Optimal placement of plasma actuators on trucks:
A drag reduction study using adjoint methods

Samuel Gottfarb Bart
Abstract

The fuel consumption of vehicles plays an important environmental role and ways of improving the fuel economy is beneficial for a sustainable future. Active flow control is one method that can potentially reduce the aerodynamic drag significantly and therefore the fuel consumption. A plasma actuator introduces momentum in its vicinity and can be used for active control of the flow.

This thesis aimed to find the optimal placement of plasma actuators on the A-pillars of a truck, using Reynolds-Averaged Navier-Stokes (RANS) simulations performed with STAR-CCM+. The adjoint solver was used to obtain a sensitivity map of the drag with respect to the variation of momentum. This was then used when the accuracy of this adjoint solution was evaluated by performing parametric studies where the actuation was placed at different locations around the point of flow separation.

A simple case consisting in a half-submerged cylinder was first studied as both wind tunnel experiments and Large Eddy Simulations (LES) have been previously performed on this flow case at KTH. When placing actuation on top of the cylinder, the drag reduction obtained with RANS was 4.3% which was comparable to the previous LES work, where the reduction was 4.65%. With the flow separating at 98 degrees in a local cylindrical coordinate system, which starts at the leading edge of the cylinder, the adjoint solution showed that the optimal placement was located at 105 degrees. The actuation was placed at several locations between 90 and 118 degrees and the minimum was found to be located at 106 degrees.

The Ground Transportation System (GTS) model representing a generic tractor-trailer combination, both in two and three dimensions, was then used with the same solution procedure. In the two-dimensional case, the flow was found to separate at 63 degrees. The optimal placement predicted by the adjoint solver was 69 degrees, while the parametric study showed the optimal location to be at 75 degrees. The added complexity of an extra dimension resulted in the wind speed having to be lowered in order to produce similar results for the actuation. However, agreement on the optimal location was observed between the two- and three-dimensional cases.

In general, the adjoint solver showed varying levels of accuracy between simulations, but still gave a qualitative indication of the optimal placement. A correlation between the point of separation of the flow and the adjoint solution was observed and along with the results from the parametric studies, the optimal placement was concluded to be slightly downstream of the separation point. In order to obtain the exact optimal position, a parametric study is needed for each individual case.
Acknowledgements

First and foremost I would like to thank my examiner Gunilla Efraïmsson and my supervisor Romain Futrzynski. Gunilla for giving me the opportunity to perform this thesis and her enthusiasm when overseeing my work and Romain for the guidance, interesting discussions and for welcoming me and making me feel part of the aerodynamics group.

Thank you to Emma Alenius for being my co-supervisor and helping me with insightful tips on how to handle STAR-CCM+ and general theory on computational fluid dynamics.

Lastly, thank you to Amin, Borja, Ciáran, Johan, Louis, Nicolas, Stefan and Wei for the interesting and pleasant group meetings, the numerous lunches as well as the after work activities.
## Contents

4 Results

4.1 Half-Submerged 2D Cylinder .............................................. 18
   4.1.1 Primal Solution .................................................... 18
   4.1.2 Adjoint Solution .................................................. 19
   4.1.3 Parametric Study ................................................. 21
   4.1.4 Multiple Actuators ................................................ 23

4.2 The GTS Model in 2D ..................................................... 24
   4.2.1 Headwind Case ...................................................... 24
      4.2.1.1 Primal Solution ............................................ 24
      4.2.1.2 Adjoint Solution ....................................... 25
      4.2.1.3 Parametric Study ..................................... 26
   4.2.2 Crosswind Case .................................................... 26
      4.2.2.1 Primal Solution ........................................ 26
      4.2.2.2 Adjoint Solution ..................................... 27

4.3 The GTS Model in 3D ..................................................... 27
   4.3.1 Primal Solution .................................................... 27
   4.3.2 Adjoint Solution .................................................. 28
   4.3.3 Parametric Study ................................................ 29

4.4 Summary ................................................................. 30

5 Conclusion ........................................................................... 31

References ............................................................................. 32
Chapter 1

Introduction

1.1 Project Background

In today’s world, where the environment is a major concern, the fuel consumption in transportation plays an important role. When a truck is travelling at its maximum allowed speed, the aerodynamic drag is significant. Reducing this drag by just a few percent is very beneficial to the fuel consumption and therefore the environment. There are a large number of ways to reduce drag with passive flow control but as the conditions on the road are highly changeable and unsteady, passive flow control methods are not always optimal. Active flow control devices have great potential as they have the benefit of being adaptable to the specific conditions at a particular time whether it be high speed, low speed, or different kinds of crosswind conditions.

Plasma actuators present one way to control the flow as they accelerate the nearby fluid. This can be used to delay separation if it is placed in the right location, which would have a significant influence on the drag. Plasma actuators also have the benefit of being energy efficient, meaning that the drag reduction observed is usually larger than the cost of powering the actuation. Since the flow tends to separate at sharp corners and large regions of separation usually go hand in hand with large drag forces, the plasma actuators could also be used so that the corners can be sharper and therefore allow for more space inside the truck while maintaining an acceptable drag level. In this thesis, the focus on reducing the drag force is the main objective. The aim of this thesis is therefore to find the optimal placement of plasma actuators on the A-pillars of a truck. The A-pillars are located where the side view mirrors are mounted (see Figure 1.1).

Figure 1.1: A Scania R620 4x2 Topline [1].
Wind tunnel experiments and Large Eddy Simulations (LES) have been performed on flow cases with plasma actuators at the Mechanics Department and the Department of Aeronautical and Vehicle Engineering at KTH respectively. In order to provide a basis for future research, the Reynolds-Averaged Navier-Stokes (RANS) model is used since it is much faster than LES. This allows for easier testing of different locations in order to find the optimal placement. Performing optimization studies in this situation by placing the actuation at different locations may not be the most efficient approach. The adjoint solver available in STAR-CCM+ introduces a faster way to obtain the same result with a single simulation. This is an optimization method from control theory which has been used in fluid dynamics for a couple of decades now. This method gives a way of obtaining the sensitivity map of a cost function so that the optimal placement can be obtained without any further analyses. This has a lot of potential so this thesis compares the methods of using a parametric study and using the adjoint solver.

1.2 Objective

The objective of this thesis is to find the optimal placement of a plasma actuator on a half-submerged cylinder and the Ground Transportation System (GTS) model for a few selected wind conditions. This will be done in a parametric study where the actuation is placed around the separation point on the geometry. In parallel to this, the adjoint solver will be used in order to help determine regions of where the optimal placement might be found. The secondary aim of this thesis is to evaluate the accuracy of the adjoint solution against the results obtained in the parametric studies.

1.3 Approach Overview

There are three main steps in the solution procedure in this thesis and they are as follows:

- Firstly, the primal solution, which is a standard flow solution, is obtained using RANS with a realizable $k-\varepsilon$ turbulence model.

- Secondly, the adjoint solver is run in STAR-CCM+ with the primal solutions as bases. This provides the sensitivity of the selected cost function, the drag force, with respect to variation of momentum at every point in space.

- Thirdly, the parametric study is performed where a new primal solution is obtained with a simplified model of the momentum added by a plasma actuator placed at different locations.
1.4 Thesis Outline

The outline of this thesis is as follows:

- **Chapter 2** describes the theory of the relevant subjects in this thesis. The theory of fluid dynamics is briefly explained where the governing equations, the Reynolds-Averaged Navier-Stokes equations and the $k-\varepsilon$ equations are presented. This is followed by a description on adjoint methods, and finishes with a description of plasma actuators.

- **Chapter 3** includes the models and methods where the model of the plasma actuator is first presented. This is followed by the numerical setup for the primal solution for the different cases. After this, the methods for obtaining the primal solutions and the adjoint solutions are presented, ending with a description of the scope of the parametric studies.

- **Chapter 4** presents the results obtained from the primal solution, the adjoint solution and the parametric study for the different cases.

- **Chapter 5** gives the conclusion along with recommendations for future work.
Chapter 2

Theory

In this section the theory relevant to this thesis will be presented. Beginning with a description of how the governing flow equations used by many Computational Fluid Dynamics (CFD) solvers are obtained. Then the RANS as well as the $k-\varepsilon$ equations are described. This is followed by a section on the adjoint solver theory. Finally, a description of the theory of plasma actuators is presented.

2.1 Fluid Dynamics

2.1.1 Governing Equations

The governing equations constitute the conservation of three different physical quantities, namely mass, momentum and energy. In words these can be described as:

- The mass of a fluid particle is constant.
- The variation of the momentum of a fluid particle is equal to the external forces.
- The variation of the energy of a fluid particle is equal to the sum of the work done by the external forces and the heat flux.

The mass flow through the boundary of a fluid particle is given by density $\rho$ times the normal component of the velocity $\mathbf{u} = [u, v, w]$ times the area. When summing up the terms, which can be seen in Figure 2.1a, and dividing them with the volume the following equation is obtained:

$$\frac{D\rho}{Dt} + \rho \frac{\partial u_k}{\partial x_k} = 0, \quad (2.1)$$

Here the notation of $\frac{D}{Dt}$ represents the material derivative and is in this case given by:

$$\frac{D\rho}{Dt} = \frac{\partial \rho}{\partial t} + \mathbf{u} \nabla \rho. \quad (2.2)$$

The momentum equations are obtained in a similar manner. When deriving these equations, one has to consider body and surface forces. For instance the pressure forces (denoted by $p$) and the viscous forces (denoted by $\tau$). Assuming a case with a Newtonian fluid and summing up the terms in each direction, where one example can be be seen in Figure 2.1b, and adding a term $F_i$ for body forces (such as gravity), the following is obtained:

$$\rho \frac{D u_i}{Dt} = -\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_i} + \rho F_i, \quad \text{where} \quad \tau_{ij} = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3} \frac{\partial u_r}{\partial x_r} \delta_{ij} \right). \quad (2.3)$$
Chapter 2. Theory

The energy equation is obtained by taking the terms for the work done by surface forces and the energy flux due to heat conduction. Using the same assumption of a case with a Newtonian fluid followed by some rewrites, the following is obtained:

\[
\frac{De}{Dt} = -p \frac{\partial u_i}{\partial x_i} + \Phi + \frac{\partial}{\partial x_i} \left( \kappa \frac{\partial T}{\partial x_i} \right), \quad \text{where} \quad \Phi = \tau_{ij} \frac{\partial u_i}{\partial x_j}. \quad (2.4)
\]

(a) Mass flow through a fluid particle. (b) The stresses in the x-direction.

Figure 2.1: Mass flow and stresses [2].

2.1.2 Reynolds-Averaged Navier-Stokes

Reynolds-Averaged Navier-Stokes, or RANS, are modified equations of the fluid flow. The assumption behind this is that it is possible to separate the time dependent turbulence velocity fluctuations from the mean flow. The variables are therefore split into a time-averaged part and a fluctuating part. The velocity vector \( \mathbf{u} \) is then equal to \( \mathbf{U} + \mathbf{u}' \) and the pressure \( p = P + p' \).

Introducing this decomposition into the instantaneous continuity equation and the momentum equations and averaging gives the following incompressible equations:

\[
\frac{\partial \bar{u}_i}{\partial t} = 0, \quad \tag{2.5}
\]

\[
\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial (\bar{u}_i \bar{u}_j)}{\partial x_j} = - \frac{1}{\rho} \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial \bar{u}_i}{\partial x_j} - \rho \bar{u}'_i \bar{u}'_j \right). \quad \tag{2.6}
\]

In this equation, \( \rho \bar{u}'_i \bar{u}'_j \) are the so called Reynolds stresses and these require specific turbulence modeling. In all the simulations performed in this thesis, the realizable \( k-\varepsilon \) model was used.

2.1.2.1 The \( k-\varepsilon \) model

The standard \( k-\varepsilon \) model is a two-equation model with one equation for the turbulent kinetic energy \( k \) and one for the turbulent dissipation rate \( \varepsilon \). The following transport equations are used:

\[
\frac{\partial (\rho k)}{\partial t} + \text{div}(\rho k \mathbf{U}) = \text{div} \left[ \frac{\mu_t}{\sigma_k} \text{grad} k \right] + 2 \mu_t S_{ij} S_{ij} - \rho \varepsilon, \quad \tag{2.7}
\]

\[
\frac{\partial (\rho \varepsilon)}{\partial t} + \text{div}(\rho \varepsilon \mathbf{U}) = \text{div} \left[ \frac{\mu_t}{\sigma_\varepsilon} \text{grad} \varepsilon \right] + C_{1\varepsilon} \frac{\varepsilon}{k} 2 \mu_t S_{ij} S_{ij} - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}, \quad \tag{2.8}
\]

where \( S_{ij} \) is the mean strain rate and \( \sigma_k, \sigma_\varepsilon, C_{1\varepsilon} \) and \( C_{2\varepsilon} \) are dimensionless constants determined by data fitting for many different cases of turbulent flow. The variables \( k \) and \( \varepsilon \) are then used to define the velocity scale \( \vartheta \) and the length scale \( \ell \):
\[ \vartheta = k^{1/2}, \quad \ell = \frac{k^{3/2}}{\varepsilon}. \tag{2.9} \]

The eddy viscosity \( \mu_t \) is then defined using dimensional analysis:

\[ \mu_t = C \vartheta \ell = \rho C_\mu \frac{k^2}{\varepsilon}, \tag{2.10} \]

where \( C_\mu \) is a dimensionless constant determined in the same way as for the previously mentioned constants. Lastly, using the Boussinesq relationship, the Reynolds stresses are obtained. [2]

\[ -\rho u_i'u_j' = \mu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} \rho k \delta_{ij} = 2 \mu_t S_{ij} - \frac{2}{3} \rho k \delta_{ij}. \tag{2.11} \]

## 2.2 Adjoint Methods

The adjoint method comes from control theory and it offers a way to understand the influence of parameter variation on a solution. In fluid dynamics, these parameters can typically be related to flow variables, the shape of a geometry or boundary conditions.

There are two approaches to the adjoint, the continuous and the discrete approach. The continuous approach is derived from the continuous governing equations. This approach has two main issues. Firstly, the differentiation followed by the discretization results in an inconsistency between the gradient and the discrete implementation. Secondly, the boundary conditions can be complicated to define. In the discrete approach, which is the one STAR-CCM+ uses, the adjoint is applied to the discretized governing equations. The numerical procedure thus becomes similar to one of a normal flow solution. Disadvantages include storage requirements and that the implementation can be difficult to differentiate. [8]

Consider \( \omega \) to represent the flow-field variables, for instance the momentum, and \( F \) to be the model parameter, for example the position of the physical boundary of a geometry. Let \( I \) be the cost function, for instance the drag over a body so that:

\[ I = I(\omega, F). \tag{2.12} \]

A change in \( F \) results in:

\[ \delta I = \frac{\partial I^T}{\partial \omega} \delta \omega + \frac{\partial I^T}{\partial F} \delta F, \tag{2.13} \]

where the superscript \( T \) stands for transpose. Now consider the independence of \( \omega \) and \( F \) and that the residuals \( R \) of the discretized flow equations are zero:

\[ R(\omega, F) = 0. \tag{2.14} \]

This means that

\[ \delta R = \frac{\partial R}{\partial \omega} \delta \omega + \frac{\partial R}{\partial F} \delta F = 0. \tag{2.15} \]

As \( \delta R \) is zero it can be multiplied by a Lagrange multiplier \( \psi \), and subtracted from Eq. 2.13:

\[ \delta I = \frac{\partial I^T}{\partial \omega} \delta \omega + \frac{\partial I^T}{\partial F} \delta F - \psi^T \left( \frac{\partial R}{\partial \omega} \delta \omega + \frac{\partial R}{\partial F} \delta F \right) \]

\[ = \left( \frac{\partial I^T}{\partial \omega} - \psi^T \frac{\partial R}{\partial \omega} \right) \delta \omega + \left( \frac{\partial I^T}{\partial F} - \psi^T \frac{\partial R}{\partial F} \right) \delta F. \tag{2.16} \]
Chapter 2. Theory

The Lagrange multiplier $\psi$ is then chosen to satisfy the adjoint equation:

$$\left(\frac{\partial R}{\partial \omega}\right)^T \psi = \frac{\partial I}{\partial \omega},$$

(2.17)

where $\partial R/\partial \omega$ is sensitivity of the flow residual with respect to the flow variables and $\partial I/\partial \omega$ is the sensitivity of the cost function with respect to the flow variables. This system of linear equations is solved by the same iterative defect-correction algorithm used in the coupled implicit solver in STAR-CCM+. This is the adjoint equation and it eliminates the first term in Eq. 2.16 which leaves us with:

$$I = \left(\frac{\partial I^T}{\partial F} - \psi^T \frac{\partial R}{\partial F}\right) F.$$

(2.18)

Eq. 2.18 is independent of the flow variables $\delta \omega$ which means that the gradient of $I$ can be determined without additional evaluations of the flow-field. It is also independent of the number of design variables which enables for example the opportunity for shape optimizations where a large number of points on the geometry would be the model parameters. [9], [10]

2.3 Plasma Actuators

Plasma actuators consist of two electrodes, separated by an insulator (dielectric). The configuration of electrodes considered here, also referred to as Single Dielectric Barrier Discharge (SDBD) and a photo of a plasma actuator can be seen in Figure 2.2. The actuator in the photograph was built at the KTH Mechanics Department [4].

![Configuration of an SDBD](image1)

(a) Configuration of an SDBD. [3]

![Photo of an actual actuator](image2)

(b) Photo of an actual actuator. [4]

Figure 2.2: Configuration and a photo of a plasma actuator.

The actuator is driven by a high frequency and high voltage current. This creates a weakly ionized plasma, meaning that only a small fraction of the air molecules become ions. These ionized molecules are accelerated towards the two electrodes depending on the charge of the electrodes as well as the charge of ions. The ions collide with the neutral air thus transferring some of their momentum. The forward discharge happens during first half-cycle when the negatively charged ions are accelerated forward towards the covered electrode. The backwards discharge occurs during the next half-cycle when the negatively charged ions accelerate back towards the exposed electrode. As the electron momentum transfer collision cross section is smaller compared to that of the positive ions and the number of positive ions are about two times larger than the negative ones, a net force is thus created during each half-cycle.

The question of why the forces from each half-cycle do not cancel out over a complete cycle remains. The reason for this is the dielectric. When the electrons accelerate towards the covered electrode, they gather on the dielectric as opposed to being neutralized when they reach the exposed electrode. This means that the charge density is around one order of magnitude higher
when the backwards discharge starts. When the negative ions reach the exposed electrode, they are neutralized thus resetting the process and the cycle can repeat. The momentum addition vectors near the end of the two half-cycles can be seen in Figure 2.3. The vectors have been scaled by a factor of 20 for the sake of clarity. [3]

Figure 2.3: The momentum vectors and the potential contours (V) near the end of the discharges [3].
Chapter 3

Methods

In this chapter the methods used for obtaining the results are presented starting with the model for the plasma actuator. The numerical setup and method for obtaining the primal solution is then presented followed by a description on how the adjoint solution was obtained. Lastly, the parametric study is described.

3.1 Model of the Plasma Actuator

Figure 3.1a presents a schematic representation of the same lines as Figure 2.3 from the previous chapter. As the force decays exponentially [5], most of the momentum addition is generated near the wall. There, the electric field lines are almost parallel with the surface so the model can be simplified without a great loss of accuracy.

Futrzynski [6] describes the following exponential model for a local Cartesian coordinate system (see Figure 3.1b) where only the component in the tangential direction is considered:

\[ f = \begin{cases} F_{\text{max}} e^{-\zeta x - \xi y} \cdot \tau & \text{if } (x, y) \geq (0, 0) \\ 0 & \text{otherwise}. \end{cases} \]  

(3.1)

Figure 3.1: Electric field lines and a local coordinate system. [6]

Futrzynski [6] describes the following exponential model for a local Cartesian coordinate system (see Figure 3.1b) where only the component in the tangential direction is considered:

\[ f = \begin{cases} F_{\text{max}} e^{-\zeta x - \xi y} \cdot \tau & \text{if } (x, y) \geq (0, 0) \\ 0 & \text{otherwise}. \end{cases} \]  

(3.1)

The coefficient \( F_{\text{max}} \) describes the maximum body force while \( \zeta \) and \( \xi \) describe the rate of decay in the normal and tangential direction. Futrzynski performed optimization studies in order to match these values against the wind tunnel experiments done with the in-house built plasma actuator at the KTH Mechanics Department by Vernet [4]. In that case, the actuator was placed at \( \theta = 90 \) degrees in quiescent air. These values can be seen in Table 3.1.
Table 3.1: Coefficients for the exponential model. [6]

<table>
<thead>
<tr>
<th>$F_{\text{max}} \ [N/m^3]$</th>
<th>$\xi$</th>
<th>$\zeta$</th>
</tr>
</thead>
<tbody>
<tr>
<td>21,900</td>
<td>2,050</td>
<td>3,375</td>
</tr>
</tbody>
</table>

Integrating Eq. 3.1 over the entire domain of $x$ and $y$ gives the following:

$$\int_0^\infty \int_0^\infty F_{\text{max}} e^{-\xi x - \zeta y} \, dx \, dy = 3.165 \cdot 10^{-3} \ \text{N per m width} \quad (3.2)$$

The magnitude of the actuation can be seen in Figure 3.2. As the force extends continually, the discrete nature of the mesh introduces inaccuracy. To ensure that the integral in Eq. 3.2 remains constant and mesh independent for different locations of the actuation, the value of $F_{\text{max}}$ was varied for each case.

Figure 3.2: Plasma Magnitude.

3.2 Numerical Setup

3.2.1 Half-Submerged 2D Cylinder

Wind tunnel experiments on a half-submerged cylinder in both quiescent air and with non-zero freestream velocities have been done by Vernet [4]. These experiments have been complimented by Large Eddy Simulations done by Futrzynski [6]. The simulations in this thesis were set up with the same domain size, albeit reduced to two dimensions, and with the same freestream velocities of 5 and 8 m/s. The ceiling was therefore a regular no-slip surface and the conditions at the inlet extracted from the wind tunnel experiment were used as a boundary condition. The domain can be seen in Figure 3.3.
Polyhedral cells were used in the whole the domain except close to the walls where a prism layer was used in addition to a refinement region in the wake. A mesh dependency study was performed in the manner described in Section 3.3 which resulted in a mesh size of 470,000. The entire mesh can be seen in Figure 3.4.

On the surface of the cylinder, 20 prism layers were used with a constant cell height giving a $y^+$ value at the wall of 1 or less. In the region where the actuator is to be placed the number of layers is 60, to allow a better resolution of the body force. This can be seen in Figure 3.5.
Using the height \( h \) of the cylinder, the following Reynolds numbers were used:

\[
\begin{align*}
\text{Re}_8 &= \frac{u_{\infty,1} h}{\nu} = \frac{8 \cdot 0.05}{1.568 \cdot 10^{-5}} = 25,500,
\end{align*}
\]

\[
\begin{align*}
\text{Re}_5 &= \frac{u_{\infty,2} h}{\nu} = \frac{5 \cdot 0.05}{1.568 \cdot 10^{-5}} = 15,900,
\end{align*}
\]

where \( \nu \) is the kinematic viscosity.

### 3.2.2 The Ground Transportation System

In order to simulate a geometry closer to a truck, a variation of the Ground Transportation System was used. NASA used the GTS model in a wind tunnel experiment where they aimed for very high precision so that CFD results could be validated against theirs [7]. This model represents a generic tractor-trailer combination without a tractor-trailer gap and a cab over engine design. The model in three dimensions and the schematics can be seen in Figure 3.6 and 3.7. The model originally included struts on which the physical model was mounted in the wind tunnel experiment, but these were removed as they introduce unnecessary complexity to the geometry while not representing wheels that well. This only affected the three-dimensional case.

![Figure 3.6: The variation of Ground Transportation System model used.](image)

![Figure 3.7: The schematics of the GTS model viewed from the side. All dimensions are normalized to one truck width \( w = 32.38 \) cm [7].](image)

#### 3.2.2.1 The GTS Model in 2D

The domain and geometry used in the two-dimensional case can be seen in Figure 3.8. This is a top-down view of half the truck. A symmetry plane was used and since the A-pillar was the target location for the actuation, the back, and therefore the wake, was considered not to be
of interest for this study and was removed. As this flow case was not directly related to wind tunnel experiments, the ceiling in this domain was setup to allow slip.

Figure 3.8: The GTS model in two dimensions.

A mesh dependency study was performed in the manner described in Section 3.3 and when concluded, a mesh size of 870,000 was reached. The polyhedral mesh can be seen in Figure 3.9.

Figure 3.9: The polyhedral mesh in the entire domain.

On the GTS surface, 10 prism layers were used with a constant cell height giving a $y^+$ value at the wall of 1 or less. In the region around the A-pillar where the actuator is to be placed the number of layers is 60. This can be seen in Figure 3.10.
One early question that emerged was which speed should be used in this case. Using the speed of 8 m/s, for the two-dimensional GTS model resulted in no separation of flow on the geometry. This was observed for speeds larger than 2.5 m/s. Running the adjoint solver on a case like this gave a solution where the adjoint solver predicted that the drag basically cannot be influenced by variation of momentum. Using the speed of 2.5 m/s resulted in a separated flow and with the length of 16 cm representing half the width of the GTS model gives:

\[
Re = \frac{2.5 \cdot 0.16}{1.568 \cdot 10^{-5}} = 25,500
\]  

(3.5)

3.2.2.2 The GTS Model in 3D

In this case, half of GTS model was used with a symmetry plane. The geometry and two refinement regions for the mesh can be seen in Figure 3.11. The outer wall and ceiling were surfaces where slip was allowed. The ground was a moving wall where a prism layer with 8 layers and a constant cell height was used. A mesh dependency study was performed in the manner described in Section 3.3 and when concluded, a mesh size of 3.0 million was reached. A polyhedral mesh was used and on the truck 25 prism layers were used with a \( y^+ \) value at the wall of 1. There, a geometric progression was used with a stretch factor of 1.1. The mesh can be seen in Figure 3.12 and 3.13.
Figure 3.11: The full domain with two mesh refinement regions around the model.

Figure 3.12: The mesh viewed from the front.
In Figure 3.14, it is shown where the actuation was located. The actuation starts at the bottom of the truck moving up until the rounded edge representing the cab over engine design begins. This equals 23 cm of actuation.

Around the A-pillar, separation did not occur for a freestream velocity of 2.5 m/s with the actuation turned off. As the two- and three-dimensional geometries have identical features, this was most likely due to three-dimensional effects from the undercarriage and the cab over engine design. In order to ensure separation, the inlet speed was lowered to 1 m/s. Using half the width of the model as the characteristic length resulted in the following Reynolds number:

$$Re = \frac{1 \cdot 0.16}{1.568 \cdot 10^{-5}} = 10,200$$

(3.6)

### 3.3 Primal Solver

Steady state RANS simulations with a realizable $k-\varepsilon$ model, an SST (Menter) $k-\omega$ model and Reynolds Stress Model (RSM) were all tested in the case of the cylinder at 8 m/s. Compared with the LES, both $k-\omega$ and RSM presented inferior results to the ones the realizable $k-\varepsilon$ which was the reason that this model was used throughout this thesis. To solve these equations, the coupled implicit solver was used with 2nd order precision.

Convergence was judged by monitoring drag and lift and in addition to this, the static pressure and velocity of a few selected points outside the boundary layer near the cylinder and GTS model respectively were also monitored. These parameters were also used to ensure a mesh independent solution, where a coarse mesh was first used and refined until the discrepancy of the monitored values was insignificant to achieve the results targeted in this thesis. When determining the separation point, both the wall shear stress and the tangential velocity in the local cylindrical coordinate system were considered.
3.4 Adjoint Solver

With the primal solutions as bases, corresponding adjoint simulations were performed. The cost function was chosen to be the pressure drag over the cylinder and truck geometries respectively. The pressure drag dominated the viscous drag in these cases, and as the viscous drag is mainly reduced by reducing the speed on the surface, this would go against the way the actuator works. Hence, only the pressure drag was considered.

As the assumption in Eq. 2.14 is that the flow residuals are zero, very good convergence is needed for the primal solution. Preferably machine precision. One way to decrease the size of the residuals is by freezing the turbulence solver when the turbulence residuals have converged. This is possible since the adjoint solver does not consider the turbulence when solving for the sensitivity.

The standard method used by STAR-CCM+ when solving the adjoint is an iterative defect-correction algorithm. For cases that are difficult to converge, a generalized minimum residual method (GMRES) is available. This aims at minimizing the $l^2$-norm of the adjoint in the direction that is given by the defect correction update at every step. Turning on the GMRES increases the iteration time and memory usage significantly. [10]

3.5 Parametric Study

Parametric studies were performed in accordance with the adjoint solutions where the plasma actuation was placed on locations surrounding the most sensitive regions according to the adjoint. The following locations were used for the different cases:

- **Cylinder**: Actuation at 22 locations from 90 to 118 degrees.
- **2D GTS Headwind**: Actuation every 2.5 degrees starting at 55 degrees with three extra locations after the rounded corner.
- **2D GTS Crosswind**: No parametric study was performed.
- **3D GTS**: Actuation every 5 degrees starting at 55 degrees, ending with actuation at 90 degrees.
Chapter 4

Results

In this chapter, the results of the primal solution, the adjoint solution and the parametric study will be presented for the different cases.

4.1 Half-Submerged 2D Cylinder

4.1.1 Primal Solution

In Figure 4.1, the velocity in the streamwise direction for the 8 m/s case can be seen. A local cylindrical coordinate system was used to localize the separation point which occurs at 98 degrees.

![Figure 4.1: The velocity field in the streamwise direction for the 8 m/s case.](image)

Velocity profiles in the case of no actuation were compared with profiles obtained by LES [6]. Three locations were used: 1.9 cm upstream of the leading edge of the cylinder, at the top and 10 cm downstream of the trailing edge of the cylinder. They can be seen in Figure 4.2.

![Figure 4.2: Comparison of velocity profiles for three different locations.](images)
In Figure 4.2, it can be seen that the profiles match well before the cylinder. On top of the cylinder, close to the surface, a difference between the two models is starting to be noticeable while downstream, inside the wake, the differences are clear.

Looking at Table 4.1, it can be observed that the absolute values for the drag coefficients have a significant difference. However, the two models produced closer results in regard to the reduction one actuator achieves. It was therefore assumed that $\Delta C_D$ obtained with RANS is meaningful.

Table 4.1: Drag coefficient with and without actuation on top of the cylinder.

<table>
<thead>
<tr>
<th></th>
<th>No Actuation</th>
<th>Actuation</th>
</tr>
</thead>
<tbody>
<tr>
<td>RANS</td>
<td>0.49</td>
<td>0.47 (-4.3 %)</td>
</tr>
<tr>
<td>LES</td>
<td>0.43</td>
<td>0.41 (-4.65 %)</td>
</tr>
</tbody>
</table>

4.1.2 Adjoint Solution

The adjoint solver converged with very small residuals in this case. Machine precision was achieved in the primal solution using the method explained in Section 3.4. The adjoint residuals can be seen in Figure 4.3. The bump around iteration 500 is due to a significant increase in the Courant Number in order to speed up convergence.

Figure 4.3: The unscaled adjoint residuals.

Figure 4.4 shows the sensitivity of the pressure drag with respect to variation of momentum in the x-direction. It tells us that if momentum is increased in the positive x-direction, where the values are high, the biggest and most favourable effect on the drag will be achieved.
Chapter 4. Results

Figure 4.4: The adjoint sensitivity to momentum in the x-direction.

Figure 4.5 shows the sensitivity of the pressure drag with respect to variation of momentum in the y-direction. Here it can be observed that if momentum is increased upwards it will have a favourable effect on the drag if it is done were the values are positive and it will have an undesirable effect if it is done where the values are negative.

Figure 4.5: The adjoint sensitivity to momentum in the y-direction.

If these two sensitivities are considered components of a vector, they can be projected onto the local cylindrical coordinate system. This is shown in Figure 4.6:
The predicted maximum location is at 105 degrees with all locations between 100 and 110 degrees being candidates. It is also worth noting that the maximum region is located a small distance above the surface.

### 4.1.3 Parametric Study

The results from the parametric study can be seen in Figure 4.7. There the adjoint sensitivity from the previous section is transformed into a straight line so that the results can be compared more easily. The location where the reduction is the highest is 107 degrees where the drag is reduced by 8.6%. As the values do not produce a smooth line, a second degree polynomial was fitted to them. This shows the minimum to be located at 106 degrees. There, the drag reduction achieved is 7.2% so the variation in drag reduction around the minimum is on the order of 1 percent.
Chapter 4. Results

In Figure 4.8, the results for the 5 m/s case can be seen. A curve fitting with a second degree polynomial gives a minimum at 108 degrees.

![Figure 4.8: Drag reduction results for the 5 m/s case.](image)

Here it should be mentioned that the model of the plasma actuator has optimized coefficients for a case with actuation at 90 degrees on a cylinder in quiescent air. The actuator is dependent on the wind speed flowing past the electrodes. In this thesis, all the cases had non-zero freestream speeds, thus introducing uncertainty regarding the effect the actuator produces. Cases where the actuator was placed far downstream of the separation point, therefore inside the wake, also introduced uncertainty as RANS has problems of correctly capturing regions of separation. In Figure 4.9, the negative tangential velocity can be seen for two cases of actuation. In Figure 4.9a it is seen that a small region of negative velocity close to the cylinder occurred before the actuator, and ahead of the actuator, the separation was therefore almost completely avoided. In Figure 4.9b however, the actuator is placed too far downstream for the separation to be avoided upstream of the actuator resulting in only minor changes in the wake region. As the results were based on coefficients for a specific case, the bulk of this thesis did not produce precise predictions of the specific actuator used in the wind tunnel experiments at the Mechanics Department. This thesis used a reasonable model which gives an indication of what could be achieved by plasma actuators.

![Figure 4.9: Negative velocity in the cylinder tangential direction with actuation.](image)

(a) Actuation at 107 degrees.  
(b) Actuation at 118 degrees.
4.1.4 Multiple Actuators

In the 5 m/s case, multiple actuators were consecutively added. The adjoint solver was run on the case of actuation at 110 degrees. The adjoint solution can be seen in Figure 4.10 below.

![Figure 4.10: Adjoint sensitivity in the case of one actuator at 110 degrees.](image)

Here it can be observed that the drag can be further decreased by the addition of momentum. Two new maxima were found at 104 and 114 degrees. In Figure 4.11, the results of adding actuation to either of these two new locations and then running the adjoint solver can be seen:

![Figure 4.11: Adjoint solutions for the cases with two actuators.](image)

It can be seen in Figure 4.11a that after adding a second actuator at 114 degrees, the previous maximum of 104 degrees was replaced by a new one at 118 degrees. In Figure 4.11b, it can be seen that when adding the second actuator at 104 degrees, the maximum at 114 degrees remained. In Figure 4.12, the results of adding another actuator to these two new maxima and running the adjoint solver can be seen. There, it is shown that the adjoint suggests more momentum further downstream in both cases regardless of which initial step was taken:
The results for the different configurations of a second and third actuator are summarized in Table 4.2.

Table 4.2: Results for the cases with multiple actuators.

<table>
<thead>
<tr>
<th>No. of Actuators</th>
<th>Location (deg)</th>
<th>Drag Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>110</td>
<td>15 %</td>
</tr>
<tr>
<td>2</td>
<td>110 &amp; 114</td>
<td>28 %</td>
</tr>
<tr>
<td></td>
<td>104 &amp; 110</td>
<td>26 %</td>
</tr>
<tr>
<td>3</td>
<td>110, 114 &amp; 118</td>
<td>38 %</td>
</tr>
<tr>
<td></td>
<td>104, 110 &amp; 114</td>
<td>36 %</td>
</tr>
</tbody>
</table>

4.2 The GTS Model in 2D

4.2.1 Headwind Case

4.2.1.1 Primal Solution

Figure 4.13 shows the velocity field in the streamwise direction. It also shows a closer view of the A-pillar with a local cylindrical coordinate system where the separation point was found to be at 63 degrees. With the low speed of 2.5 m/s (Re = 25,500), there was one problem, namely that the strength of the actuation was too large. It was observed when performing the parametric study that when the plasma actuator was placed at 75 degrees in the local cylindrical coordinate system, separation did not occur and a significant drag reduction was achieved. When the actuation was moved further downstream the flow still remained fully attached and the drag reduction remained constant. With this in mind, the approach was altered in such a way that the values of $F_{\text{max}}$ were lowered to 10% of their original optimized values. This allowed a drag reduction level similar to that of the case of the cylinder.
4.2.1.2 Adjoint Solution

The adjoint solution for this case required the use of the Flexible GMRES to converge. The time per iteration became 20 times higher, but in return the solver converged after only a small number of iterations. This can be seen in Figure 4.14. It can also be noted that the level of convergence was not as good as in the case of the cylinder (see Figure 4.3).

The adjoint sensitivity of the pressure drag with respect to momentum in the surface tangential direction is shown in Figure 4.15. Here, the adjoint predicts the maximum influence downstream of the separation point, more precisely at 69 degrees.
4.2.1.3 Parametric Study

The actuation was placed every 2.5 degrees and the results in Figure 4.16 are plotted against the adjoint sensitivity which is transformed into a straight line. In this case, a smooth variation in the drag as a function of actuator position is observed. The maximum drag reduction is found at 75 degrees where the drag is reduced by 7.5%. This location is 6 degrees downstream of the location the adjoint solver predicted.

![Figure 4.16: Adjoint sensitivity and results in the 2D GTS case projected on a straight surface.](image)

4.2.2 Crosswind Case

4.2.2.1 Primal Solution

Lowering the speed to 2.5 m/s was necessary to ensure separation. This speed is about 10% of the maximum allowed speed for a truck in many countries, so a case with a steady crosswind was also tested. In that case, the wind speed was 25 m/s with a crosswind angle of 10 degrees. In these conditions, the simulation proved to be transient, due to the interaction between the two separations that were produced at the two trailing edges of the truck. As the adjoint solver does not support transient solutions, the domain and geometry was altered so that the this interaction was removed and the solution thus became steady. The geometry setup and velocity in the streamwise direction can be seen in Figure 4.17. It can also be seen that the separation occurs at the A-pillar at 55 degrees.

![Figure 4.17: The velocity field in the streamwise direction of the entire domain (left) and close to the A-pillar (right).](image)
4.2.2.2 Adjoint Solution

In Figure 4.18 below, the adjoint sensitivity in the tangential direction is shown. The region for which the maximum influence would occur is located at 60 degrees.

![Adjoint Solution](image)

Figure 4.18: The adjoint sensitivity of the pressure drag.

The domain setup did not present a case with a clear way to calculate the drag as the different sides were of different length. In addition to this, the actuation is not strong enough to produce a significant influence on the flow. It was therefore decided that no parametric study would be performed and solely allow the adjoint solver to provide a prediction. The adjoint solver performed similar to the other cases in that it predicts the optimal location to be downstream of the separation. In addition to this, the magnitude of the adjoint sensitivity is considerably lower with this higher speed of 25 m/s. These two results indicate that a drag reduction of similar significance is achievable with an actuator powerful enough.

4.3 The GTS Model in 3D

4.3.1 Primal Solution

The velocity field in a horizontal plane cutting the GTS model 15 cm from the bottom can be seen in Figure 4.19 below. Looking more closely on the A-pillar, it is found that the separation occurs at 68 degrees.

![Primal Solution](image)

Figure 4.19: The velocity field in the streamwise direction viewed from above of the entire domain (left) and close to the A-pillar (right).
4.3.2 Adjoint Solution

Using the Flexible GMRES once again produced convergence in a low number of iterations, which can be seen in Figure 4.20.

![Figure 4.20: The unscaled adjoint residuals.](image1)

The adjoint solver did produce results similar to previous cases and gave a similar indication of how the drag would be affected by momentum. In Figure 4.21, the adjoint sensitivity of the pressure drag in the surface tangential direction can be seen, where the maximum is located at 75 degrees.

![Figure 4.21: The adjoint sensitivity of the pressure drag.](image2)
4.3.3 Parametric Study

The results from the parametric study can be seen in Table 4.3. Note the practically constant level of reduction for the locations of 75, 80 and 85 degrees.

Table 4.3: Drag reduction for the 3D GTS model.

<table>
<thead>
<tr>
<th>Location (deg)</th>
<th>Drag Reduction</th>
</tr>
</thead>
<tbody>
<tr>
<td>55</td>
<td>2.5 %</td>
</tr>
<tr>
<td>60</td>
<td>3.7 %</td>
</tr>
<tr>
<td>65</td>
<td>5.3 %</td>
</tr>
<tr>
<td>70</td>
<td>7.6 %</td>
</tr>
<tr>
<td>75</td>
<td>8.7 %</td>
</tr>
<tr>
<td>80</td>
<td>8.9 %</td>
</tr>
<tr>
<td>85</td>
<td>8.9 %</td>
</tr>
<tr>
<td>90</td>
<td>4.4 %</td>
</tr>
</tbody>
</table>

The results presented in the Table 4.3 shows a constant reduction for the locations of 75, 80 and 85 degrees. Similar results were observed in the two-dimensional case with low speed and full actuation. There, separation did not occur when placing the actuator at 75 degrees or further downstream. When reducing the strength of the actuator, the optimal location was found to be at 75 degrees. In other words, the beginning of the region where the placement of the actuator resulted in the flow remaining completely attached. This was observed in the three-dimensional case as well. The region of separation was small to begin with and it almost disappeared for these locations of the actuator. As the adjoint solution predicted 75 degrees to be the optimal placement, concluding that the optimal placement is at the beginning of this region of constant reduction would give perfect correspondence with the adjoint solution.

However, the previously mentioned conclusion is questionable. The adjoint solver has been consistently underpredicting the optimal actuation angle and it is therefore unlikely that perfect correspondence is the case between two and three dimensions. The three dimensional case is the most complex flow case compared to the other ones in this thesis. In addition to this, limitations in mesh size and low speed also provide uncertainty. At 90 degrees, the drag reduction is noticeably lower than at 85 degrees. This was not observed in the two-dimensional case with full actuation where the drag reduction remained constant at 90 degrees as well. The actual optimal location in the three-dimensional case is therefore difficult to pin down so the result is left as the optimal placement being between 75 and 85 degrees.
4.4 Summary

The results regarding the point of separation from the primal solutions, the point where the adjoint sensitivities are the highest as well as the optimal placement found in the parametric studies are summarized in Table 4.4 below.

Table 4.4: Summary of the results.

<table>
<thead>
<tr>
<th>Case</th>
<th>Separation</th>
<th>Adjoint Max</th>
<th>Parametric Study</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cylinder: 8 m/s</td>
<td>98°</td>
<td>105°</td>
<td>106°</td>
</tr>
<tr>
<td>Cylinder: 5 m/s</td>
<td>95°</td>
<td>103°</td>
<td>108°</td>
</tr>
<tr>
<td>2D GTS: Headwind</td>
<td>63°</td>
<td>69°</td>
<td>75°</td>
</tr>
<tr>
<td>2D GTS: Crosswind</td>
<td>55°</td>
<td>60°</td>
<td>-</td>
</tr>
<tr>
<td>3D GTS</td>
<td>68°</td>
<td>75°</td>
<td>75-85°</td>
</tr>
</tbody>
</table>

In the case of the cylinder with the speed of 8 m/s, the correspondence between the adjoint solution and the parametric study is very good. This was the only case out of the five where machine precision was achieved in the primal solution which in turn allowed for the adjoint solver to converge with higher precision. This was not the case in the other flow cases.

The adjoint solver in general has been underpredicting the optimal actuation angle. It has consistently predicted this location to be 5 to 8 degrees downstream of the separation point while the parametric studies showed this location to be a few degrees further downstream. Regardless of the case or the size of the residuals, this correlation between the separation point and the adjoint maximum has been obvious.
Chapter 5

Conclusion

The flow over two geometries, a half-submerged cylinder and the GTS model, was simulated with several configurations of Reynolds numbers. A plasma actuator model was used to report the drag reduction against the position of the actuator which was compared to sensitivity results from solving the adjoint problem.

Placing plasma actuators on half-submerged cylinders or around the A-pillars of GTS models in order to reduce the drag is best done by adding the momentum slightly downstream of the separation point. The adjoint method currently implemented in STAR-CCM+ gives an area of high sensitivity which can be used to predict this location. The parametric study shows that the drag reduction varies around a minimum although the exact minimum varies between the two methods. In order to find the optimal placement with high precision, a parametric study is required for each case.

Future work

As a next step, studying cases with actual wind speeds trucks experience would be of interest. In these conditions, separation might not occur at the A-pillar unless the curvature of the rounded edge is increased. It would be interesting to investigate whether plasma actuation can keep the flow attached or at least significantly reduce the drag for sharper curvatures.

Studying the relationship between the wind speed and the magnitude of the actuation is also of interest, so the understanding of the potential of the plasma actuator can be generalized.
References


[10] CD-adapco, STAR-CCM+ Documentation