5 °	0	0.01	-0.13	-0.12	-0.09
DRAG	-2.5 °	0	-0.29	0.01	-0.11	-0.22
	0°	0	-0.19	-0.06	-0.32	-0.74
	2.5 °	0	-0.07	-0.08	0.00	-0.03
MOMENT	-2.5 °	0	-0.05	-0.04	-0.01	-0.03
	0°	0	-0.04	-0.06	-0.06	-0.09
	2.5 °	0	-0.03	-0.10	-0.11	-0.11

The change of coefficients ranges from a minimum of $\Delta C_L = -0.15$ to a maximum of $\Delta C_L = 0.07$ for lift, from $\Delta C_D = -0.74$ to $\Delta C_D = 0.01$ for drag, and from $\Delta C_M = -0.11$ to $\Delta C_M = -0.01$ for moments. The resulting difference of forces for a design wind speed of 30m/s is calculated for the maximum absolute deviations exemplary. For the lift coefficients the difference of $\Delta C_L = -0.15$ for Model B5 equals an increase of vertical loads of approximately 3.3kN/m. By comparison the drag force increases by 1.2kN/m for the deviation of $\Delta C_D = -0.74$, which was observed for Model B2. For moment coefficients the deviation is $\Delta C_M = 0.11$ (Model B3 and B4), which leads to an increase of 96.0kNm/m. The effect is of limited extend, which can be explained by the large distance between the two barriers. After the disturbance of the flow by the windward barrier, the flow mostly normalises until it is interfered by the leeward wind barrier. This leads

to the counteraction of the barriers. As the geometry of the barriers is mirrored, the effect of the windward barrier is reversed by the leeward one. Figure 8.3 illustrates the normalisation of the flow as a separate wake can be recognised for each segment of the bridge deck.

Considering the total increase of forces, the variance of the lift and moment coefficients play a major role. The highest percentual impact is given by the moment coefficients. For the criterion of the minimal wind loads on the bridge deck Model B1, that means the original design with an inclination of the wind barrier of 20° towards the center, gives the best results of drag and moment coefficient. For the other models (B2 to B5) the coefficients increase in average. Furthermore, for the drag force and moment improvement is observed for Model B3 at -2.5° only, but at a very low rate so that the positive effect for the single value is negligible in the global context.

A reduction of force coefficients is obtained more occasional for the lift coefficients for Models B2, B3 and B4. Model B2 shows the best result for the basic position of 0° with an improvement of 0.07, which equals 1.5kN/m for the exemplary wind speed of 30m/s. Also, the average advance of the three angles of attack is the maximum for Model B2. Model B5 features negative divergences only and therefore, clearly leads to a negative effect on the aerodynamic characteristics of the bridge deck. However, the choice about which section geometry is the best depends on which of the coefficients is the determining one. This in turn depends on the vertical, horizontal, and rotational stiffness of the bridge deck. Whereas one design bears vertical loads better than horizontal ones, others might offer more resistance against moments. Especially for lift coefficients the divergence has to be analysed with respect to its sign of the lift coefficient. To a certain degree positive lift forces found for positive angles, do not put additional stress on the system but unload the bridge deck as they act against dead load. Thus, an increase of positive lift coefficients is intentional and rated as positive effect.

8.3 Effects on the Vehicles

The most important factors for the safety of vehicles are the moment coefficient and the fluctuation of the drag coefficient. Both will be observed separately in this section. Whereas high moment coefficients lead to the overturning of the vehicle, the sudden change of the drag coefficient leads to sideslip [31] (see Figure 8.6). Both are typical reasons for accidents and need to be limited to a tolerable level by reducing the loads on vehicles due to wind.

Figure 8.6: Overturning and sideslip of vehicles due to wind after Kwon et al.[31]

In the current study the effect on the vehicles is considered separately for a vehicle on the windward section and another one on the leeward section of the bridge deck. The position of both vehicles is on the right lane

seen from the driving direction. An illustration of the location of the vehicles on the bridge deck is given for the example of Model B1 in Table 8.2. A truck was investigated with the width of approximately 2.5m and height of 3.5m. The parameters used to calculate the coefficients of the vehicles are given in Table 8.4. A illustration, which shows the definition of the forces and moments in positive direction and a review of the equations to calculate the force coefficients is given in the last row.

Table 8.4: Reference values for computing coefficients of vehicle

PARAMETER		VALUE	
B	Width of vehicle	2.43	m
H	Height of vehicle	3.48	m
L	Length	1.0	m
ρ	Density of air	1.2	kg/m^3
ν	Viscosity of air	$1.5 \cdot 10^{-5}$	m^2/s
u_∞	Free stream velocity	15	m/s

Figures 8.7 to 8.9 show the average flow pattern of the bridge deck including vehicles for the angles of attack of -10° , 0° and 10° . Even if the most important angles of attack range between -2.5° and 2.5° the exaggerated illustration of -10° and 10° is chosen. Furthermore, Model B4 was chosen as an example, as the characteristics of the flow pattern are similar for Models B1 to B5 in general. For 0° the wake is horizontal and is spread from the top of the vehicles to the bottom of the bridge approximately. The width of the wake increases with increasing angle. For 10° higher velocities are obtained in the light gap between the two main girders. The wind is captured and diverted upwards. It is assumed that thereby, additional turbulences are introduced, which leads to lower velocities in the region of the leeward vehicle. Moreover, the leeward vehicle is protected by the windward section and vehicle due to the excessing height. For the angle of attack of -10° two separated wakes can be observed for each bridge deck. The windward vehicle is still located in a velocity field with lower values influenced by the wake created by the wind barrier, but the flow around the leeward vehicle is more similar to the one of a vehicle in the free flow.

Figure 8.7: Velocity field around Model B1 including vehicles (0°)

Figure 8.8: Velocity field around Model B1 (10°)

Figure 8.9: Velocity field around Model B1 including vehicles (-10°)

The average values of the moment coefficients of the vehicles for the different wind barrier designs are displayed in Figures 8.10 and 8.11. Again, the positive effect of lower coefficients is marked by a plus sign and the negative effect by a minus sign accordingly. At this point the lift and drag coefficients are not presented due to their minor influence when it comes to the analysis of the overturning of the vehicles. Nevertheless, Tables B.7 to B.9 and B.13 to B.15 give the statistic values for all coefficients in more detail. Additionally

to the mean values, the minimum, maximum, and standard deviation are given in these tables.

Figure 8.10: Moment coefficients by comparison of windward vehicle

Figure 8.11: Moment coefficients by comparison of leeward vehicle

According to the definition of positive directions for the forces and moments, negative moment coefficients express counter-clockwise moments. Negative moment coefficients occur for the windward vehicle for the majority of angles, which is illustrated in Figure 8.10. Similarly, the drag force for the windward vehicle is negative for most of the cases, which can be taken from Table B.8. The wind loads on this vehicle act in the reverse direction of the wind flow. The effect can occur due to the creation of vortices behind the wind barrier, causing a local suction effects. On the contrary, the majority of the moment coefficients for leeward vehicles is positive (see Figure 8.11). Besides, the wind load is higher than for the windward vehicles for negative angles. In contrast, the leeward vehicles are better protected against wind loads for positive angles of attack so that the moment coefficient is close to zero for angles between 2.5° and 10° . Comparing the height of the leeward vehicle, which increases the height of the wind barrier, changes in the moment coefficients can be deduced. Additional to the wind barrier height, the protection due to the windward vehicle is taken into account. It is located in the wake of the windward vehicle for 0° and 10° , whereas it is almost fully exposed to the free wind flow for -10° . The excess height directly influences the drag force of the vehicle and therefore, the moment coefficient.

Analysing the deviations for the windward vehicle first, it can be observed that improvement is achieved for certain angles of attack and models. A decrease of the moment coefficients can be observed for the design with perpendicular wind barriers (Model B2) for the majority of angles. In comparison with Model B3 (outward facing barriers) the effect on the coefficients is preferable. Good results are also gained by the rounded geometry (Model B5) for angles between 0° and 10° . For negative angles the shape has an negative effect on the moment coefficient instead. Poorer results are obtained for Model B4, where a decrease of the moment coefficient occurs only occasionally. In conclusion, Model B2 shows the best results regarding the windward vehicle, as the absolute decrease of the moment coefficient is the highest.

Regarding the leeward vehicles the percentual variation is relatively narrowed. The distance of the vehicle to the wind barrier is approximately 30m and therefore, the influence of the wind barrier decreases. Furthermore, an analysis is chosen with both vehicles modelled at the same time, so that the leeward vehicle is located downstream of the windward vehicle. Thus, the shape of the barrier is of lower influence. High loads are expected to occur for the leeward vehicle if there is no temporary barrier due to another vehicle. The addition of a vertical wind barrier in the center of the bridge deck is a possible solution to reduce the wind loads on the vehicle on the leeward lane and improve the safety for traffic participants. An increase of loads on the bridge deck is expected as a side effect of this solution. However, further investigations with different settings of vehicles and additional wind barriers should be carried out to study further details.

Additionally, the moment coefficients obtained for the angels of attack of -2.5° to 2.5° , which are decisive for the limitation of overturning, are reviewed in Table 8.5. The absolute deviations of the mean coefficients to the coefficients of the original design, represented in Model B1, is given. The table confirms that the best results concerning the moment coefficients of the vehicles are obtained for Model B2. Higher positive deviations are gained ($\Delta C_M = 0.31$, $\Delta C_M = 0.28$, $\Delta C_M = 0.18$) and a negative effect is observed for the windward vehicle at 0° only. The entire values for all angles of attack, which have been investigated, are given in Table B.12 and B.18. Additionally, the deviations for lift and drag coefficients are given in Tables B.10, B.11, B.16, and B.17.

Table 8.5: Summary of deviations of moment coefficients (vehicles)

INFLUENCE OF WIND BARRIER DESIGN ON VEHICLE						
MOMENT COEFFICIENT		B1 (Reference)	B2	B3	B4	B5
WINDWARD	-2.5 °	0	0.04	0.05	-0.06	-0.13
	0 °	0	-0.06	-0.15	-0.04	0.12
	2.5 °	0	0.28	0.00	0.05	0.21
LEEWARD	-2.5 °	0	0.18	0.12	0.31	0.26
	0°	0	0.31	0.28	0.01	0.26
	2.5 °	0	0.02	0.02	0.02	-0.01

The maximum differences of the moment which act on the vehicle are calculated for the example of 30m/s according to Table 8.5. The maximum decrease of the moment coefficient is obtained for Model B2 (leeward vehicle, 0°) with $\Delta C_M = 0.31$. This equals a decrease of 1kNm/m (per linear meter of the vehicle). The minimum deviation is obtained for Model B3 of the windward vehicle at an angle of 0° , with $\Delta C_M = -0.15$, and thus the moment increases by 0.5kNm/m. The deviations range between 0.2kN/m (decrease) and 0.1kN/m (increase) for the lift force and 0.4kN/m (decrease) and 0.1kN/m (increase) for the drag force. The angles of attack between -2.5° and 2.5° are taken into consideration. The deviations of the force coefficients are given in Appendix B (Table B.10 to B.12 and Table B.16 to B.18).

Table 8.6: Summary of standard deviations of drag coefficients (vehicles)

INFLUENCE OF WIND BARRIER DESIGN ON VEHICLE						
DRAG COEFFICIENT		B1 (Reference)	B2	B3	B4	B5
WINDWARD	-2.5°	0	0.01	0.00	-0.01	0.01
	0°	0	-0.01	0.00	-0.01	-0.01
	2.5°	0	-0.06	-0.05	0.01	0.01
LEEWARD	-2.5°	0	0.01	0.05	-0.08	0.02
	0°	0	0.01	0.02	-0.08	-0.07
	2.5°	0	0.00	-0.05	-0.06	-0.05

The standard deviation for the drag coefficient is analysed in more detail to investigate the risk of sideslip due to the sudden change of wind forces on vehicles. The results given as the difference of the standard deviation in reference to Model B1 are summarized in Table 8.6. The absolute values for the standard deviation of the drag forces are given in Table B.11 and B.17 in the Appendix B. By analysing the absolute values the statement that the windward vehicle is more protected than the leeward one can be supported. The maximum standard deviation of the drag coefficient for the windward vehicle is $\sigma_{C_D} = 0.24$ and $\sigma_{C_D} = 0.67$ for the leeward one. The higher value for the leeward vehicle indicates higher fluctuations due to higher exposure.

Regarding the deviations (see Table 8.6), positive values denote the positive effect of decreased standard deviation. On the contrary, negative values indicate the increase of fluctuations of the drag force. Model B2 which showed the best results regarding the limitations of the overturning moment, shows now higher standard deviations for the windward vehicle. Slight improvement is observed for the leeward vehicle, but considering the absolute values of the standard deviation in this case ($\sigma_{C_D} \approx 0.4$ in average for Model B1) the influence is negligible. Overall, the wind barrier designs of Model B4 and B5 have a negative effect on the fluctuations of cross wind for both vehicles. However, the evaluation of the standard deviation of Model B3 gives a controversial result because improvement and worsening is greatly varying with the different angles of attack.

8.4 Summary

In conclusion, it can be said that the original design leads to the lowest forces on the bridge deck itself. On the other hand, it is the least effective regarding the moments on the vehicles. The original design leads to the highest moment coefficients for both, windward and leeward vehicle and hence, entails the highest risk for the overturning of trucks. Improvement is achieved for every wind barrier design and for most of the angles of attack. The best results are gained by Model B2, which is the perpendicular wind barrier for which the windward and leeward vehicle experience the least forces and moments. Regarding the analysis of sideslip the wind barrier design of Model B2, B3, B4 and B5 show a negative effect in average. The best solution is obtained for the original design. As explained in the beginning of the current chapter (see Table 8.1) the optimization of the wind barrier depends on many criteria. Depending on the parameter selection, e.g. decreasing the forces for the bridge deck or increasing the safety for traffic participants, different designs give the optimal solution. As explained before, VXFlo does not include three-dimensional effects. Due to the short length of vehicles, this can be of major influence for the results. The extend of the influence needs to be studied in more detail to eliminate the uncertainty of reducing the analysis to a two-dimensional problem. Additionally, uncertainties are given because the number of runnings is restricted due to the limitation of computational time and cost. A more extensive analysis should include the variation of the number of vehicles, the variation of positions on the four lanes of the cross section and the influence on the dynamic behaviour of the bridge.

9 Conclusion

Reproducing the reality in its entirety is not the task of any model. This is expressed very clearly in the aphorism of George Box. However, the question is not if the model is right or wrong. In fact, the question which matters is if the model represents the reality in a sufficient way, so that the results of the test do not deviate too much from the properties of the full-scale structure. This is put into words by the second part of Box's aphorism, which states that some "models are useful". If the model is designed correctly, the test leads to meaningful results even if the prototype is not reproduced perfectly.

"Essentially, all models are wrong, but some are useful."

- George Edward Pelham Box (1919 - 2013) -

Both wind tunnel tests and CFD packages are tools to simulate the reality. However, both have limitations, which should be considered in dependence on the application.

The current work describes the investigation of wind loads on a bridge deck. The applied CFD tool is validated by reference data, which are found in the Eurocode and in literature. A further comparison is drawn between the data of VXFlow and the results of a wind tunnel test, carried out as a rigid section model of a bridge deck. The application of VXFlow is performed for the study of different wind barrier designs, regarding the loads of the bridge deck and the safety of traffic participants. The findings concerning the approach and results of the investigations are reviewed in the following paragraphs.

CFD tools are based on Navier-Stokes Equations, which are introduced in Chapter 4. These equations describe the flow in every detail but the analytical resolvability is restricted to a few theoretical cases and it takes a lot of resources to solve the equations. Thus, numerical models are utilized to solve the equations, which include assumptions to enable solutions with reasonable computational time and cost. The Vortex Particle Method, on which the VXFlow code is based, is one of the methods. The Euler Equations for inviscid flows are basis of the method and viscosity is reintroduced by an irregular parameter generated by the Random Walk Method. Furthermore, it is a grid free method based on particles which carry vorticity depending on the pressure and velocity characteristics in the computational domain.

The comparison with the data in the Eurocode and literature showed that the investigation of pressure coefficients is possible with VXFlow only for certain cases, where a reference pressure is known. As the pressure coefficients are related to a unknown value instead of to the pressure in the free flow as for the common approach, the pressure coefficients can be evaluated qualitatively but not quantitatively if they are not scaled with respect to a known reference pressure. After scaling the pressure coefficients for the circular cylinder conformity to the Eurocode is found for the windward side of the section but, suction is overestimated on the leeward half of the cylinder. The same phenomena of overestimation is found for the drag coefficient of the

CONCLUSION

cylinder. The variance of the drag coefficient for different Reynolds Numbers has not been observed on results achieved by VXFlow. Therefore, a further study addressed to the restriction of computing the variation of the separation point for rounded shapes should be carried out. Better results are found for rectangular sections, because the separation point is fixed to the protruding edge. In particular, good conformance is achieved for flat plates with aspect ratios $B/H > 1$. However, higher deviations are obtained for aspect ratios B/H smaller than 1.0. The analysis shows that rounded and streamlined shapes have to be analysed deeply, because the flow characteristics are highly dependent on the Reynolds Number. Bridge decks are mostly bluff bodies but sometimes streamlined elements are included. Therefore, these studies are applied with reservation and results have to be evaluated in more detail. For the investigation of bridge decks, the aspect ratios are usually within the ranges for which good results could be found for the rectangular geometries, but it is recommended to complement further studies to support the expectation of conformance.

The comparison of static wind loads on a bridge deck between the data of a wind tunnel test and results of VXFlow is based on the measurement of pressures on a rigid model. The investigation included three different model types, where the geometry ranged from simple to more complex. By integrating the pressures gained by the wind tunnel test along the surface of the cross section, drag and lift coefficients are approximately calculated to enable the comparison with results of VXFlow. The method of calculating the force coefficients involves uncertainties due to the large spacing between pressure taps and the assumption of constant pressure along the tributary area. Additional missing measurement locations, for example at the inner side of the wind barrier, lead to inaccuracy of the lift and drag coefficient gained by the wind tunnel test. Furthermore, the results of the wind tunnel test might be influenced by inaccuracies due to the instruments for measurement and the behaviour of the model. For example, small vibrations which were observed for higher blockage ratios at the leeward top edge can lead to inaccuracies of the data. Additionally, blockage effects influence the measured pressures. Especially in the case of Model T1 which includes wind barriers and therefore features the highest blockage ratio, the impact is expected to be perceptible. Another factor which restricts the comparability between a wind tunnel test and VXFlow is that wind tunnel tests can capture three-dimensional effects, while VXFlow is developed for the computation of two-dimensional problems. The criteria influences the comparison of Model T1 and T2. The geometry is constant for Model T3 in longitudinal direction and thus, the influence vanishes. Considering the different limitations, increasing conformance is expected with increasing simplicity of the model. The deviations between the data sets of the wind tunnel and VXFlow show a corresponding increase of accuracy.

For all results the drag forces achieved through static simulations are overestimated by VXFlow in comparison to the Eurocode or literature. The deviations to the test data of the wind tunnel test and observed underestimation are mainly referred to restrictions of the wind tunnel test. In general, closer results are obtained for the lift coefficient. It should be noted that the findings of the current study base on static simulations, exclusively. The applicability for dynamic simulation is not studied.

By a technical application of VXFlow the influence of the geometry of the wind barrier is studied. Five different geometries are investigated and the influence on the wind loads on the bridge deck as well as the influence on the traffic safety is taken into consideration. A large number of criteria and parameters determine the optimization of the geometry. In the current study the static loads on the bridge deck are studied,

CONCLUSION

as well as the average moment coefficient and the fluctuations of the drag coefficient of the vehicles. The best results for the limitation of loads on the bridge deck is gained by the original design with the outward facing wind barrier. Regarding the fluctuations and the risk for sideslip of vehicles, the performance of the original design is preferable as well. But concerning the risk for the overturning of the vehicle due to cross winds, the perpendicular wind barrier reached the best results and the original design entails the highest risk. The comparison is not conclusive for the different wind barrier designs because other arrangements of vehicles can lead to different results. Furthermore, the optimization for the geometry of the wind barrier depends on the criteria of the highest importance. Additional to the effect on the static wind loads, the change of dynamic properties need to be considered. While a design can improve the static loading of the bridge deck, it can have a reverse effect on its dynamic behaviour.

In conclusion, VXFlow is a CFD tool which allows fast and cost effective analysis of static wind loads on bluff bodies. Accuracy is given in a sufficient way for comparative studies of different geometries and structural optimization. It can be applied to investigate the wind loads in the preliminary design of a structure or the analysis of different design proposals to discard those which do not present good aerodynamic design. Therefore, the application of the VXFlow tool can minimize the number of alternatives for which wind tunnel tests need to be performed. The main limitations of the tool seem to be the restriction to two-dimensional problems and the overestimation of the drag coefficient. The fact that the force coefficients are predicted to high, gains safety in the application of the tool but, it can lead to uneconomic design if the predictions are out of a certain range. Hence, it is recommended to carry out a wind tunnel test to confirm the results of the CFD analysis especially if the structure is sensitive for wind loads or if an overestimation of loads increases the building costs severely.

Further studies comparing the performance of VXFlow with other CFD tools would allow a more profound rating of the code. As mentioned before the accuracy is expected to be higher for other codes, which are based on numerical models including less simplifications. However, an increase of cost is expected for the application of these tools due to the prolonged computing time. Moreover, the handling of VXFlow is very clear and comprehensible. By contrast with the VXFlow tool the amount of settings is extensive for other software packages (e.g. Star CCM+) which makes the application of CFD tools more complicated and prone to user errors. The comparison between the application of VXFlow and another tool would allow an assessment if the additional computational effort would increase the accuracy in a worthwhile way.

A VXFlow Software

A sample code is given in the following for the analysis of Model B4 including vehicles. The angle of attack is chosen to be 2.5° .

```
NSLICES=-1
U0X=15
DYNAMIC=0
STRCMODEL=1
DELTAT=1.0
NSTEPS=10000
NPANELFACT=1.0

P3MINTP=2
P3MFD=-1
NR=2
P3MNRRELAX=100

WINMINX=-80
WINMAXX=400
WINMINY=-120
WINMAXY=120

PRFL=1
PRSTEP=1

FVFL=1
FVSTEP=500

RSTFL=0
RSTSTEP=100

VXFL=1
VXSTEP=500

VIZFL=0

ISOMINX=-80
ISOMAXX=400
ISOMINY=-120
ISOMAXY=120
ISODIMX=800
ISODIMY=400
```

APPENDIX

!!

NUMSEC=4

ASEC0=28.5708

ASEC1=28.5703

ASEC2=8.5994

ASEC3=8.5994

GEOMROT=2.5

ROTCENTREX=0.0

ROTCENTREY=0.0

CORER=1.2

GRIDMINX=-80

GRIDMAXX=400

GRIDNX=511

GRIDMINY=-120

GRIDMAXY=120

GRIDNY=255

AVGPRESS=1

!!

19//num_ cornerpoints

0.2 0.0 //release_ distance*spacing_ hull

0.02 0.001 0.0 -0.1 //merg1*merg2*merg3*merg4

-4 -3 -1 -3 -1 //section_ color_ coding:drag*lift*moment*displ*rotation

1	-2.4001	1.5987	10
1	-4.9851	1.4953	1
1	-5.0001	1.4203	48
1	-17.0001	1.1203	1
1	-17.0151	1.1953	10
1	-19.599	1.2936	10
1	-18.7127	3.7286	1
1	-18.9007	3.7970	14
1	-20.1001	0.5016	1
1	-19.8873	0.5016	2
1	-19.6589	1.0815	11
1	-16.7995	1.0253	1
1	-16.6994	0.7277	5
1	-15.3851	0.7606	1
1	-15.3	1.0627	1
1	-14.9878	1.0706	6
1	-14.2501	-0.1015	47
1	-2.5001	-1.5984	1
1	-2.4001	-1.5984	13

```

19//num_ cornerpoints
0.2 0.0 //release_ distance*spacing_ hull
0.02 0.001 0.0 -0.1 //merg1*merg2*merg3*merg4
-4 -3 -1 -3 -1 //section_ color_ coding:drag*lift*moment*displ*rotation
  1    2.4      -1.5984    1
  1    2.5      -1.5984   47
  1   14.25     -0.1015    6
  1  14.9877    1.0706    1
  1  15.2999    1.0627    1
  1  15.385     0.7606    5
  1  16.6993    0.7277    1
  1  16.7994    1.0253   11
  1  19.6588    1.0815    2
  1  19.8872    0.5016    1
  1  20.1       0.5016   14
  1  18.9006    3.7970    1
  1  18.7126    3.7286   10
  1  19.5989    1.2936   10
  1  17.015     1.1953    1
  1  17         1.1203   48
  1   5         1.4203    1
  1   4.985     1.4953   10
  1   2.4003    1.5987   13

```

```

4//num_ cornerpoints
0.2 0.0 //release_ distance*spacing_ hull
0.02 0.001 0.0 -0.1 //merg1*merg2*merg3*merg4
-4 -3 -1 -3 -1 //section_ color_ coding:drag*lift*moment*displ*rotatio
  1  -12.6241    1.7598   10
  1  -10.113     1.8111   14
  1  -10.2253    5.2901   10
  1  -12.6538    5.2416   14

```

```

4//num_ cornerpoints
0.2 0.0 //release_ distance*spacing_ hull
0.02 0.001 0.0 -0.1 //merg1*merg2*merg3*merg4
-4 -3 -1 -3 -1 //section_ color_ coding:drag*lift*moment*displ*rotatio
  1   10.1129    1.8111   10
  1   12.624     1.7598   14
  1   12.6537    5.2416   10
  1   10.2252    5.2901   14

```

B Investigation of different Wind Barriers

B.1 Force and Moment Coefficients for Bridge Deck

Note for Table B.4, B.4 and B.6: Positive values denote a positive effect, meaning the decrease of the coefficients, which results in lower loads on the bridge deck. Vice versa, negative values express a negative effect given by the increase of the coefficient.

B.2 Force and Moment Coefficients for Windward Vehicle

Note for Table B.10, B.11 and B.12: Positive values denote a positive effect, meaning the decrease of the coefficients, which results in lower loads on the bridge deck. Vice versa, negative values express a negative effect given by the increase of the coefficient.

B.3 Force and Moment Coefficients for Leeward Vehicle

Note for Table B.16, B.17 and B.18: Positive values denote a positive effect, meaning the decrease of the coefficients, which results in lower loads on the bridge deck. Vice versa, negative values express a negative effect given by the increase of the coefficient.

Table B.1: Lift coefficients of Model B1 to B5 for bridge deck

LIFT COEFFICIENT OF BRIDGE DECK						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	-1.90	-2.11	-2.08	-2.08	-2.36
	MAX	0.16	0.21	-0.05	0.00	-0.22
	ST. DEVI.	0.38	0.40	0.36	0.42	0.35
	MEAN	-1.02	-1.07	-1.14	-1.23	-1.32
-7.5	MIN	-2.03	-2.25	-2.07	-2.29	-2.48
	MAX	0.01	0.06	0.12	0.07	-0.09
	ST. DEVI.	0.41	0.41	0.39	0.42	0.4
	MEAN	-0.97	-1.04	-1.10	-1.12	-1.19
-5	MIN	-1.91	-1.99	-2.11	-2.05	-2.21
	MAX	0.29	0.16	-0.01	0.41	0.23
	ST. DEVI.	0.41	0.37	0.38	0.42	0.39
	MEAN	-0.89	-0.99	-1.03	-1.06	-1.07
-2.5	MIN	-1.59	-1.69	-1.73	-1.54	-1.92
	MAX	0.41	0.03	0.33	0.22	0.41
	ST. DEVI.	0.38	0.34	0.36	0.30	0.35
	MEAN	-0.82	-0.88	-0.8	-0.75	-0.97
0	MIN	-1.44	-1.39	-1.38	-1.36	-1.51
	MAX	0.38	0.40	0.14	0.36	0.15
	ST. DEVI.	0.26	0.28	0.25	0.25	0.29
	MEAN	-0.70	-0.64	-0.69	-0.70	-0.73
2.5	MIN	-0.79	-0.80	-0.76	-0.84	-1.02
	MAX	0.89	0.94	0.93	0.81	0.96
	ST. DEVI.	0.31	0.32	0.27	0.26	0.29
	MEAN	0.02	0.01	-0.15	-0.15	-0.11
5	MIN	-0.43	-0.44	-0.41	-0.40	-0.71
	MAX	1.10	1.12	1.17	0.92	1.07
	ST. DEVI.	0.32	0.33	0.34	0.31	0.34
	MEAN	0.17	0.23	0.22	0.12	0.14
7.5	MIN	-0.38	-0.29	-0.34	-0.33	-0.36
	MAX	1.34	1.42	1.43	1.27	1.40
	ST. DEVI.	0.33	0.33	0.35	0.33	0.35
	MEAN	0.35	0.37	0.39	0.35	0.33
10	MIN	-0.20	-0.19	-0.24	-0.23	-0.30
	MAX	1.37	1.43	1.28	1.34	1.27
	ST. DEVI.	0.30	0.33	0.29	0.31	0.32
	MEAN	0.55	0.54	0.52	0.50	0.41

Table B.2: Drag coefficients of Model B1 to B5 for bridge deck

DRAG COEFFICIENT OF BRIDGE DECK						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	3.49	3.9	3.48	3.86	3.18
	MAX	10.22	10.94	10.19	9.83	10.84
	ST. DEVI.	1.25	1.22	1.29	1.38	1.49
	MEAN	6.62	6.92	6.72	7.00	6.49
-7.5	MIN	2.87	2.89	2.65	2.85	2.93
	MAX	8.80	8.87	9.74	9.76	9.63
	ST. DEVI.	1.23	1.23	1.27	1.31	1.11
	MEAN	5.84	5.98	5.68	5.91	5.97
-5	MIN	2.61	2.41	1.95	2.41	2.36
	MAX	7.49	8.70	8.24	8.25	9.44
	ST. DEVI.	0.96	0.99	1.13	1.03	1.12
	MEAN	5.06	5.17	4.97	5.18	5.00
-2.5	MIN	2.12	2.23	1.51	-1.17	1.52
	MAX	7.08	7.83	7.38	10.25	8.60
	ST. DEVI.	0.87	0.94	0.89	1.44	0.89
	MEAN	4.19	4.48	4.18	4.3	4.41
0	MIN	1.08	1.29	1.12	1.29	1.78
	MAX	5.74	6.61	5.60	6.17	7.66
	ST. DEVI.	0.82	0.94	0.84	0.89	0.75
	MEAN	2.95	3.14	3.00	3.26	3.69
2.5	MIN	-0.23	0.08	0.14	0.71	-0.32
	MAX	4.55	5.24	4.53	3.85	6.86
	ST. DEVI.	0.73	0.74	0.57	0.52	0.75
	MEAN	2.03	2.10	2.10	2.03	2.05
5	MIN	0.64	0.70	0.33	0.41	0.26
	MAX	5.50	4.97	4.80	4.28	5.12
	ST. DEVI.	0.82	0.79	0.80	0.72	0.81
	MEAN	2.49	2.58	2.55	2.35	2.37
7.5	MIN	1.46	1.36	1.44	1.20	1.03
	MAX	6.08	6.46	6.39	5.69	6.44
	ST. DEVI.	0.96	0.96	0.94	0.94	1.00
	MEAN	3.12	3.28	3.26	3.02	3.08
10	MIN	1.99	2.22	2.34	1.81	1.85
	MAX	6.71	6.93	6.91	6.78	6.40
	ST. DEVI.	0.93	0.99	1.03	1.04	1.02
	MEAN	3.79	4.10	4.11	3.80	3.80

Table B.3: Moment coefficients of Model B1 to B5 for bridge deck

MOMENT COEFFICIENT OF BRIDGE DECK						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	-0.16	-0.11	-0.10	-0.13	-0.16
	MAX	0.35	0.33	0.29	0.33	0.30
	ST. DEVI.	0.09	0.08	0.08	0.09	0.09
	MEAN	0.11	0.14	0.13	0.13	0.08
-7.5	MIN	-0.11	-0.11	-0.13	-0.13	-0.13
	MAX	0.37	0.38	0.35	0.36	0.37
	ST. DEVI.	0.10	0.10	0.09	0.10	0.09
	MEAN	0.15	0.16	0.16	0.15	0.15
-5	MIN	-0.08	-0.09	-0.06	-0.06	-0.05
	MAX	0.43	0.42	0.41	0.41	0.40
	ST. DEVI.	0.10	0.10	0.10	0.10	0.09
	MEAN	0.18	0.21	0.2	0.2	0.19
-2.5	MIN	-0.01	0.05	0.04	0.03	-0.02
	MAX	0.44	0.43	0.49	0.42	0.46
	ST. DEVI.	0.09	0.09	0.09	0.06	0.09
	MEAN	0.22	0.27	0.27	0.23	0.25
0	MIN	-0.03	0.06	0.08	0.06	0.12
	MAX	0.48	0.51	0.49	0.49	0.48
	ST. DEVI.	0.08	0.08	0.08	0.08	0.07
	MEAN	0.20	0.24	0.26	0.26	0.30
2.5	MIN	-0.14	-0.13	-0.14	-0.08	-0.11
	MAX	0.37	0.39	0.41	0.42	0.48
	ST. DEVI.	0.12	0.12	0.11	0.10	0.10
	MEAN	0.09	0.12	0.19	0.20	0.20
5	MIN	-0.12	-0.13	-0.10	-0.06	-0.10
	MAX	0.37	0.40	0.42	0.37	0.50
	ST. DEVI.	0.11	0.13	0.13	0.13	0.12
	MEAN	0.13	0.13	0.14	0.17	0.16
7.5	MIN	-0.08	-0.09	-0.15	-0.09	-0.13
	MAX	0.35	0.42	0.42	0.41	0.44
	ST. DEVI.	0.11	0.12	0.13	0.11	0.13
	MEAN	0.17	0.17	0.17	0.16	0.21
10	MIN	-0.11	-0.04	-0.04	-0.08	-0.09
	MAX	0.34	0.37	0.41	0.37	0.44
	ST. DEVI.	0.10	0.10	0.11	0.11	0.11
	MEAN	0.17	0.19	0.22	0.19	0.23

Table B.4: Deviations of lift coefficients in relation to Model B1 for bridge deck

LIFT COEFFICIENT OF BRIDGE DECK					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	-0.05	-0.12	-0.21	-0.30
-7.5 °	0	-0.06	-0.12	-0.15	-0.22
-5 °	0	-0.10	-0.14	-0.16	-0.18
-2.5 °	0	-0.06	0.02	0.07	-0.15
0 °	0	0.07	0.01	0.01	-0.02
2.5 °	0	0.01	-0.13	-0.12	-0.09
5 °	0	-0.06	-0.04	0.06	0.03
7.5 °	0	-0.02	-0.04	0.00	0.01
10 °	0	0.01	0.03	0.05	0.13

Table B.5: Deviations of drag coefficients in relation to Model B1 for bridge deck

DRAG COEFFICIENT OF BRIDGE DECK					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	-0.29	-0.10	-0.37	0.13
-7.5 °	0	-0.14	0.16	-0.07	-0.13
-5 °	0	-0.11	0.09	-0.12	0.06
-2.5 °	0	-0.29	0.01	-0.11	-0.22
0 °	0	-0.19	-0.06	-0.32	-0.74
2.5 °	0	-0.07	-0.08	0.00	-0.03
5 °	0	-0.08	-0.06	0.14	0.12
7.5 °	0	-0.16	-0.14	0.10	0.04
10 °	0	-0.31	-0.32	-0.01	-0.02

Table B.6: Deviations of moment coefficients in relation to Model B1 for bridge deck

MOMENT COEFFICIENT OF BRIDGE DECK					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	-0.04	-0.03	-0.02	0.02
-7.5 °	0	-0.02	-0.01	0.00	0.00
-5 °	0	-0.03	-0.02	-0.02	-0.01
-2.5 °	0	-0.05	-0.04	-0.01	-0.03
0 °	0	-0.04	-0.06	-0.06	-0.09
2.5 °	0	-0.03	-0.10	-0.11	-0.11
5 °	0	0.00	-0.02	-0.04	-0.03
7.5 °	0	0.00	0.00	0.01	-0.04
10 °	0	-0.02	-0.06	-0.02	-0.06

Table B.7: Lift coefficients of Model B1 to B5 for windward vehicle

LIFT COEFFICIENT OF WINDWARD VEHICLE						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	-0.69	-0.83	-0.75	-0.66	-0.71
	MAX	0.93	0.82	0.96	1.18	1.21
	ST. DEVI.	0.23	0.25	0.27	0.26	0.36
	MEAN	0.06	0.06	0.13	0.17	0.22
-7.5	MIN	-0.74	-0.82	-0.92	-0.85	-0.76
	MAX	0.68	0.71	0.75	0.87	1.15
	ST. DEVI.	0.21	0.22	0.23	0.24	0.30
	MEAN	-0.02	-0.09	0.00	0.00	0.09
-5	MIN	-0.89	-0.89	-0.96	-0.92	-0.86
	MAX	0.52	0.52	0.52	0.66	0.70
	ST. DEVI.	0.19	0.20	0.20	0.21	0.23
	MEAN	-0.07	-0.12	-0.10	-0.07	-0.05
-2.5	MIN	-0.88	-0.83	-0.76	-0.74	-0.89
	MAX	0.49	0.45	0.43	0.47	0.52
	ST. DEVI.	0.16	0.16	0.17	0.17	0.18
	MEAN	-0.08	-0.14	-0.13	-0.07	-0.08
0	MIN	-0.61	-0.70	-0.71	-0.61	-0.61
	MAX	0.39	0.30	0.35	0.42	0.39
	ST. DEVI.	0.15	0.15	0.16	0.16	0.16
	MEAN	-0.12	-0.20	-0.19	-0.13	-0.10
2.5	MIN	-0.77	-0.85	-0.99	-0.81	-0.89
	MAX	0.50	0.75	0.48	0.51	0.34
	ST. DEVI.	0.19	0.20	0.20	0.19	0.18
	MEAN	-0.15	-0.11	-0.16	-0.17	-0.20
5	MIN	-0.93	-0.75	-0.85	-0.86	-0.83
	MAX	0.60	0.85	0.84	0.89	0.81
	ST. DEVI.	0.22	0.26	0.25	0.24	0.24
	MEAN	-0.05	-0.02	-0.02	-0.08	-0.06
7.5	MIN	-0.95	-1.22	-1.33	-0.82	-0.77
	MAX	0.91	1.37	1.13	1.07	0.83
	ST. DEVI.	0.26	0.34	0.35	0.30	0.28
	MEAN	0.00	-0.01	0.01	0.00	-0.03
10	MIN	-0.89	-1.49	-1.53	-1.22	-1.32
	MAX	1.09	1.61	1.75	1.23	1.21
	ST. DEVI.	0.30	0.41	0.44	0.36	0.37
	MEAN	0.03	0.09	0.12	0.05	0.00

Table B.8: Drag coefficients of Model B1 to B5 for windward vehicle

DRAG COEFFICIENT OF WINDWARD VEHICLE						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	-0.27	-0.10	-0.17	-0.05	-0.03
	MAX	1.22	1.40	1.38	1.42	1.60
	ST. DEVI.	0.22	0.22	0.25	0.24	0.24
	MEAN	0.56	0.59	0.64	0.68	0.83
-7.5	MIN	-0.30	-0.37	-0.43	-0.31	-0.11
	MAX	1.10	0.99	1.34	1.33	1.39
	ST. DEVI.	0.23	0.25	0.26	0.23	0.23
	MEAN	0.39	0.35	0.49	0.49	0.61
-5	MIN	-0.37	-0.47	-0.43	-0.46	-0.33
	MAX	0.90	0.84	1.01	1.09	1.00
	ST. DEVI.	0.21	0.23	0.22	0.24	0.20
	MEAN	0.18	0.12	0.18	0.30	0.36
-2.5	MIN	-0.69	-0.59	-0.60	-0.47	-0.37
	MAX	0.53	0.63	0.44	0.62	0.62
	ST. DEVI.	0.15	0.14	0.14	0.16	0.13
	MEAN	-0.07	-0.14	-0.09	-0.01	0.08
0	MIN	-0.61	-0.71	-0.69	-0.7	-0.71
	MAX	0.22	0.19	0.20	0.20	0.21
	ST. DEVI.	0.11	0.12	0.11	0.12	0.13
	MEAN	-0.27	-0.34	-0.34	-0.28	-0.25
2.5	MIN	-0.82	-1.03	-1.03	-0.99	-0.91
	MAX	0.32	0.44	0.41	0.13	0.23
	ST. DEVI.	0.18	0.24	0.24	0.18	0.17
	MEAN	-0.35	-0.31	-0.36	-0.41	-0.39
5	MIN	-1.06	-1.27	-1.32	-1.08	-1.15
	MAX	0.80	0.97	0.78	0.63	0.42
	ST. DEVI.	0.28	0.34	0.33	0.27	0.26
	MEAN	-0.29	-0.25	-0.30	-0.35	-0.29
7.5	MIN	-1.31	-1.73	-1.67	-1.4	-1.71
	MAX	0.92	1.34	1.76	0.97	1.26
	ST. DEVI.	0.32	0.54	0.52	0.40	0.40
	MEAN	-0.27	-0.20	-0.21	-0.27	-0.26
10	MIN	-1.61	-1.63	-1.62	-1.64	-1.59
	MAX	1.27	1.54	1.57	1.54	1.41
	ST. DEVI.	0.45	0.56	0.57	0.55	0.53
	MEAN	-0.30	-0.19	-0.25	-0.23	-0.23

Table B.9: Moment coefficients of Model B1 to B5 for windward vehicle

MOMENT COEFFICIENT OF WINDWARD VEHICLE						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	-1.68	-1.92	-2.11	-2.28	-2.59
	MAX	0.61	0.59	0.55	0.64	0.64
	ST. DEVI.	0.40	0.42	0.48	0.47	0.60
	MEAN	-0.55	-0.60	-0.65	-0.68	-0.85
-7.5	MIN	-0.82	-0.87	-1.01	-1.28	-1.20
	MAX	0.74	0.68	0.82	0.60	0.85
	ST. DEVI.	0.25	0.25	0.28	0.28	0.34
	MEAN	-0.09	-0.10	-0.15	-0.16	-0.17
-5	MIN	-0.47	-0.51	-0.46	-0.49	-0.42
	MAX	0.69	0.65	0.60	0.78	0.86
	ST. DEVI.	0.14	0.16	0.15	0.17	0.18
	MEAN	0.16	0.10	0.14	0.16	0.20
-2.5	MIN	-0.70	-0.66	-0.65	-0.53	-0.40
	MAX	0.86	0.86	0.68	0.83	0.86
	ST. DEVI.	0.20	0.18	0.20	0.20	0.18
	MEAN	0.07	-0.03	0.02	0.14	0.21
0	MIN	-1.01	-1.17	-1.13	-1.07	-1.03
	MAX	0.52	0.42	0.36	0.50	0.47
	ST. DEVI.	0.22	0.26	0.21	0.23	0.23
	MEAN	-0.30	-0.36	-0.45	-0.34	-0.18
2.5	MIN	-1.73	-1.94	-2.03	-2.09	-1.86
	MAX	0.56	0.81	0.96	0.39	0.48
	ST. DEVI.	0.36	0.50	0.49	0.46	0.47
	MEAN	-0.72	-0.44	-0.72	-0.67	-0.50
5	MIN	-2.65	-3.46	-3.62	-2.91	-2.90
	MAX	2.34	2.73	2.09	2.02	1.48
	ST. DEVI.	0.75	0.88	0.85	0.73	0.68
	MEAN	-0.74	-0.57	-0.73	-0.85	-0.57
7.5	MIN	-4.40	-5.60	-5.69	-4.31	-5.33
	MAX	3.57	5.35	6.51	3.37	4.95
	ST. DEVI.	1.00	1.79	1.76	1.30	1.26
	MEAN	-0.82	-0.53	-0.58	-0.88	-0.66
10	MIN	-6.18	-6.06	-6.40	-6.32	-6.04
	MAX	6.08	7.06	7.43	7.10	6.33
	ST. DEVI.	1.80	2.27	2.33	2.27	2.02
	MEAN	-1.06	-0.62	-0.92	-0.75	-0.63

Table B.10: Deviations of lift coefficients in relation to Model B1 for windward vehicle

LIFT COEFFICIENT OF LEEWARD VEHICLE					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	-0.01	-0.07	-0.11	-0.17
-7.5 °	0	-0.06	0.02	0.02	-0.07
-5 °	0	-0.05	-0.03	0.00	0.02
-2.5 °	0	-0.06	-0.05	0.01	0.00
0 °	0	-0.07	-0.06	-0.01	0.02
2.5 °	0	0.04	0.00	-0.02	-0.05
5 °	0	0.03	0.03	-0.03	-0.01
7.5 °	0	-0.01	-0.01	0.00	-0.03
10 °	0	-0.05	-0.08	-0.02	0.03

Table B.11: Deviations of drag coefficients in relation to Model B1 for windward vehicle

DRAG COEFFICIENT OF LEEWARD VEHICLE					
Angle of attack	B1	B2	B3	B4	B5
-10°	0	-0.02	-0.08	-0.12	-0.27
-7.5°	0	0.04	-0.10	-0.10	-0.22
-5°	0	0.05	0.00	-0.12	-0.18
-2.5°	0	-0.07	-0.03	0.06	-0.01
0°	0	-0.07	-0.07	-0.01	0.02
2.5 °	0	0.05	0.00	-0.06	-0.03
5 °	0	0.04	-0.01	-0.06	0.00
7.5 °	0	0.07	0.06	0.00	0.01
10 °	0	0.11	0.04	0.07	0.07

Table B.12: Deviations of moment coefficients in relation to Model B1 for windward vehicle

MOMENT COEFFICIENT OF LEEWARD VEHICLE					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	-0.05	-0.10	-0.13	-0.30
-7.5 °	0	-0.01	-0.06	-0.07	-0.08
-5 °	0	0.06	0.02	0.00	-0.04
-2.5 °	0	0.04	0.05	-0.06	-0.13
0 °	0	-0.06	-0.15	-0.04	0.12
2.5 °	0	0.28	0.00	0.05	0.21
5 °	0	0.16	0.01	-0.12	0.17
7.5 °	0	0.29	0.24	-0.06	0.16
10 °	0	0.44	0.14	0.31	0.43

Table B.13: Lift coefficients of Model B1 to B5 for leeward vehicle

LIFT COEFFICIENT OF LEEWARD VEHICLE						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	-19.74	-21.1	-19.72	-17.57	-22.26
	MAX	10.88	9.95	11.63	9.88	11.21
	ST. DEVI.	6.07	5.92	5.82	5.76	6.01
	MEAN	0.68	0.63	0.63	0.61	0.54
-7.5	MIN	-22.51	-20.69	-23.12	-21.60	-26.00
	MAX	11.69	12.13	10.79	8.84	12.72
	ST. DEVI.	7.05	6.41	7.07	5.91	7.30
	MEAN	0.54	0.51	0.46	0.54	0.46
-5	MIN	-22.72	-22.73	-19.34	-20.65	-24.23
	MAX	9.48	13.60	11.55	15.30	9.71
	ST. DEVI.	5.97	6.18	5.09	6.20	5.91
	MEAN	0.41	0.35	0.38	0.41	0.40
-2.5	MIN	-15.77	-18.24	-15.24	-18.37	-13.52
	MAX	16.36	11.48	12.15	11.58	11.48
	ST. DEVI.	4.01	4.23	4.00	4.37	3.93
	MEAN	0.35	0.27	0.27	0.21	0.28
0	MIN	-11.86	-11.33	-12.79	-12.05	-13.70
	MAX	6.18	7.41	6.23	7.49	6.62
	ST. DEVI.	3.55	3.46	3.27	3.82	3.53
	MEAN	0.15	0.11	0.07	0.12	0.16
2.5	MIN	-6.65	-8.49	-7.82	-8.89	-10.97
	MAX	10.01	9.93	9.09	7.44	7.25
	ST. DEVI.	3.36	3.81	3.62	3.48	3.67
	MEAN	0.01	0.01	0.01	0.02	0.01
5	MIN	-9.39	-9.21	-10.07	-10.19	-10.01
	MAX	10.61	10.41	11.58	10.36	10.19
	ST. DEVI.	4.58	4.48	4.79	4.69	4.92
	MEAN	0.04	0.07	0.07	0.04	0.01
7.5	MIN	-6.84	-7.91	-7.5	-7.67	-9.44
	MAX	12.39	11.42	11.41	13.08	14.43
	ST. DEVI.	4.46	4.47	4.75	4.80	4.89
	MEAN	0.08	0.08	0.09	0.05	0.04
10	MIN	-5.91	-5.9	-7.54	-7.10	-7.02
	MAX	12.44	12.28	12.20	12.78	12.27
	ST. DEVI.	4.37	4.74	4.79	4.52	4.56
	MEAN	0.12	0.11	0.11	0.13	0.07

Table B.14: Drag coefficients of Model B1 to B5 for leeward vehicle

DRAG COEFFICIENT OF LEEWARD VEHICLE						
ANGLE		B1	B2	B3	B4	B5
-10	MIN	-1.70	-1.32	-1.81	-1.67	-1.66
	MAX	4.37	4.61	4.52	4.92	5.23
	ST. DEVI.	1.18	1.09	1.13	1.12	1.24
	MEAN	1.37	1.39	1.53	1.31	1.23
-7.5	MIN	-1.45	-1.93	-1.47	-1.57	-1.22
	MAX	4.55	3.98	4.53	4.90	5.53
	ST. DEVI.	1.13	1.03	1.08	1.07	1.12
	MEAN	1.29	1.34	1.37	1.36	1.20
-5	MIN	-0.86	-1.24	-1.03	-1.35	-0.96
	MAX	4.11	4.79	3.34	3.67	4.64
	ST. DEVI.	0.92	0.90	0.79	0.89	0.97
	MEAN	1.10	1.10	1.23	1.14	1.17
-2.5	MIN	-1.12	-1.26	-1.22	-1.06	-0.84
	MAX	2.76	3.25	2.81	3.02	2.23
	ST. DEVI.	0.58	0.57	0.54	0.67	0.57
	MEAN	1.03	0.96	0.99	0.86	1.00
0	MIN	-1.15	-1.23	-1.19	-1.21	-1.26
	MAX	1.02	0.96	1.14	2.02	1.87
	ST. DEVI.	0.36	0.35	0.34	0.44	0.43
	MEAN	0.70	0.60	0.50	0.69	0.79
2.5	MIN	-1.24	-1.18	-1.25	-1.20	-1.13
	MAX	1.03	0.70	1.22	1.12	1.36
	ST. DEVI.	0.32	0.32	0.37	0.38	0.37
	MEAN	0.08	0.13	0.12	0.14	0.18
5	MIN	-1.13	-1.08	-1.36	-1.23	-1.33
	MAX	1.28	1.25	1.56	1.57	1.59
	ST. DEVI.	0.44	0.44	0.49	0.50	0.53
	MEAN	0.06	0.08	0.06	0.13	0.11
7.5	MIN	-1.03	-1.17	-1.18	-1.13	-1.20
	MAX	1.76	1.76	1.75	1.76	1.97
	ST. DEVI.	0.50	0.51	0.57	0.59	0.58
	MEAN	-0.04	0.05	-0.05	-0.09	0.02
10	MIN	-1.04	-0.98	-1.14	-1.20	-1.01
	MAX	1.95	2.01	1.77	2.04	2.27
	ST. DEVI.	0.61	0.61	0.62	0.64	0.66
	MEAN	-0.12	-0.17	-0.14	-0.07	-0.05

Table B.15: Moment coefficients of Model B1 to B5 for leeward vehicle

MOMENT COEFFICIENT OF LEEWARD VEHICLE						
-10	MIN	-1.61	-0.24	-2.52	-2.41	-2.07
	MAX	11.56	10.01	12.18	9.94	9.89
	ST. DEVI.	2.06	1.82	2.12	1.95	1.94
	MEAN	5.80	5.79	6.37	5.45	5.11
-7.5	MIN	-1.97	-2.48	-3.02	-1.43	-2.61
	MAX	9.50	9.50	8.97	8.97	9.33
	ST. DEVI.	1.84	1.83	1.98	1.61	1.76
	MEAN	4.55	4.57	4.66	4.70	4.11
-5	MIN	-1.81	-3.05	-2.46	-2.13	-2.38
	MAX	6.52	7.07	6.54	7.07	6.73
	ST. DEVI.	1.32	1.73	1.44	1.41	1.31
	MEAN	2.99	3	3.28	3.09	3.14
-2.5	MIN	-2.82	-2.05	-2.65	-2.02	-2.06
	MAX	4.75	4.83	4.75	4.38	3.82
	ST. DEVI.	0.90	1.12	1.12	1.05	0.99
	MEAN	2.10	1.92	1.98	1.79	1.84
0	MIN	-2.60	-2.65	-2.41	-2.64	-2.16
	MAX	2.49	2.31	2.24	2.43	2.43
	ST. DEVI.	0.90	0.93	0.97	0.92	0.82
	MEAN	0.93	0.63	0.65	0.92	0.68
2.5	MIN	-1.76	-1.91	-2.10	-1.70	-1.99
	MAX	1.06	1.26	1.04	1.15	1.27
	ST. DEVI.	0.61	0.50	0.64	0.56	0.52
	MEAN	-0.03	0.01	0.00	0.01	0.03
5	MIN	-0.72	-0.79	-0.63	-0.78	-0.78
	MAX	1.05	1.27	1.29	0.98	0.88
	ST. DEVI.	0.25	0.23	0.21	0.23	0.22
	MEAN	-0.04	-0.05	-0.05	-0.08	-0.05
7.5	MIN	-1.52	-1.98	-1.78	-1.60	-1.42
	MAX	3.00	2.37	2.98	2.60	3.21
	ST. DEVI.	0.72	0.67	0.77	0.77	0.71
	MEAN	0.03	-0.06	0.06	0.07	0.00
10	MIN	-2.80	-2.54	-2.86	-3.12	-2.60
	MAX	4.17	4.37	4.14	4.28	4.39
	ST. DEVI.	1.29	1.25	1.20	1.24	1.22
	MEAN	0.13	0.26	0.19	0.07	0.07

Table B.16: Deviations of lift coefficients in relation to Model B1 for leeward vehicle

LIFT COEFFICIENT OF LEEWARD VEHICLE					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	0.04	0.05	0.06	0.13
-7.5 °	0	0.03	0.07	0.00	0.08
-5 °	0	0.06	0.03	0.00	0.01
-2.5 °	0	0.08	0.08	0.14	0.07
0 °	0	0.04	0.08	0.03	-0.01
2.5 °	0	0.00	0.00	-0.01	0.00
5 °	0	-0.03	-0.03	-0.01	0.02
7.5 °	0	0.00	0.00	0.04	0.04
10 °	0	0.01	0.01	-0.01	0.04

Table B.17: Deviations of drag coefficients in relation to Model B1 for leeward vehicle

DRAG COEFFICIENT OF LEEWARD VEHICLE					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	-0.02	-0.16	0.06	0.14
-7.5 °	0	-0.04	-0.07	-0.07	0.09
-5 °	0	0.00	-0.13	-0.05	-0.07
-2.5 °	0	0.07	0.03	0.17	0.02
0 °	0	0.10	0.19	0.01	-0.09
2.5 °	0	-0.05	-0.03	-0.06	-0.09
5 °	0	-0.02	0.00	-0.08	-0.06
7.5 °	0	-0.01	-0.01	-0.05	0.02
10 °	0	-0.04	-0.02	0.06	0.07

Table B.18: Deviations of moment coefficients in relation to Model B1 for leeward vehicle

MOMENT COEFFICIENT OF LEEWARD VEHICLE					
Angle of attack	B1	B2	B3	B4	B5
-10 °	0	0.01	-0.57	0.35	0.69
-7.5 °	0	-0.01	-0.11	-0.14	0.44
-5 °	0	-0.01	-0.29	-0.10	-0.15
-2.5 °	0	0.18	0.12	0.31	0.26
0 °	0	0.31	0.28	0.01	0.26
2.5 °	0	0.02	0.02	0.02	-0.01
5 °	0	-0.01	-0.01	-0.04	-0.01
7.5 °	0	-0.03	-0.03	-0.04	0.03
10 °	0	-0.13	-0.06	0.06	0.06

Bibliography

- [1] C. Geurts. Technical note: The use of wind tunnel experiments for wind loads on structures. *TNO built environment and geosciences*, Delft, Netherlands.
- [2] Claes Dyrbye and Svend Ole Hansen. *Wind loads on structures*. John Wiley & Sons, Chichester, 1997.
- [3] You-Lin Xu. *Wind effects on cable-supported bridges*. John Wiley & Sons, Singapore, 2013.
- [4] Ion Paraschivoiu. *Subsonic Aerodynamics*. Presses internationales Polytechnique, Montreal, 2003.
- [5] Emil Simiu and Toshio Miyata. *Design of buildings and bridges for wind: a practical guide for ASCE-7 standard users and designers of special structures*. John Wiley & Sons, Hoboken, New Jersey, 2006.
- [6] Merle C. Potter and D. C. Wiggert. *Schaum's outline of fluid mechanics*. McGraw-Hill Contemporary Learning and McGraw-Hill, Dubuque, Iowa and New York, 2008.
- [7] Robert A. Granger. *Fluid mechanics*. Dover Publications, New York, 1995.
- [8] George Keith Batchelor. *An introduction to fluid dynamics*. Cambridge University Press, Cambridge, 2000.
- [9] R. I. Issa. Rise of total pressure in frictional flow. *AIAA Journal*, 33(4):772–774, 1995.
- [10] Stuart E. Norris. The effect of nonuniform viscosity on stagnation point pressure. *Journal of Fluids Engineering*, 133(4):1–4, 2011.
- [11] Oleg Zikanov. *Essential computational fluid dynamics*. John Wiley & Sons, Hoboken, New Jersey, 2010.
- [12] Stefan Lecheler. *Numerische Strömungsberechnung: Schneller Einstieg durch anschauliche Beispiele*. Vieweg+Teubner Verlag and Springer Fachmedien Wiesbaden GmbH, Wiesbaden, 2011.
- [13] John F. Wendt and John David Anderson. *Computational Fluid Dynamics - An introduction: With contributions by John D. Anderson*. Springer, Berlin, Heidelberg, 2009.
- [14] J. Franke, C. Hirsch, A. G. Jensen, H. W. Krüs, M. Schatzmann, P. S. Westbury, S. D. Miles, J. A. Wisse, and N. G. Wright. *Recommendations on the use of CFD in wind engineering*. Proceedings of the International Conference on Urban Wind Engineering and Building Aerodynamics. Karman Institute, Sint-Genesius-Rode, Belgium, 2004.
- [15] Guido Morgenthal. *Aerodynamic Analysis of Structures Using High-resolution Vortex Particle Methods*. University of Cambridge, UK, Ph.D. Dissertation, 2002.
- [16] Heinz Schade, Frank Kameier, and Ewald Kunz. *Strömungslehre*. De-Gruyter-Lehrbuch. Berlin and New York, 2007.
- [17] David J. Tritton. *Physical Fluid Dynamics*. University Press, Cambridge, 1977.
- [18] Oskar Karl Gustav Tietjens and Ludwig Prandtl. *Applied hydro-and aeromechanics: based on lectures of L. Prandtl*. Dover Publications, New York, 1957.
- [19] Eurocode 1. *DIN EN 1991-1-4:2010-12 Actions on structures - Part 1-4: General actions - Wind actions*.

- [20] Ulf Mohrlök. *Einführung in die Hydromechanik*. KIT Scientific Publishing, Karlsruhe, 2007.
- [21] Mohammad Sadraey. *Aircraft performance analysis*. VDM Verlag Dr. Müller, Saarbrücken, 2009.
- [22] Da-hai Yu and A. Kareem. Two-dimensional simulation of flow around rectangular prisms. *Journal of Wind Engineering and Industrial Aerodynamics*, 62(2):131–161, 1996.
- [23] Kai Liaw. *Simulation of flow around bluff bodies and bridge deck sections using CFD*. University of Nottingham, Ph.D. Dissertation, 2005.
- [24] W. Fritz, B. Bienkiewicz, B. Cui, O. Flamand, T. Ho, H. Kikitsu, C. Letchford, and E. Simiu. International comparison of wind tunnel estimates of wind effects on low-rise buildings: Test-related uncertainties. *Journal of structural engineering*, 134(12):1887–1890, 2008.
- [25] M. Ogueta-Gutiérrez, S. Franchini, and G. Alonso. Effects of bird protection barriers on the aerodynamic and aeroelastic behaviour of high speed train bridges. *Engineering Structures*, 81:22–34, 2014.
- [26] Gordon M. Heisler and David R. Dewalle. Effects of windbreak structure on wind flow. *Agriculture, Ecosystems; Environment*, 22-23:41–69, 1988.
- [27] Simon James. Information note: Bridge windshielding. 2007.
- [28] Hyoung-Bum Kim and Sang-Joon Lee. Hole diameter effect on flow characteristics of wake behind porous fences having the same porosity. *Fluid Dynamics Research*, 28(6):449–464, 2001.
- [29] Hrvoje Kozmar, Lorenzo Procino, Alessandra Borsani, and Gianni Bartoli. Optimizing height and porosity of roadway wind barriers for viaducts and bridges. *Engineering Structures*, 81:49–61, 2014.
- [30] Marijo Telenta, Jožef Duhovnik, Franc Kosel, and Viktor Šajn. Numerical and experimental study of the flow through a geometrically accurate porous wind barrier model. *Journal of Wind Engineering and Industrial Aerodynamics*, 124:99–108, 2014.
- [31] Soon-Duck Kwon, Dong Hyawn Kim, Seung Ho Lee, and Ho Sung Song. Design criteria of wind barriers for traffic -part 1: Wind barrier performance. *Wind and Structures An International Journal*, 14(1):55–70, 2011.

Glossary

a_x, a_y, a_z	Acceleration components
A_T	Tributary area
α	Angle of attack
B	Width of bridge deck
C_p	Pressure coefficient
C_D	Drag coefficient
C_L	Lift coefficient
C_M	Moment coefficient
C_N	Normal force coefficient
c_p, c_v	Specific heat capacity (for constant pressure/volume)
d_x, d_y, d_z	Dimensions of infinitesimal fluid element
e	Specific internal energy
E_{in}	Internal energy
E_{kin}	Kinetic energy
E_{pot}	Potential energy
E_{tot}	Total energy
f	Frequency of vortex shedding
F_D	Drag force
F_H	Horizontal force
F_L	Lift force
F_V	Vertical force
g	Gravitational acceleration
H	Height of the section
γ_0	Initial surface vorticity
Γ	Vortex strength
h	Specific enthalpy
L	Longitudinal dimension of the section
λ	Thermal conductivity
m	Fluid mass

M	Moment
μ	Dynamic viscosity
n_0	Normal vector
N_s	Number of vortex particles
∇	Nabla operator
ν	Kinematic viscosity
p	Pressure
p_∞	Free stream pressure
ϕ	Velocity potential
ψ	Stream function
$\dot{Q}$	Heat flux
$\dot{q}_b$	Specific heat flux of boundary
$\dot{q}_L$	Thermal conduction
$\dot{q}_S$	Thermal radiation
R	Gas constant
Re	Reynolds number
ρ	Density
t	Time
T	Temperature
τ_{xx}	Normal stresses in x-direction perpendicular to the x-constant area $dy \cdot dz$
τ_{xy}	Shear stresses in y-direction along the x-constant area $dy \cdot dz$
τ_{xz}	Shear stresses in z-direction along the x-constant area $dy \cdot dz$
τ_{yx}	Shear stresses in x-direction along the y-constant area $dx \cdot dz$
τ_{yy}	Normal stresses in y-direction perpendicular to the y-constant area $dx \cdot dz$
τ_{yz}	Shear stresses in z-direction along the y-constant area $dx \cdot dz$
τ_{zx}	Shear stresses in x-direction along the z-constant area $dx \cdot dy$
τ_{zy}	Shear stresses in y-direction along the z-constant area $dx \cdot dy$
τ_{zz}	Normal stresses in z-direction perpendicular to the z-constant area $dx \cdot dy$
u_∞	Free stream velocity
$\vec{u} = u, v, w$	Velocity components
V	Volume

w	Vorticity
W	Width of the wake
$\dot{W}$	Rate of working done on volume element
x, y, z	Spatial coordinates