Improving Aerodynamic Performance of a Truck: a Numerical Based Analysis

JOHAN MALMBERG

Master’s Thesis at KTH: Mechanical Department
Supervisor: Mihai Mihaescu
Examiner: Laszlo Fuchs

TRITA xxx 2015-06
Abstract

With a change in EU-legislation regarding the dimensions of heavy trucks [1] coming up, specifically allowing aerodynamic devices fitted to the back of trailers, the transportation industry is set to largely decrease its environmental impact in the near future. These aerodynamic devices have been researched for quite some time [2] but have not yet gained widespread market acceptance, partly because of the transportation industry’s complex owner structures and partly because of size regulations of the past. Though with this new legislation and other expected new EU-legislation on CO₂ emissions, development of these aerodynamic devices looks set to become a field of intense study.

The study explores different options regarding the influence of changing the rear shape of a generalized ground transportation system (GTS), a simplified model of a truck geometry. By changing the rear shape the drag force induced by the oncoming air flow was reduced, and the overall coefficient of drag ($C_D$) value lowered. There is also an investigation as to how an added suction slot, and that slots location, affects the $C_D$.

Computational Fluid Dynamics (CFD) in the form of the commercial computer software STAR-CCM+ was used to simulate the flow around the GTS, and the results were verified with similar studies and experiments [3] on the same geometry. RANS equations and Menter k-ω SST turbulence model was used for all simulations.

The results show that $C_D$ can be lowered by 21% compared to a baseline case. Further on the added suction slots can reduce drag, but depending on slot location also can add to the drag force experienced by the GTS.
Acknowledgements

I would like to thank my supervisor Mihai Mihaescu for all his encouragement and feedback during my work with this thesis. Elias Sundström has been invaluable as support for all questions regarding CFD.

I have Anna Svensson, Peter Georén and Per Gyllenspetz to thank for getting me started on the thesis idea.

Thanks to Mustafa Güdüçü for bringing some spirit to our work room, without him the place would have been way too quiet.

Finally, a huge thank you to my family who have been supportive all throughout my, unbelievably long, school career as they have never given up on me.

And Kristina, who I can’t live without.
## Contents

1 Introduction ........................................ 1
   1.1 Background .................................. 2
       1.1.1 Historical perspective ................. 2
   1.2 Motivation .................................. 4

2 Bluff body aerodynamics ............................ 5
   2.1 Flow regimes .................................. 5
   2.2 Flow characteristics ........................... 6
   2.3 Forces, coefficients and representative scales 9

3 Numerical approach ................................ 11
   3.1 Flow governing equations ..................... 11
   3.2 Turbulence modelling ........................... 12
       3.2.1 Wall treatment .......................... 13

4 Case description & setup ........................... 15
   4.1 Mesh .......................................... 17
   4.2 Initial & boundary conditions ................. 18
   4.3 Solver ........................................ 19

5 Results ............................................. 23
   5.1 Verification ................................... 23
       5.1.1 Mesh quality .............................. 23
       5.1.2 Grid convergence study: Baseline configuration 24
   5.2 Case 1: Baseline ................................ 26
   5.3 Case 2: Boat-tails .............................. 29
   5.4 Case 3: Rounded boat-tail with suction slots 30

6 Conclusions & Discussion ............................ 47

7 Future work ........................................ 49

A The Trailair project ................................ 51
B  Case 2: Additional plots 53
C  Case 3: Additional plots 69
D  GTS Geometries 75
Bibliography 77
Chapter 1

Introduction

Figure 1.1: The MAN Concept S and Krone Aeroliner truck and trailer combo gives an idea of what the future brings to the aerodynamics of heavy trucks. Notice the tapering and extension at the back of the trailer. Image by Krone (http://www.krone-trailer.com/english/products/future-projects/aeroliner-concept-s/).

This thesis aims to reduce the drag of the generalized shape of a truck by adding a geometry known as a boat-tail to the back. All calculations are done using the commercial software STAR-CCM+ which uses CFD to simulate the airflow around the geometry. The thesis gives a brief account of the history of vehicle aerodynamics and continues on with a motivation as to why the thesis was carried out. Further on the concepts of aerodynamics and CFD are explained in short, and then comes the description of the different cases that were investigated. The results are presented, followed by conclusions and discussion and finally some topics for future work are proposed.
1.1 Background

1.1.1 Historical perspective

The field of aerodynamics traces its origins back to the first wind tunnels of the late 1890s. The Wright brothers, credited with building the first aeroplane, built a wind tunnel to examine lift for wing profiles, and in Sweden Carl Richard Nyberg used a wind tunnel to construct his aeroplane Flugan, although it never really flew.

The pioneers of flight and aerodynamics came to influence the road vehicle industry, which in its early days focused on increasing performance for race cars by reducing drag [11]. Later on the focus was shifted to making the designs more appealing to buyers, and these designs evolved and went on to take more inspiration from the efficient shapes of aeroplanes rather than from the blunt shapes of horse carriages. Many cars followed this design trend and a few trucks as well, for example see fig. 1.2, although often the designs were purely aesthetic as opposed to making the vehicles more aerodynamically efficient.

![1936 Labatt Beer delivery truck. Image by Coachbuilt](http://www.coachbuilt.com/)

Post-war, the advances in the aerospace industry gave useful insights into understanding air flows, and the use of wing profiles in racing became commonplace [12]. Surprisingly, ordinary vehicles took a step back in aerodynamic performance, but vehicles still grew faster while larger and more powerful engines kept the drag forces in check. Nonetheless, beginning with the 1970s energy crisis and new stricter emissions laws [13], the focus was brought back on aerodynamics and drag reduction.

Starting in the late 1970s and early 1980s there was also a growing interest in examining and improving the aerodynamics of heavier trucks, a field which earlier had been somewhat overlooked. Various testing of trucks, both in the field and in wind tunnels was conducted [2][10][14], also see fig. 1.3. The bodies of the tractors and trailers showed many areas where the flow could be improved, for example by deflectors, fairings and boat-tails attached to the back of the trailer.

To their help the aerodynamicists had experience, wind tunnels and also Computational Fluid Dynamics (CFD). CFD as a branch of fluid dynamics had grown quicker post-war since its slow start in the 1920s [28]. And as CFD relies heavily on computers the advances in computer design helped CFD become a viable alter-
native to the wind tunnels. Starting in the 1960s the CFD community developed more methods, and these could be used as super computers emerged. Commercial jet transport was born, sparking interest in transonic flow prediction. In the 1970s more CFD methods were developed, and in the 1980s computers became faster and more commonplace and CFD started seeing more use in the aeroplane industry. Since then CFD has gradually replaced many wind tunnel experiments, Boeing has for example reduced their wind tunnel testing by 50% since 1980 [40].

In 1991 the European Union amended an old 1970s directive on air pollution [17] which led to the Euro 1 emissions standard for passenger cars. The standard evolved and eventually led to Euro 2 (1998) through Euro 6 (2014). Since Euro 3 (2000) the standard has included trucks, and have led to a significant reduction of CO, NO\textsubscript{x}, particles and other emissions in the transportation industry. Actually the legislation have been so effective that pollutant emissions from vehicle engines have been reduced by almost 100% since 1991 [17]. The Euro 7 standard is not yet set, or even have a name, but is expected to include more focus on CO\textsubscript{2} emissions while considering the vehicle as a whole.

Legislation regarding aerodynamic devices has not been updated as often as the one on engine emissions and has therefore not seen as much progress. Although, the last few years have seen some development [18], and the latest amendment [1] looks set to bring a lot of change to the external shapes of trucks. With 70% of all EU inland transports going by road [19], the change in legislation will definitely help reduce green house gases in the future, and in the long run help create a cleaner environment.
1.2 Motivation

Society can make large savings, both economical and environmental, by improving the aerodynamic performance of a truck. A reduction by 12.5% of $C_D$ for a typical sized truck running 80 km/hr leads to an approximate reduction in fuel consumption by 5%, and cost savings for a typical long-haul truck driver [22] in the range of 5000 EUR/year [23]. These reductions in fuel would be the same for CO$_2$ and other greenhouse gases, helping considerably with the goals of both EU and manufacturers associations in reducing the environmental impact of heavy duty trucks.

There have been a lot of development of the design of tractors in the last 40 years, but from manufacturers there has not been very much focus on the trailers that are pulled. The drag created at the base of a trailer accounts for 29% of the total drag of a tractor-trailer-combo [25] so an improvement can make a large impact on fuel economy.

So called boat-tails, see fig. 1.4, have been researched for a long time, and have always shown to contribute to a substantial decrease in the drag of a tractor and trailer combo, but there have been various problems using the boat-tails. One problem is that truck manufacturers sell tractors and not the trailers that are pulled, so the truck manufacturers mainly focus on their tractors. The truck drivers that buy the trucks usually pull many different trailers, and do not usually own the trailers, so there is an economical aspect there as to who should carry the cost of an upgraded trailer. Then there has been the legislation which previously gave maximum dimensions for a tractor trailer combo, and the truck drivers wanted to use the space as efficiently as possible. On top of all this the boat-tail design in itself has to be easy to use and durable, and either be standardized to fit loading docks or foldable in some way. But, as previously mentioned, the new EU legislation will give more room for improvements on the aerodynamics of both the tractors and the trailers.

![Figure 1.4: A boat-tail added to the base of a truck trailer. Image from [29].](image)

This thesis examines the effect of boat-tails on a generic truck shape. Different sizes and shapes are tested to learn how it influences the flow characteristics. Further on, a suction slot is used to examine its influence on the drag of a sub-optimal boat-tail geometry. The sub-optimal shape was used since many times a design might have certain constraints, such as aesthetics or legislation, and suction could be used to improve the performance within these constraints.
Chapter 2

Bluff body aerodynamics

A body immersed in a fluid is subjected to drag forces. The body is characterized differently depending on which force dominates the total drag force. If the force on the body is mainly frictional, due to surface roughness, the body is considered streamlined. A typical example of a streamlined body is a fish, or an airfoil with a low angle of attack. Bluff bodies on the other hand have drag forces that arise from pressure differences, and a typical shape would be a simple brick or an airfoil with a high angle of attack, see fig. 2.1 for a simple comparison.

![A streamlined vs. a bluff body](image)

**Figure 2.1:** A streamlined vs. a bluff body

A truck and trailer combination, or the GTS as in the case in this thesis, is a bluff body. The forces acting on the GTS are mainly due to pressure differences between the front and the back of the shape, and to change the drag force one needs to reduce these pressure differences by changing the shape of the GTS. The concepts involved are further explained in the next section. For a more thorough explanation the reader could for example look into [31].

2.1 Flow regimes

Depending on what forces are dominant in a flow, convective or viscous, the flow can be divided into different regimes that show different flow behaviour. In a laminar flow the fluid moves in parallel layers with no mixing between them, and the important effects are viscous. Laminar flow is very orderly, does not change with time, and usually occurs at lower flow velocities. The structures involved in laminar flow can in most cases be considered two-dimensional.
As velocity increases the flow can become turbulent, and the viscous effects are lessened and convective effects become prominent. In this thesis, and as in most natural flow cases, the flow is turbulent. This means there are elements of randomness in the flow, and that the flow is seemingly chaotic and unpredictable. The flow shows a nonlinear behavior, showing acceleration because of change in position and it has a high diffusivity rate concerning momentum and heat. With fluctuations in vorticity, structures in the flow move, stretch and spin with very large differences in scale. These structures are called eddies, and the larger ones hand down energy to smaller eddies until finally the energy is dissipated by viscosity. A turbulent flow is also something that very much exists in all three dimensions of space, and is highly dependent on time.

laminar

transitional

Figure 2.2: The three different flow regimes.

turbulent

When the flow moves from a laminar to a turbulent regime, or the opposite, the flow can become transitional. Transitional flow exhibits characteristics from both the laminar and turbulent regimes, and shows large sensitivity to small changes in flow conditions.

2.2 Flow characteristics

When a fluid flow moves over a body it can at some points be brought to rest. Such points are called stagnation points, and the local velocity there will be zero and the local pressure will be at a maximum. The flow in the vicinity of the stagnation point is called a stagnation flow. A simple illustration is shown in fig. 2.3.

Figure 2.3: Flow lines toward a body, circle indicating the stagnation point.
If we study the continued flow over the body, it will develop a thin boundary layer where fluid viscosity dictates the flow. Viscosity imposes the no-slip condition, which means that the flow closest to the wall can not slip or move along the surface. The boundary layer can, just like the flow as a whole, also be characterized as laminar, transitional or turbulent. A laminar boundary layer is thin and smooth, and sensitive to free stream fluctuations and surface roughness. As the laminar boundary layer evolves over a surface, it slowly thickens and may transition and evolve into a turbulent boundary layer. The turbulent boundary layer is more stable and thicker, with faster growing thickness, and is irregular in its nature. The shear stress is significantly higher in the turbulent compared to the laminar boundary layer, and this gives rise to more skin friction drag. A turbulent boundary layer can be divided into two regions with very different characteristics. The inner region and its innermost part, the viscous sublayer, is heavily influenced by viscosity. Moving out from the wall the buffer region starts, working as a transition, and this is where the maximum turbulence production is. Further out the velocity in the boundary layer follows a logarithmic profile, where turbulent and viscous stresses are in balance. Then closest to the free flow is the outer region where viscous stresses are considered negligible and turbulent stresses are prominent. See fig. 2.4 for an illustration.

![Figure 2.4: Sketch of a typical velocity profile for a turbulent boundary layer.](image)

If the body surface is curved, the boundary layer will grow quicker. The flow next to the wall will decelerate and the pressure will rise, creating an adverse pressure gradient. At one point the flow next to the wall will reverse and form a region of backward flow, this will meet the flow from further out and the fluid from close to the surface will be transported outward. The point where the flow starts moving outwards from the boundary layer is called the separation point, and at that point the flow separates from the wall. See fig. 2.5. A turbulent boundary layer can reduce drag by delaying this separation, and thereby reducing pressure drag. Drag is further explained in section 2.3.
As the pressure increases and the flow decelerates, the boundary layer grows and the flow reaches a point of separation, here marked with a circle. As the flow separates from the body and reverses, a turbulent wake is formed. In the wake region the pressure is lower than the pressure before the stagnation point. This pressure difference is the main source of drag for a blunt body. Just outside the wake area, where the wake flow interfaces with the free flow outside of the body, a shear layer is formed. The shear layer is the interface between the inner turbulent flow and the outer irrotational flow. This thin layer is as thin as the smallest scales in the flow, the Kolmogorov microscale, with these scales only depending on viscosity $\nu$ and the rate of dissipation $\epsilon$ [38]. In the shear layer flow instabilities are initiated and turbulence is produced, to further downstream turn into a fully turbulent flow.

In the wake of the body, after separation, a recirculation bubble can form. In the recirculation bubble the flow circulates back towards the body, creating a higher pressure at the base of the wake. Further downstream of the body the flow becomes self-similar, and show the same characteristic velocity profile continuously behind the body. See illustrations of these phenomena in fig. 2.6.

During certain flow conditions a von Karman vortex street can be formed in the wake of a bluff body. The vortex street is composed of eddies that periodically shed off of the body, and gives rise to oscillating forces on the body itself. These vortex streets can form behind a variety of different blunt objects, and on all different scales. For an example of a large-scale vortex street see fig. 2.7.
2.3 Forces, coefficients and representative scales

Aerodynamic force on a body can be characterized as *drag force*, oriented opposite the flow direction, or *lift force* that acts perpendicular to the flow. These forces can arise from different aspects of the flow, either from the influence of viscosity or the influence of the pressure distribution around the body. Viscosity gives rise to stresses and thereby a shear force that is parallel to the flow, whereas pressure differences create a force that is normal to the body. Ergo, the lift force arises from the pressure difference. The pressure differences also have the largest impact when considering the drag force on a blunt body with a separated boundary layer.

To be able to compare the values of drag of different geometries, without respect to scale, the *coefficient of drag* (2.1) is very useful. $F_D$ is the drag force, and $A$ is the reference area of the body. In this thesis $A$ is the cross-sectional area perpendicular to the flow around the GTS.

$$C_D = \frac{2F_D}{\rho U^2 A} \quad (2.1)$$

To study the pressure in a flow field, also without respect to scale, one can use
the coefficient of pressure (2.2)

\[ C_P = \frac{2(p - p_\infty)}{\rho U^2}, \quad (2.2) \]

where \( p \) is the local pressure and \( p_\infty \) is the ambient pressure. \( C_P \) describes how the local pressure in every point of a flow field relates to the pressure far away. Consequently, if \( C_P \) is zero, the local pressure is the same as the ambient pressure and if \( C_P \) is 1, the local pressure is the stagnation pressure and that point is a stagnation point.

In order to compare different flows and predict flow behaviour Reynolds number \( Re \) (2.3) can be used. \( Re \) is a non-dimensional quantity that is a function of density \( \rho \), mean velocity \( U \), characteristic length \( L \) and dynamic viscosity \( \mu \). For exact values used in the thesis see table 4.1.

\[ Re = \frac{\rho U L}{\mu} \quad (2.3) \]

\( Re \) is essentially the ratio of inertial forces to viscous forces, and can therefore be used to predict flow regime and behaviour. A laminar flow depends highly on viscous forces and hence it occurs for low \( Re \) which can be attributed to either low inertia or high viscosity. As the flow moves into the transitional regime the ratio of inertia to viscosity balances, and depending on small changes in any parameter the balance can change quickly. As either velocity or any other parameter in the numerator grows, or if viscosity decreases, the flow becomes turbulent and the inertial forces are most dominant. Because of the non-dimensionality of \( Re \), transitions between flow regimes occur at the same value of \( Re \).

\( Re \) can be used as a scaling parameter, and is very useful when the size of a body is too large to fit in a wind tunnel. Since the flow behaviour depends only on \( Re \), the different parameters can be varied but still create the same flow characteristic. For the case of the GTS, a 1/8th scale model is used, but in order to have the same \( Re \) as for a full scale model the speed of the flow is increased.

Another dimensionsless number that is useful is the Strouhal number (2.4),

\[ St = \frac{fL}{U}, \quad (2.4) \]

where \( f \) is the frequency of vortex shedding, previously explained in 2.2. The frequency \( f \) can be an important number when building structures or designing vehicles. If \( f \) is close to the natural frequency of a body, the shedding can induce oscillations which can lead instabilities, even breakdown of a structure, or to sound emissions with a specific tone.
Chapter 3

Numerical approach

3.1 Flow governing equations

CFD aims to numerically solve problems concerning fluid flow. Nowadays the calculations are done by computer, although it did not start out that way some 100 odd years ago [28]. Since the 1950s and the advent of the modern digital computer CFD has evolved almost just as quickly as the processing speed of computers, making it a more powerful tool every day.

The basis for most CFD problems, and certainly the problems in this thesis, are the Navier-Stokes equations, here for an incompressible Newtonian fluid (with viscosity independent of stress), omitting the energy equation

\[
\rho \left( \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} \right) = - \frac{\partial p}{\partial x_i} + \mu \frac{\partial^2 u_i}{\partial x_j \partial x_j},
\]

(3.1)

\[
\frac{\partial u_i}{\partial x_i} = 0
\]

(3.2)

where \( u \) is velocity, \( \rho \) is density, \( p \) is pressure, \( \nu \) is the kinematic viscosity. Body forces, such as \( g \) are omitted. The equations follow incompressibility if the speed of the flow is low enough, that is if the Mach number (3.3) is below 0.3.

\[
Ma = \frac{U}{c},
\]

(3.3)

with \( c \) being the local speed of sound.

The Navier-Stokes equations are derived from the basic fundamental principles of conservation of mass, momentum and energy. By converting these partial differential equations to a system of algebraic equations the solutions can be obtained by means of a computer. STAR-CCM+, the software utilised in this thesis, uses the Finite Volume Method (FVM) to convert, or discretize, the Navier-Stokes equations at discrete points on a meshed geometry. Each point in the mesh has a surrounding volume and then the flux on each corresponding surface is calculated, conserving the fluxes all throughout.
The Navier-Stokes equations can be solved directly by numerical simulation (DNS), and the results are very accurate. But, solving directly is extremely time consuming, and for the majority of flow cases it is just not feasible. The number of operations involved in a DNS grow as $Re^3$, which means that the computational costs are very high for more turbulent flows, for example like the one conducted in this thesis.

### 3.2 Turbulence modelling

A turbulent flow, and all the different scales involved, is time consuming to calculate directly with numerical simulation. The equations that govern the flow need to model turbulence somehow to cut the computing costs. So if one does not concern oneself with all the different scales of a turbulent flow, and is just interested in the gross characteristics of it, the Navier-Stokes equations (3.1) can be averaged so they are easier to solve. The averaging is done using Reynolds decomposition, which divides an instantaneous quantity into one part that is time-averaged and one part that is fluctuating, for example velocity as in the following:

$$ u = \bar{u} + u', \quad (3.4) $$

where the mean of the fluctuating component $u' = 0$. If the flow is considered incompressible and Newtonian, the Navier-Stokes equations (3.1) together with Reynolds decomposition (3.4) results in the Reynolds-Averaged Navier-Stokes (RANS) equations

$$ \rho \left( \frac{\partial \bar{u}_i}{\partial t} + \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} \right) = -\frac{\partial \bar{p}}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho u'_i u'_j \right) \quad (3.5) $$

and

$$ \frac{\partial \bar{u}_i}{\partial x_i} = 0 \quad (3.6) $$

which look very similar to the ordinary Navier-Stokes equations (3.1), with the exception of an additional term with the Reynolds stress tensor

$$ -\rho u'_i u'_j \quad (3.7) $$

This gives rise to six extra variables, and the resulting equations need to be closed somehow, which is done differently depending on turbulence model.

There are many different ways to provide closure to the RANS-equations (3.5); zero-, one-, two-equation, second-order closure and algebraic stress models. This thesis uses the SST $k$-$\omega$-model. The choice of this model was based on results from [32] and [6], who worked on the same geometry, and the STAR-CCM+ User Guides[34] suggested best practices for vehicle simulations.

The SST $k$-$\omega$-model (by Menter[35]) is a two-equation model that is a hybrid between the classic $k$-$\omega$ and the $k$-$\epsilon$-model. All these models use the Boussinesq hypothesis which assumes that the turbulent shear stress is linearly related to the mean rate of strain and the proportionality factor is called the *eddy-viscosity*. 


3.2.1 Wall treatment

At wall boundaries, where the flow has high velocity gradients, the mesh needs to be very fine to resolve the viscous effects. In order to keep the computing costs down and still have a fine mesh with reasonable cell aspect ratio, a wall treatment can be used. This usually involves some sort of wall function, which can take into account viscous effects near the wall and blend them with the region far from the wall where the flow is near inviscid. The theory behind wall functions is the law-of-the-wall

\[
\frac{U}{u_*} = f(y_+),
\]

(3.8)

where \( U \) is the flows velocity and

\[
u = \sqrt{\frac{\tau_0}{\rho}},
\]

(3.9)

is called the friction velocity which in turn depends on \( \tau_0 \), the shear stress at the wall, and the density \( \rho \). Together these two velocities are a function of \( y_+ \), the wall distance, non-dimensionalized by viscosity \( \nu \) and \( u_* \). The \( y_+ \) value can be used to further explain the regions of the turbulent boundary layer, previously discussed in section 2.2. Low \( y_+ \)-values in the interval of \( 0 < y_+ < 5 \) is in the viscous sublayer, \( 5 < y_+ < 30 \) is the buffer layer, and then the log-layer starts somewhere in \( 30 < y_+ < 200 \) which turns into the outer region. Also take a look at fig. 3.1, an update on a figure previously used.

![Figure 3.1: A typical velocity profile for a turbulent boundary layer, with the different regions pointed out and correlated with \( y_+ \).](image)

This study uses STAR-CCM+’s ‘all \( y_+ \)’ wall treatment model, which is a hybrid that adapts to the coarseness of the mesh and uses wall laws that blend between the viscous sublayer, buffer- and log-layer. In order to resolve the viscous sublayer near the wall, the \( y_+ \) value need to be lower than 1.
Chapter 4

Case description & setup

The shape used in this study (fig. ??) is called the Ground Transportation System (GTS) and was developed by Sandia National Laboratories in Albuquerque, New Mexico, USA. It is designed to work as a baseline to evaluate vehicle aerodynamics, both in wind tunnels and by CFD. The model is built as a 1/8th scale truck and trailer, because a full size GTS would have been too big to fit in any but the largest wind tunnels in the world.

The GTS has a 2.4761 m length $l$, 0.3238 m width $w$ (also used as characteristic length for $Re$) and 0.4507 m height $h$, its shape is similar to a truck, but simplified, and does for example not include wheels or tractor-trailer gap. The projected frontal area $A$ is 0.152 m$^2$, including the area from the struts protruding underneath the body. The GTS shape has been used throughout a lot of research [3][7][6], and is simple enough for ease of meshing, but still has the characteristics of a modern truck. The shape and some dimensions of the GTS are shown in fig. 4.1. For complete dimensions see [3].

The three boat-tails were modelled within STAR-CCM+. Case 2.1, referred to as ‘straight’, is a box with a $10^9$ symmetrical slant, extending 10 cm behind the GTS. The straight geometry was used after examining results from [41]. Case 2.2, referred to as ‘smooth’, is a smooth symmetrical profile extending 12.5 cm behind the GTS. It was modelled as a truncated wing-profile, or Kammback, and designed freehand on basis of the flow field behind the straight boat-tail. Case 2.3, referred to as ‘round’, is a slightly asymmetrical profile extending 13.75 cm behind the GTS. The round tail is a first draft of a design from the Trailair project, see section A. The boat-tails are shown in fig. 4.2.

The round design is also used for the suction slot cases 3.1-3.3. The suction slots have a width of 0.015w (4 cm full scale) and are placed symmetrically around the boat-tail at three different locations with no spacing in between. The first slot begins 0.31w behind the base of the boat-tail, the next one at 0.325w and the last one at 0.34w. The slots sit flush with the boat-tail so the flow encounters no step in the geometry. The slots were placed in close proximity of the point of separation determined by the case run without suction slot. Also see fig. 4.3.
Figure 4.1: Dimensions of the GTS model used.

Figure 4.2: The three different boat-tail designs; straight, smooth and round.

Figure 4.3: The location of the three suction slots shown in pink.

The GTS is surrounded by a fluid domain that is loosely modelled after the NASA Ames 7- by 10-Ft wind tunnel used in [3] to be able to compare the results in this study with the results of a wind tunnel experiment. The fluid domain is
a box shaped symmetric shape that is approximately 48 \( w \) (GTS body widths) in length, 9 \( w \) in width and 7 \( w \) in height. The fluid domain with GTS is shown in fig. 4.4.

![Figure 4.4](image)

**Figure 4.4:** GTS and the surrounding fluid domain with the boundaries specified.

### 4.1 Mesh

![Figure 4.5](image)

**Figure 4.5:** Perspective view of the mesh refinements. Flow direction in figure is left to right, \( i \)-direction.

In line with the ‘Best Practices’ guideline on aerodynamics calculations in the STAR-CCM+ User Guide [34] the fluid domain is mainly meshed with hexahedral cells together with prismatic cells at the GTS and the ground. The mesh is coarser far away from the GTS, but gradually turning finer closer to it and in the wake of the GTS. The mesh is refined using volumetric controls consisting of 16 boxes that add cells to areas with large vorticity and gradients. The mesh is also further refined in the wake region with refinements growing from the boundaries of the struts and the rear panel of the GTS. For a full picture on all mesh refinements see fig. 4.5 and fig. 4.6. Figs. 4.5-4.11 shows further details on the meshing structure and the increase in mesh density closer to the GTS.
4.2 Initial & boundary conditions

The flow domain has a velocity inlet at the wall in front of the GTS and a pressure outlet behind it, see fig. 4.4. The pressure at the outlet is 0 Pa gauge, i.e. the ambient air pressure at 101 325 Pa is set as zero. The GTS model and the floor beneath it are modelled as walls with a no-slip boundary condition, i.e. zero velocity at the wall. The other boundaries surrounding the GTS are modelled with a
symmetry boundary condition, meaning the normal velocity is zero. For cases 3.1-
3.3 the suction slots were modelled as a pressure outlet with a negative pressure. 
The pressure was decided on by evaluating work by others [41][42] and comparing 
to a typical pressure of a normal vacuum cleaner [43]. All initial and boundary 
conditions data can be viewed in tables 4.1 and 4.2, respectively. Initial conditions 
follow from [7].

4.3 Solver

STAR-CCM+ uses a specific workflow, reference fig. 4.12 for the following section. 
Geometry, boundary conditions and meshing are described in previous sections. In 
order to get the correct results in the desired time the correct physics models need
Table 4.1: Initial conditions and values derived therefrom.

<table>
<thead>
<tr>
<th>Name</th>
<th>Denotion</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Air density</td>
<td>( \rho )</td>
<td>1.225 kg/m(^3)</td>
</tr>
<tr>
<td>Freestream velocity</td>
<td>( U_\infty )</td>
<td>91.64 m/s</td>
</tr>
<tr>
<td>Characteristic length</td>
<td>( w )</td>
<td>0.3238 m</td>
</tr>
<tr>
<td>Dynamic viscosity</td>
<td>( \mu )</td>
<td>1.85508 \times 10^{-5} Pa·s</td>
</tr>
<tr>
<td>Reynolds number</td>
<td>( Re )</td>
<td>1.96 \times 10^6</td>
</tr>
</tbody>
</table>

Table 4.2: Boundary parameters.

<table>
<thead>
<tr>
<th>Boundary</th>
<th>Boundary condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Velocity inlet</td>
<td>91.64 m/s</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure outlet</td>
<td>0 Pa (gauge)</td>
</tr>
<tr>
<td>GTS</td>
<td>Wall</td>
<td>-</td>
</tr>
<tr>
<td>Floor</td>
<td>Wall</td>
<td>-</td>
</tr>
<tr>
<td>Ceiling and tunnel walls</td>
<td>Symmetry</td>
<td>-</td>
</tr>
<tr>
<td>Suction slot</td>
<td>Pressure outlet</td>
<td>-7.5 kPa</td>
</tr>
</tbody>
</table>

to be chosen. First off, as the flow is highly turbulent with a high \( Re \), and structures in a turbulent wake are to be examined, a 3D model of space is chosen. A steady-state time model is chosen as it has the least computational cost compared to the results. The material is gaseous air.

The user manual [34] suggests using a segregated flow model and a constant density equation of state for incompressible external aerodynamics. The segregated solver uses an Algebraic Multigrid (AMG) linear approach for velocity, pressure and the transport variables \( k \) and \( \omega \). The use of AMG dampens both high and low frequency errors in the solution. The segregated solver calculates the flow parameters uncoupled, and mass and momentum iterates independently with a predictor-corrector linkage combined with the SIMPLE algorithm. The solver solves the discretized momentum equation and the discrete equation for pressure correction, see the flow chart in fig. 4.13 for a schematic view. To promote convergence an under-relaxation factor (URF) is used to control how big a portion of the old solution should be used for the next iteration. The URF ranges from 0 to 1, where 1 means that no part of the old solution is used. So a larger URF means more drastic changes for every iteration, but it may also lead to numerical instability.

The viscous regime is chosen as turbulent. The turbulence model chosen is the SST (Menter) \( k-\omega \), as previously explained (section 3.2.1), as is the ‘all \( y_+ \)’ wall treatment used. Initial conditions are presented in section 4.1. The simulations were stopped when convergence was reached, which is explained in the next section.

All analyzations of the results were carried out either with STAR-CCM+ or MATLAB.
Figure 4.12: The steps involved in a CFD simulation with STAR-CCM+.
Figure 4.13: A schematic view of the SIMPLE algorithm.
Chapter 5

Results

The study aims to improve the aerodynamic performance of a truck by reducing the drag by means of a boat-tail. The investigation was divided into a few cases to study the effects different parameters of the boat-tail have on the drag coefficient. The first case is simply the GTS only, run to be used as a baseline to compare the other cases drag improvements to. The 2nd case is the GTS with three different boat-tail geometries, to investigate the influence on the flow of the shape and size of the boat-tail. The 3rd case uses the rounded geometry from case 2, together with a suction slot set in three different places to see how suction can influence the boundary layer and reduce drag of a non-optimal shape. See table 5.1 for a schematic view of the cases investigated.

<table>
<thead>
<tr>
<th>Case</th>
<th>Name</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Baseline</td>
</tr>
<tr>
<td>2.1</td>
<td>Straight</td>
</tr>
<tr>
<td>2.2</td>
<td>Smooth</td>
</tr>
<tr>
<td>2.3</td>
<td>Rounded</td>
</tr>
<tr>
<td>3.1</td>
<td>Rounded: Suction Top</td>
</tr>
<tr>
<td>3.2</td>
<td>Rounded: Suction Mid</td>
</tr>
<tr>
<td>3.3</td>
<td>Rounded: Suction Low</td>
</tr>
</tbody>
</table>

5.1 Verification

5.1.1 Mesh quality

To get accurate results from the simulation, the mesh needs to be carefully constructed. In order to assess the quality of the mesh some parameters need to be examined. Firstly, the skewness angle of the cells should avoid going over 85°. A perfect cell would have 0° skewness and would freely permit diffusion of quantites
between cells. If the skewness goes above $90^\circ$, which can occur in concave cells, the solver can get convergence issues. The final mesh used has a maximum skewness of $88.475^\circ$, and cells with skewness over $85^\circ$ are shown in fig. 5.1. The highly skewed cells are all on the boundaries of the mesh, but because of the use of wall functions the boundary skewness angle becomes less important.

![Cells with skewness angle greater than $85^\circ$. All of them on the boundaries of the mesh.](image)

Another measure of quality is the face validity, which assesses the correctness of the cell face normals relative to their centroid. A good quality cell has all face normals pointing outwards, whereas a bad quality cell can have one or more normals pointing inwards towards the centroid. Face validity ranges from 0 to 1, where 1 means all normals point outward from the centroid, and values below 0.5 indicates the cell has a negative volume. All cells in the mesh used have a validity of 1.

The volume change metric describes how the volume of a cell compares to its largest neighbour. A large difference in volume between cells can lead to instability in the solver, and a volume change value below $1 \cdot 10^{-5}$ should be avoided. No cells in the mesh used have a volume change value below $1 \cdot 10^{-2}$. All facts about mesh quality can be further explored in STAR-CCM+'s User Manual [34].

### 5.1.2 Grid convergence study: Baseline configuration

CFD simulations demand a lot of computer power and time, and the more cells a mesh has the more power and time is needed. Smaller cells can resolve more flow structures than a coarser mesh, but there is a point where a finer mesh gives very little in return for the result. A mesh independence study was therefore carried out, as to not use unnecessary many cells in the mesh and to get confidence in the results.

Six meshes were used to study mesh independence, all refined by using relative cell sizes in STAR-CCM+ and then varying the base size on which the relative size
depends. By using relative cell sizes the task was easily carried out by just changing one parameter. The number of cells ranged from the coarsest with 1.5 million cells to the finest with 18.9 million cells. Table 5.2 shows the number of cells and $C_D$ value for each mesh. In order to even out fluctuations the $C_D$ was averaged over at least 500 iterations after the solution was converged. Mesh 4 was chosen for the study because of its balance between accuracy and computer cost.

Table 5.2: Mesh independence study.

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Avg. cell length [m]</th>
<th>No. of cells</th>
<th>$C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.04</td>
<td>1.5 million</td>
<td>0.349</td>
</tr>
<tr>
<td>2</td>
<td>0.03</td>
<td>3.2 million</td>
<td>0.334</td>
</tr>
<tr>
<td>3</td>
<td>0.025</td>
<td>6.4 million</td>
<td>0.328</td>
</tr>
<tr>
<td>4</td>
<td>0.022</td>
<td>8.8 million</td>
<td>0.325</td>
</tr>
<tr>
<td>5</td>
<td>0.02</td>
<td>12.5 million</td>
<td>0.323</td>
</tr>
<tr>
<td>6</td>
<td>0.017</td>
<td>18.9 million</td>
<td>0.324</td>
</tr>
</tbody>
</table>

Experiments measure different $C_D$, 0.249 [3](without struts) and 0.35 [4]. Studies with similar CFD modelling show values 0.318 [5] and 0.298 [6], more in the range of those presented here. $C_D$ values from references can be found in tables 5.4 and 5.5, note the difference in measurements as some references do not include the struts. The references also use slightly different geometries, see Appendix D.

Figure 5.2: Number of cells vs. $C_D$, black points from results and the blue curve is a two-term power series fit.
5.2 Case 1: Baseline

The first case is run as a baseline, or reference, case to compare the other cases to. Since the work is carried out numerically, with errors from a wide range of parameters such as numerical method, model, round-off errors etc. one has to keep in mind that the results may not be the same as for an experiment. That said, the trends for the results should be reflected in reality. The baseline case has also been investigated by others, see ref. [3]-[6], so the results in this thesis are compared with those.

All simulations are run with RANS, and as discussed earlier it is an averaged solution which loses information, for example regarding turbulent eddies. As a reference a comparison view with a Detached Eddy Simulation (DES) can be seen in fig. 5.3, where the DES solution shows a snapshot in time to compare with the time-averaged RANS solution.

![Comparison of i-velocities in the wake of the GTS. Steady-state RANS (top) and a time-dependent DES solution (bottom).](image)

**Figure 5.3:** Comparison of $i$-velocities in the wake of the GTS. Steady-state RANS (top) and a time-dependent DES solution (bottom).

To make sure that the solution is converged, the residuals were examined. The residuals show the difference of some quantity from one solution iteration to the next. In STAR-CCM+ the residual is actually the root-mean-square (RMS) value
of the absolute error normalized with the largest value from the last five iterations, such as

\[ R_{\text{pres}} = \frac{R_{\text{rms}}}{R_{\text{norm}}} \]  

(5.1)

\[ R_{\text{norm}} = \max\{|R_1|, |R_2|, ..., |R_m|\} \]  

(5.2)

where \( m \) is set to 5 as default. Residuals alone are not a sufficient measure of convergence, the evolution of \( C_D \) is also examined is that is of interest to the study. In fig. 5.4 the residuals and the \( C_D \) plots for the Baseline case are presented.

![Figure 5.4: The residuals and \( C_D \) for the Baseline geometry.](image)

The solution is started with a coarser mesh as pre-conditioning to provide faster turn-around time, as the coarser grid solution leads to a better initial guess for the final mesh. Hence, the residuals jump back up once the solution is started with the finer mesh, and does not drop as quickly again. The same with the \( C_D \) plot, the value jumps when the mesh is further refined. Convergence is determined to be reached when both plots flatten out and only shows periodical oscillations. These oscillations are an indication of the inherent unstable nature of the problem at hand. Therefore, for any results presented by a number, that number was reached by averaging the quantity over at least 500 iteration to limit the effect of the fluctuations.

Fig. 5.5-5.13 shows the velocity and turbulent kinetic energy (TKE) in different planes for the Baseline geometry. Descriptions follow in the captions. Measurements taken in the wake show distance from the base of the Baseline geometry, distances are normalized by \( w \), as is done in [3].
Table 5.6 shows the wake closure length \( l_c \) of all tried cases, where \( l_c \) is defined as the distance from the base of the Baseline-GTS geometry to the point where the mean streamwise velocity is zero. The table also shows the lengths behind the tail geometries.

As can be seen in the plots, many values presented are normalized by either GTS width \( w \), freestream velocity \( U_\infty \) or maximum TKE (637.3 J/kg) for the Baseline case.

**Figure 5.5:** Velocity in \( i \)-direction, view from the side, in the XY-plane at \( z=w/2 \). The highest velocity is at the front, just below the nose of the GTS, where the flow is accelerated because of the curvature and decrease in area. The lowest velocity is in the wake, where the flow moves back towards the GTS. Right at the front there is a stagnation point, as explained in section 2.2.

**Figure 5.6:** Wake velocity profiles of \( i \)-direction velocity in the XY-plane at \( z=w/2 \). Legend shows distance from the back of the GTS. The velocity profiles show that there is a rather large region of negative \( i \)-velocity if one examines the profiles for \( 0.25w \), \( 0.5w \) and \( 1w \). Further away from the GTS the profiles start to show self-similarity, as discussed earlier and shown in fig. 2.6.
Figure 5.7: Velocity in $i$-direction for the GTS-Baseline geometry, view from the top, in the XZ-plane at $y=h/2$. Here one can see that the flow not only separates in the wake, but also right at the sides of the nose of the GTS. The boundary layer is thicker in this plane, compared to XY, and grows as it travels further downstream.

Figure 5.8: Wake velocity profiles of $i$-direction velocity in the XZ-plane at $y=h/2$. Legend shows distance from the back of the GTS. The velocity profiles for the wake show behaviour similar to the profiles for the XY-plane.

5.3 Case 2: Boat-tails

The second case involved running simulations on the three different boat-tail geometries. Comparisons to the Baseline can be seen in fig. 5.15-5.18. See table 5.2 for the results and improvement of $C_D$. Comparing with experiments, see table 5.4, the improvements look to be in line with some others, but vary quite a lot because of the large difference in boat-tail geometries. Measurements taken in the wake show distance from the base of the Baseline geometry, which means that for the boat-tails the measurements are closer to the base of the actual geometry.

Individual plots showing all the boat-tail geometries in the same manner as the Baseline case 1 can be seen in Appendix B. In fig. B.1 through B.29 it can be seen that the simple straight tail and the smooth tail show similar flow structures, but with less velocity drop for the smooth tail. Comparing with the straight tail,
the smooth tail shows an increase in velocity at $h/2$, as seen in fig. B.14, and also less TKE in the XZ-plane, seen in fig. B.19. The round tail shown in fig. B.24-B.29 exhibits some instabilities, especially evident in fig. B.29 where the TKE is asymmetric, and there looks to be shearing layers in several places in the wake.

As the case with the round boat-tail showed unstable solutions, all results (except the vector plot in fig. 5.17) have been averaged over 500 iterations in order to achieve more stable values. The same for Case 3 where the same geometry was used.

**Table 5.3:** $C_D$ improvements with different boat-tail geometries.

<table>
<thead>
<tr>
<th>Name</th>
<th>$C_D$</th>
<th>Improvement</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>0.325</td>
<td>-</td>
</tr>
<tr>
<td>10°/0.3w straight</td>
<td>0.266</td>
<td>18.1%</td>
</tr>
<tr>
<td>0.39w smooth</td>
<td>0.256</td>
<td>21.2%</td>
</tr>
<tr>
<td>0.42w rounded</td>
<td>0.295</td>
<td>9.2%</td>
</tr>
</tbody>
</table>

**Table 5.4:** Experimental $C_D$ values from [3][4].

<table>
<thead>
<tr>
<th>Reference</th>
<th>Geometry</th>
<th>$C_D$</th>
<th>Improvement</th>
<th>Re</th>
</tr>
</thead>
<tbody>
<tr>
<td>[3]</td>
<td>Baseline (no struts)</td>
<td>0.249</td>
<td>-</td>
<td>2.10^6</td>
</tr>
<tr>
<td>[3]</td>
<td>0.29w boat-tail plates (no struts)</td>
<td>0.205</td>
<td>17.7%</td>
<td>2.10^6</td>
</tr>
<tr>
<td>[4]</td>
<td>Baseline</td>
<td>0.35</td>
<td>-</td>
<td>1.6.10^6</td>
</tr>
<tr>
<td>[4]</td>
<td>0.59w boat-tail</td>
<td>0.35</td>
<td>0%</td>
<td>1.6.10^6</td>
</tr>
<tr>
<td>[4]</td>
<td>0.94w boat-tail</td>
<td>0.33</td>
<td>5.7%</td>
<td>1.6.10^6</td>
</tr>
</tbody>
</table>

**Table 5.5:** CFD modelling $C_D$ values from [4]-[7].

<table>
<thead>
<tr>
<th>Reference</th>
<th>Geometry</th>
<th>$C_D$</th>
<th>Improvement</th>
<th>Re</th>
<th>Mesh size</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>[4]</td>
<td>Baseline</td>
<td>0.47</td>
<td>-</td>
<td>1.6.10^6</td>
<td>0.6.10^6</td>
<td>RNG k-ε</td>
</tr>
<tr>
<td>[4]</td>
<td>0.59w boat-tail</td>
<td>0.42</td>
<td>10.6%</td>
<td>1.6.10^6</td>
<td>0.6.10^6</td>
<td>RNG k-ε</td>
</tr>
<tr>
<td>[4]</td>
<td>0.94w boat-tail</td>
<td>0.41</td>
<td>12.8%</td>
<td>1.6.10^6</td>
<td>0.6.10^6</td>
<td>RNG k-ε</td>
</tr>
<tr>
<td>[5]</td>
<td>Baseline</td>
<td>0.318</td>
<td>-</td>
<td>2.1.10^6</td>
<td>1.25.10^6</td>
<td>SA</td>
</tr>
<tr>
<td>[6]</td>
<td>Baseline</td>
<td>0.298</td>
<td>-</td>
<td>2.10^6</td>
<td>20.10^6</td>
<td>k-ω SST</td>
</tr>
<tr>
<td>[7]</td>
<td>Baseline (no struts)</td>
<td>0.25</td>
<td>-</td>
<td>2.10^6</td>
<td>13.8.10^6</td>
<td>RNG k-ε</td>
</tr>
<tr>
<td>[7]</td>
<td>Baseline (no struts)</td>
<td>0.253</td>
<td>-</td>
<td>2.10^6</td>
<td>13.8.10^6</td>
<td>DES</td>
</tr>
</tbody>
</table>

**5.4 Case 3: Rounded boat-tail with suction slots**

Fig. 5.22 through 5.25 show the round tail geometries with suction slots in different locations. Descriptions follow in the captions. Individual plots showing all the boat-tail geometries in the same manner as the Baseline case 1 can be viewed in Appendix C.
Table 5.6: Wake closure length $l_c/w$ of the different geometries, $(l_c$-tail length)/$w$ for actual wake length.

<table>
<thead>
<tr>
<th>Name</th>
<th>$l_c/w$</th>
<th>$(l_c$-tail length)/$w$</th>
<th>Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Baseline</td>
<td>1.95</td>
<td>1.95</td>
<td>-</td>
</tr>
<tr>
<td>$10^\circ/0.3w$ straight</td>
<td>1.79</td>
<td>1.48</td>
<td>7.9%</td>
</tr>
<tr>
<td>0.39$w$ smooth</td>
<td>1.54</td>
<td>1.16</td>
<td>20.6%</td>
</tr>
<tr>
<td>0.42$w$ rounded</td>
<td>1.02</td>
<td>0.60</td>
<td>47.6%</td>
</tr>
</tbody>
</table>

All values from Case 3 have been averaged over 500 iterations to counter the instability and fluctuations in the solution.

Table 5.7: $C_D$ improvements with suction slots on round boat-tail. $C_{DS}$ is the shear part of $C_D$.

<table>
<thead>
<tr>
<th>Name</th>
<th>$C_D$</th>
<th>Improvement</th>
<th>$C_{DS}$</th>
<th>$C_{DS}/C_D$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Round</td>
<td>0.295</td>
<td>-</td>
<td>0.063</td>
<td>21%</td>
</tr>
<tr>
<td>Suction Hi</td>
<td>0.283</td>
<td>4.0%</td>
<td>0.066</td>
<td>23%</td>
</tr>
<tr>
<td>Suction Mid</td>
<td>0.304</td>
<td>-2.8%</td>
<td>0.060</td>
<td>20%</td>
</tr>
<tr>
<td>Suction Low</td>
<td>0.291</td>
<td>1.4%</td>
<td>0.066</td>
<td>23%</td>
</tr>
</tbody>
</table>
Figure 5.9: Evolution of vorticity throughout the wake of the GTS-Baseline geometry. Vorticity moving from the boundary layer around the GTS and diminishing further down the wake. Bubble shapes on the side induced by vorticity downstream, at the slanted front edges of the body. Asymmetries in the area below the base, right behind the struts.
Figure 5.10: TKE for the GTS-Baseline geometry, view from the side, in the XY-plane at $z=w/2$.

Figure 5.11: TKE profiles for the GTS-Baseline geometry, in the XY-plane at $z=w/2$. One can see how the shear layers show up in the profile as there are small peaks in TKE at the bottom and top.

Figure 5.12: TKE for the GTS-Baseline geometry, view from the top, in the XZ-plane at $y=h/2$. There are shear layers right at the boundaries between the wake and the boundary layer of the GTS, and there is also some shearing at the interface between the GTS boundary layer and the freestream.

Figure 5.13: TKE profiles for the GTS-Baseline geometry, in the XZ-plane at $y=h/2$. 
Figure 5.14: Pressure for the GTS-Baseline geometry, in XY-plane at $z=w/2$ and XZ-plane at $y=h/2$. High pressure at the stagnation point in the front and low pressure at the base of the wake. The wake also shows a raised pressure downstream of the GTS, where velocities in j- and k-direction decelerate. Between the high and low pressure regions is a saddle point with ambient pressure.
Figure 5.15: Comparison between isosurfaces of zero $i$-velocity for the different geometries. Inside the bubble there are negative $i$-direction velocities. Coloured with $C_P$ according to legend. One can see that the surface shrinks in line with $C_D$, as negative velocities are part of the reason for drag.
(a) Baseline geometry, $l_c/w = 1.95$, $C_D = 0.325$.

(b) Straight tail geometry, $l_c/w = 1.79$, $C_D = 0.266$.

(c) Smooth tail geometry, $l_c/w = 1.54$, $C_D = 0.256$.

(d) Rounded geometry, $l_c/w = 1.02$, $C_D = 0.295$.

Figure 5.16: Comparison of $i$-direction velocity fields of the different boat-tails, note that only negative velocities are represented. (a), (b) and (c) show that shorter wake length is in line with the $C_D$ improvements. The rounded geometry in (d) shows a short but unstable velocity profile with separation not clearly defined, leading to uncertainty in the measurement of its $l_c$. 
Figure 5.17: Velocity vectors for the different boat-tail geometries at $z=w/2$ and $y=h/2$ respectively. Magnitude according to color bar. Velocity vectors show the development of the recirculation bubble in the wake of the different geometries. The Baseline-GTS (a, b) has a long and large bubble with counter-rotating vortices driving the flow in the wake. The straight (c, d) and the smooth tail (e, f) show smaller wakes and similar velocity vectors, with these geometries pulling the flow towards the center of the wake. The round tail geometry (g, h) shows separation before the end of the tail, and quite a difference in flow structures. Viewing the round tail from above, as in (h), one notes an increase in velocity in the middle of the wake.
Figure 5.18: Velocity profiles in the wake of the GTS, 1w behind the Baseline, for the different boat-tail geometries at z=w/2 and y=h/2 respectively. The velocities are higher for the boat-tail geometries, and does not include as much negative velocity. The round tail looks to have more turbulence and less smooth velocity profiles, and shows an increase in velocity at the center just as seen previously.
Figure 5.19: Comparison of the vorticity in the wake at $x=1w$. With narrower tail the shear layers forming from the boundary layer of the GTS moves further in towards the centerline. All images show large disturbances in the bottom of the wake, an area heavily influenced by the struts below the GTS.
(a) Baseline geometry, XY-plane
(b) Baseline geometry, XZ-plane.

(c) Straight tail geometry, XY-plane.
(d) Straight tail geometry, XZ-plane.

(e) Smooth tail geometry, XY-plane.
(f) Smooth tail geometry, XZ-plane.

(g) Rounded geometry, XY-plane.
(h) Rounded geometry, XZ-plane.

Figure 5.20: TKE profiles in the wake of the GTS, 1w behind the Baseline, for the different boat-tail geometries at z=w/2 and y=h/2 respectively. TKE is lower for the straight and round boat-tails, XZ profiles follow Baseline but smaller and show distinct shear layers top and bottom.
Figure 5.21: Comparison of pressure fields for the different boat-tails. Baseline shows large pressure difference between front and back, with a low pressure region at the base of the wake. The boat-tail geometries show a pressure increase at the base of the wake compared to Baseline. The round tail shows large pressure gradients at base of wake.
**Figure 5.22:** Comparison of the rounded tail geometries without suction and with different suction slot locations, at $z=w/2$ and $y=h/2$. The wake is shorter with the suction applied. The geometries with suction top and bottom show higher negative velocities in the wake. There is also more ground interaction for the geometries with slots, as can be seen on the bottom of fig. (c), (e) and (g).
No suction.

High slot location.

Mid slot location.

Low slot location.

Figure 5.23: Comparison of velocity profiles of \( i \)-direction velocity in the XY-plane at \( z = \frac{w}{2} \). The section at 0.5\( w \) shows large negative velocities for the high and low suction locations.
Figure 5.24: Comparison of velocity profiles of $i$-direction velocity in the XZ-plane at $y=h/2$. The profiles look a lot more similar than in the XY-plane, but still there are larger differences when comparing the mid slot location to the other locations. All profiles with suction show asymmetry although they are from symmetric geometry, and the results have been averaged over 500 iterations.
Figure 5.25: Comparison of pressure fields for different suction slot locations. With suction there is a large increase in pressure at the base of the wake. Instabilities in the solution showing in the wake area.
CFD was used to simulate the flow behind a 1/8th scale GTS model of a simplified tractor and trailer geometry. Three different boat-tails were added to the GTS to assess their impact on velocity, pressure, drag force and overall flow field structures. Further on suction slots were added to one boat-tail geometry, and three different slot locations were investigated to determine their effect on drag and flow behaviour.

There is no contest to the benefits of adding a boat-tail to the back of a blunt shape. Both this study and others show that the improvements are quite drastic. The results in this study showed a reduction of drag by 22%, and other studies show results that are similar. The shape does not even need to be that complex to be efficient, which case 2.2 with the straight tail showed with its 18% decrease in drag. Hopefully the new EU legislation will urge manufacturers and truckers to work towards using boat-tails to effectively save the environment, and reduce the costs both for society and themselves.

The different boat-tail shapes showed the design process can not always be intuitive. A shape that looks smooth and streamlined may not always have those qualities, as evidenced by case 2.3 with the round tail. By using CFD the effectiveness of a design can be easily measured and some of its qualities be deduced. Designers and CFD engineers working together cross-disciplinary would probably benefit the design process and lead to more efficient geometries quicker.

In cases 3.1-3.3 it was investigated how a suction slot could be used to improve a sub-optimal design of a boat-tail. The results showed that the location of the slot highly influenced the outcome, even if the location was only shifted slightly. The effect of suction on trucks, and similarly pressure, is a field that has garnered a lot of study in recent years [41][44], and will probably be the next step in optimizing the aerodynamics of trucks and trailers.

The results from cases 3.1-3.3 are a bit questionable, as the flow field looks unstable. The round tail profile does not have a clearly defined separation point which leads to instabilities in the flow field. Residuals and $C_D$ plots (fig. D.2) does not show big differences in convergence compared to the Baseline (fig. 5.4).
Probably the RANS model has a hard time reaching a stable solution and when the suction slots are added a clear solution is even harder to reach. These cases would benefit from using a DES simulation which is more accurate in predicting wake flows [34].
Chapter 7

Future work

The work started in this thesis could be continued by running more simulations on the different geometries, but instead using more accurate numerical models such as the DES modification of RANS. Better models would lead to more accurate results and the complex flow field could be more faithfully captured, in turn leading to a better prediction of the real life flow field.

The cases involving suction would also benefit from a more accurate numerical model which can resolve fluctuations in an unstable wake. Suction slots could be investigated with different pressures and slot angles. Further on a pressure outlet could be added, as this would be closer to a situation in reality and would also be a more efficient use of the power needed for such a device. Other flow controlling techniques such as the use of pulsing flow or using a plasma actuator could be of interest too. With accurate enough models, a feedback loop could be used that modelled the pressures to the frequencies of the vortex shedding of the shape and actively promote a more stable and efficient flow, for example in crosswinds.

Another approach would be to use a geometry that resembles reality more, and the interaction between trailer and complex geometry like side mirrors, wheels and tractor-trailer gap would be investigated. This could in turn lead to an optimized boat-tail shape that could be used for a full scale experiment with a real truck.

Building directly on this thesis, and moving on to a real, full-size truck experiment, the easiest route would probably be to use the straight tail shape. The straight tail shape is easy to build, can easily be modified, and still shows good improvements on drag. The straight tail shape could first be used for a parametric study with a Design of Experiments approach together with CFD to investigate the effect of slant angle on side, top and bottom, together with the length of the tail. Then with the results from the study the full-size experiment would have more data to build a model boat-tail from.

All of the above would be more easily performed once legislation has been set so the optimizations can be done with the proper constraints.
Appendix A

The Trailair project

This thesis started with the help of Peter Georén, project manager at KTH Transport Labs, who put me in contact with Per Gyllenspetz who works with the Trailair project. Trailair targets the reduction of fuel consumption of trucks by focusing on the aerodynamic drag of the trailer. Trailair is not only about reducing drag, but also about making a product that can be easily implemented, used and put on market. Within the project a business analysis has been made, and there are parallel studies such as [23] on logistics.

My part has been to investigate what kind of geometries could be used for a boat-tail. The findings from this thesis will come to use later in the Trailair project when there could be a real size field-testing of a boat-tail on a tractor-trailer combo.
Appendix B

Case 2: Additional plots

Figure B.1: Velocity in $i$-direction for the straight tail in the XY-plane at $z=w/2$. The wake is smaller than the Baseline, and tapers off as a consequence of the geometry.

Figure B.2: Velocity profiles of $i$-direction velocity in the XY-plane at $z=w/2$. 
Figure B.3: Velocity in $i$-direction for the straight tail in the XZ-plane at $y=h/2$.

Figure B.4: Velocity profiles of $i$-direction velocity in the XZ-plane at $y=h/2$. 

![Velocity profile diagram](image-url)
Figure B.5: Evolution of vorticity throughout the wake of the straight tail geometry. White area in (a) is inside the geometry.
Figure B.6: TKE for the straight tail geometry, view from the side, in the XY-plane at $z=w/2$.

Figure B.7: TKE profiles for the straight tail geometry, in the XY-plane at $z=w/2$.

Figure B.8: TKE for the straight tail geometry, view from the top, in the XZ-plane at $y=h/2$.

Figure B.9: TKE profiles for the straight tail geometry, in the XZ-plane at $y=h/2$. 
Figure B.10: Pressure for the straight tail, in XY-plane at $z=w/2$ and XZ-plane at $y=h/2$.

Figure B.11: Velocity in $i$-direction for the smooth tail geometry, view from the side, in the XY-plane at $z=w/2$.

Figure B.12: Velocity profiles of $i$-direction velocity in the XY-plane at $z=w/2$. 
Figure B.13: Velocity in $i$-direction for the smooth tail geometry, view from the top, in the XZ-plane at $y=h/2$.

Figure B.14: Velocity profiles of $i$-direction velocity in the XZ-plane at $y=h/2$. 

Velocity profiles XZ-plane

- $0.25w$
- $0.5w$
- $1w$
- $2w$
- $3w$
Figure B.15: Evolution of vorticity throughout the wake of the smooth tail geometry. White area in (a) is inside the geometry.
Figure B.16: TKE for the smooth tail geometry, view from the side, in the XY-plane at $z=w/2$.

Figure B.17: TKE profiles for the smooth geometry, in the XY-plane at $z=w/2$. 
**Figure B.18:** TKE for the smooth tail geometry, view from the top, in the XZ-plane at $y=h/2$.

**Figure B.19:** TKE profiles for the smooth tail geometry, in the XZ-plane at $y=h/2$.

**Figure B.20:** Pressure for the smooth tail, in XY-plane at $z=w/2$ and XZ-plane at $y=h/2$. 
Figure B.21: Velocity in $i$-direction for the rounded geometry, view from the side, in the XY-plane at $z=w/2$.

Figure B.22: Velocity profiles of $i$-direction velocity in the XY-plane at $z=w/2$. 
Figure B.23: Velocity in $i$-direction for the round geometry, view from the top, in the XY-plane at $y=h/2$.

Figure B.24: Velocity profiles for the round geometry in the XZ-plane at $y=h/2$. 
Figure B.25: Evolution of vorticity throughout the wake of the round tail geometry. Assymetries showing in solution. White area in (a) is inside the geometry.
Figure B.26: TKE for the round geometry, view from the top, in the XY-plane at $z=w/2$.

Figure B.27: TKE profiles for the round geometry, in the XY-plane at $z=w/2$. 
**Figure B.28:** TKE for the round geometry, view from the top, in the XZ-plane at $y=h/2$.

**Figure B.29:** TKE profiles for the round geometry, in the XZ-plane at $y=h/2$.

**Figure B.30:** Pressure for the round boat-tail, in XY-plane at $z=w/2$ and XZ-plane at $y=h/2$. 
Figure B.31: Residuals plot for the round boat-tail.

Figure B.32: $C_D$-plot for the round boat-tail.
Appendix C

Case 3: Additional plots

Figure C.1: Velocity in $i$-direction for the round tail with high suction slot, in the XY-plane at $z=w/2$.

Figure C.2: Velocity in $i$-direction for the round tail with high suction slot, in the XZ-plane at $z=w/2$. 
**Figure C.3:** TKE for the round tail with high suction slot, view from the side, in the XY-plane at $z=w/2$.

**Figure C.4:** TKE for the round tail with high suction slot, view from the top, in the XZ-plane at $y=h/2$. Asymmetry showing in wake.

**Figure C.5:** Pressure for the round tail with high suction slot, in XY-plane at $z=w/2$ and XZ-plane at $y=h/2$. Asymmetry in the wake in the XZ-plane as a result of an unstable solution.
Figure C.6: Velocity in $i$-direction for the round tail with mid suction slot, in the XY-plane at $z=w/2$.

Figure C.7: Velocity in $i$-direction for the round tail with mid suction slot, in the X-plane at $z=w/2$.

Figure C.8: TKE for the round tail with mid suction slot, view from the side, in the XY-plane at $z=w/2$.

Figure C.9: TKE for the round tail with mid suction slot, view from the top, in the XZ-plane at $y=h/2$. 

Figure C.10: Pressure for the round tail with mid suction slot, in XY-plane at $z=w/2$ and XZ-plane at $y=h/2$. Asymmetry in XZ-plane as a result of an unstable solution.

Figure C.11: Velocity in $i$-direction for the round tail with low suction slot, in the XY-plane at $z=w/2$.

Figure C.12: Velocity in $i$-direction for the round tail with low suction slot, in the XZ-plane at $y=h/2$. 
Figure C.13: TKE for the round tail with low suction slot, view from the side, in the XY-plane at $z=w/2$.

Figure C.14: TKE for the round tail with low suction slot, view from the top, in the XZ-plane at $y=h/2$.

Figure C.15: Pressure coefficient for the round tail with low suction slot, in XY-plane at $z=w/2$ and XZ-plane at $y=h/2$. Unstable solution showing in XZ-plane.
Figure C.16: Residuals plot for the round boat-tail with high suction slot.

Figure C.17: $C_D$-plot for the round boat-tail with high suction slot.
Appendix D

GTS Geometries

Figure D.1: GTS geometry used in [3][7]. All measurements non-dimensionalized by trailer width $w$. Struts are used in simulations, but not included in this figure or their results.
Figure D.2: GTS geometry used in [4][5][6].


See for example the Indianapolis 500 winners 1915 vs. 1916, the 1924 Djelmo Land Speed Record Car or Louis Coatalen’s 1927 1000 bhp Sunbeam.

See for example the Chaparral 2B or 2E.


K. R. Cooper "The Wind Tunnel Testing of Heavy Trucks to Reduce Fuel Consumption". SAE 821285. 1982


R. McCallen, F. Browand, J. Ross "The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains". 2009


Trucking.org, "Trucking Industry Fact Sheet", 2014

J. Zaya, "Effektivare transportkedjor för näringslivet - Förstudie Aerodynamik", 2014

Eurostat, "Number of lorries, road tractors, trailers and semi-trailers in the EU 2012", (Eurostat, online data code: road_eqs_lrstn) 2012


[28] Lewis F. Richardson, "Weather Prediction by Numerical Process". 1922


[34] CD-adapco, "STAR-CCM+ User Guide v9.06".


[42] M. Mihaescu (Personal communication per e-mail, non-official report on suction slots), 2015-04-30


[45] CFD Online http://www.cfd-online.com