CFD Study of the Flow around a High-Speed Train

Florian GUILLOU
Supervisor : Tomas Muld
Examinator : Gunilla Efraimsson

Master Thesis
Kungliga Tekniska Högskolan, Stockholm
Abstract

This document is a report summing the master thesis work dealing with the Computational Fluid Dynamic (CFD) study of the flow around a high-speed train. The model is a scaled 1:50 generic train with two cars, one inter-car gap and simplified bogies. A platform is set on the side of the train since one of the aim of the study is to look at the consequences of the phenomena in the wake on people or objects standing on the platform. The slipstream is one of this phenomena, it is due to the fact that the viscous air is dragged when the train is passing. If too strong, it can move or destabilize people or objects on the platform.

In addition of the slipstream study, a velocity profile study, a drag and lift coefficients analyze as well as a Q-factor study and a frequency study have been realized. Some results of these different studies are compared with the ones obtained on the same model with a Delayed Detached Eddy Simulation (DDES).

Since the flow is turbulent, for those different studies, the flow has been simulated with a Reynolds Averaged Navier–Stokes equation model (RANS) which is the k-ω SST model for the turbulence.

The study of the slipstream allowed to calculate the Technical Specification for Interoperability (TSI) which must not be higher that the European Union requirement set at 15.5 m/s, the result obtained is 8.1 m/s which is then lower than the limit.

The velocity profile shows similarities with the DDES results even though it is less detailed. The same conclusion is done for the Q-plot where is clearly visible the two counter-rotating vortices in the wake.

Finally, a Fast Fourier Transform algorithm has been applied to instantaneous velocity results in the wake of the train in order to get the frequency of the aerodynamic phenomena in that wake. The main frequency is 25 Hz and corresponds to a Strouhal number of 0.1, quite closed to the results obtained with DDES which is 0.085.
The results of the RANS and DDES are reasonably similar and by regarding at the large difference between the cell numbers (respectively 8 500 000 and 20 000 000) it can be conclude that in some ways the RANS model can be preferred at the DDES to save time for the computation but it does not contain the small scales resolved by the DDES.

Descriptors: Computational Fluid Dynamic, slipstream, Reynolds Averaged Navier Stokes model, k-ω SST model, aerodynamic, Aerodynamic Train Model, wake, high-speed train

Acknowledgment

First I would like to thank Tomas Muld who offered me this subject and supervised me during my 20 weeks of master thesis in KTH. He gave me a lot of useful advices which helped me to progress in my work and he helped me in the redaction of my report.

Also, I would like to thank Gunilla Efraimsson for her support and for her assistance in my work.

I do not forget to thank my three master thesis colleagues in my office for the 20 weeks spend together.

Finally, I will thank all my friends and people who were next to me during this master thesis and also during my two years in Sweden.
Table of Contents

Abstract ......................................................................................................................... 2

Acknowledgment ........................................................................................................... 3

1. Introduction .................................................................................................................. 5

2. Method .......................................................................................................................... 8
   2.1. Theory ....................................................................................................................... 8
   2.2. Train model and mesh study .................................................................................. 10
   2.3. Drag and lift coefficients study .............................................................................. 15
   2.4. Velocity profile study ........................................................................................... 16
1. Introduction

Among the different ways to travel such as airplane, car, bus or boat, the train is the safest. According to the report of the ECST [7], between 2001/2002 the death per 100 million person kilometers is in the European Union 0.035 and the death per 100 million person hours is 2 which makes the rail the safest way to travel before ferry (respectively 0.25 and 8), aircraft (respectively 0.035 and 16) and bus (respectively 0.07 and 2). In addition, with the progress of technology, train is able to drive at high-speeds. On, the 3rd of April 2007 a TGV in France reached the velocity of 574.8 km/h. However this speed is only a record, and the TGV trains drive generally between 200 and 330 km/h. The International Union of Railways (UIC) defines that the speed of a high-speed train must be at least 200 km/h for upgraded track and 250 km/h for new track. For example, the TGV line between Paris and Strasbourg runs between 310 and 320 km/h.
With the increase of the train speed, aerodynamics has become an important key in the rail vehicles field. The study of the aerodynamics of a train can lead to substantial cost savings and more environmental friendly trains. The most studied phenomenon in the realization of a train is the drag generated by the displacement of the train in the air flow. Reducing the drag leads to a reduction of the amount of energy needed. But some others aerodynamic phenomena are of interests, such as the pressure variation while the train is driving in a tunnel, the study of the consequences of a crosswind, the study of the train rollover or the slipstream. A short overview of different aerodynamic phenomena is given below.

When a train runs, a strong head pressure pulse is created at the very front of the train which leads to a change of pressure in the surrounding. A low pressure bubble is also created at the rear part of the train but is less strong than the first one, and is mostly a problem for people or objects standing near to the track.

The pantograph is situated in an area where the flow conditions change a lot. In order to avoid unauthorized large variations of the contact surface it is important that the flow around the pantograph is not too turbulent, which can be enabled by adding some so called fairings.

While entering a tunnel the air at the train nose is compressed which creates an overpressure wave that migrates at the speed of sound. When this waves reaches the end of the tunnel a part of the wave is reflected and goes back as an underpressure wave. As the train tail enters the tunnel an underpressure wave is created and migrates to the end of the tunnel. The pressure variation is maximum when an underpressure wave meets a reflected overpressure wave. The pressure difference reaches then a peak value. This mostly causes discomfort for the passenger since he is subject to a high pressure difference in a short time, for example the legislation in Sweden is 1500 Pa in 4 seconds.

The bogies movement is restricted to the tracks. A suspension system connects the train body to the bogies. The train can then roll, yaw and pitch. A yawing moment can increase in strong crosswinds. This can be very dangerous in case of strong crosswind and particular yaw angle and can lead to overturning.

Aerodynamic noise comes from the pressure fluctuations that occur in the turbulent boundary layer and as the flow separates. Vortices produce also a lot of aerodynamic noise (von Karman vortex during flow separation). It is therefore important to have attached flow since the separation of the boundary layer creates a lot of noise. One
can say that the power of external aerodynamic noise grows with about the sixth power.

In this report, the aerodynamic phenomenon that is studied is slipstream. The train surfaces are affected by strong shear stress. The area where the friction force significantly affects the air velocity is called the boundary layer. Inside it, the flow can be either laminar or turbulent. The length of the train being quite long, the flow inside the boundary layer is assumed turbulent which leads to strong local velocity variation. The boundary layer thickness is defined as the area where the velocity is 99% of $U_\infty$. The viscosity of the fluid leads to the fact that the air is dragged by the train which creates a slipstream. The slipstream can be decomposed in four regions that can be seen on the figure 1:

![Figure 1: Different regions of the slipstream](image)

The region 1 is characterized by the velocity which increases and then decreases creating a pressure pulse. The region 2 contains the boundary layer which gets thicker and thicker. The region 3 is the near wake which appears after the passage of the rear part of the train and contains different phenomena depending on the type of the train and the shape of the rear part since the flow can separate as a separation
bubble or two counter-rotating vortices. And the region 4 is the far wake, which appears long after the train has passed and contains disturbed flow but less than in the near wake region. Depending on the type of the trains some regions are more important. For example, for a freight train the boundary layer region is where the largest slipstream velocities occur than for a high-speed train it is more the near wake region which predominates in the slipstream.

The slipstream can generate some risks. People or objects such as pushchair and luggage can be moved and be destabilized if the slipstream velocities are too strong. In the Official Journal of the European Union [5], some regulations are mentioned regarding slipstream: at a height of 0.2 m above the top of rail and at a distance of 3 m from the track center, the full length train running should not create an exceedance of the air speed of 20 m/s if it is running between 190 and 249 km/h and 22 m/s if it is running between 250 and 300 km/h. This regulation is called the Technical Specification for Interoperability (TSI).

Aerodynamic studies can be either experimental or computational. In this master thesis, the study has been done by mean of computational fluid dynamic. The software used is StarCCM+ V.6 from CD-Adapco.
2. Method

2.1. Theory

In this thesis, the flow around the train is solved with StarCCM+ which solves the incompressible Navier–Stokes equation:

\[ \rho \frac{\partial v_i}{\partial t} + \rho v_j \frac{\partial v_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{2}{3} \frac{\partial}{\partial x_i} \left( \frac{\partial v_j}{\partial x_j} \right) f_i \]  \hspace{1cm} (1)

and the incompressible continuity equation:

\[ \text{div} \mathbf{v} \equiv 0 \]  \hspace{1cm} (2)

The equations (1) and (2) form a system of four equations with four unknowns: the three velocity components \( v_i \) and the pressure \( P \). These equations are non-linear partial differential equation which means that there is no analytical solution for a problem with arbitrary boundary conditions. Instead, the flow around the train is solved numerically by discretizing the equations.

Depending on the behavior of the flow, different model can be used. It has to be noticed that the flow can become unstable with a high Reynolds number and the flow becomes chaotic in which the pressure and the velocity change continuously with time within substantial regions. It is the cause of the difference between a laminar and a turbulent flow. If a laminar flow used good and practical model to solve the Navier–Stokes equation, a turbulent flow is difficult to model due to the chaotic behavior of the flow which contains small scales. The flow around a high-speed train becomes turbulent a few centimeters after the front part.

The Direct Numerical Simulation (DNS) of turbulent flow using very fine mesh and very small times step are used to resolve the smallest turbulent phenomena and the fastest fluctuations. It then resolves the turbulence without modeling. Even though DNS is a very solid method, it is impossible to resolve the turbulence for high Reynolds numbers and this method is not used for industrial flow computations since it is too expensive [1]. Therefore, the turbulence needs to be modeled.
The method used in the simulations of this study is the Reynolds–Averaged Navier–Stokes equations model (RANS) which is based on the decomposition of the flow parameters in two components. This decomposition will give the so-called Reynolds Averaged Navier–Stokes equation which contains a new term that will be described and interpreted depending on the chosen model.

Within the Reynolds decomposition, the different properties like velocity can be decomposed into a mean value and fluctuations.

For example:

\[ \bar{v} = \bar{v} + \bar{v}' \]  

(3)

Where \( \bar{v} \) is the mean velocity and \( \bar{v}' \) is a fluctuating velocity.

The Reynolds–Averaged Navier–Stokes equation is obtained by introducing the Reynolds decomposition (3) in the equation (1) and by taking the average of it. The result is then:

\[
\frac{\partial}{\partial x_j} \left( \bar{v}_i \frac{\partial \bar{v}_j}{\partial x_i} \right) = - \frac{\partial}{\partial x_i} \bar{P} \frac{\partial}{\partial x_j} \bar{v}_i \frac{\partial}{\partial x_j} \bar{v}_j - \bar{v}_i \bar{v}_j' - \bar{v}_i' \bar{v}_j
\]  

(4)

The last term of the equation is the turbulent stress tensor. The challenge is to model this stress tensor in terms of the mean flow quantities, and hence provide closure of the governing equations [12]. Eddy viscosity models use the concept of a turbulent viscosity \( \mu_t \) to model the Reynolds stress tensor. The most common model is the Boussinesq approximation:

\[
- \bar{v}_i \bar{v}_j' = \mu_t \frac{\partial \bar{v}_i}{\partial x_j} \frac{\partial \bar{v}_j}{\partial x_i}
\]  

(5)

Now, \( \mu_t \) needs to be determined. Some simpler models rely on the concept of mixing length to model the turbulent viscosity in terms of the mean flow quantities. The eddy viscosity models in STAR-CCM+ solve additional transport equations for scalar quantities that enable the turbulent viscosity \( \mu_t \) to be derived. These include one-equation models such as Spalart–Allmaras, and two-equation models such as k–\( \varepsilon \) and k–\( \omega \) models.
**k-ω SST model**

The model used for the different simulations is the k-ω SST model. It is a hybrid solution which uses the k-ε model for the far field and the k-ω model near the wall.

The k-ε model takes mainly in consideration how the turbulent kinetic energy is affected. This model has two model equations, the transport equation for mean turbulent kinetic energy (k) and the transport equation for the dissipation of mean kinetic energy (ε). The dissipation occurs in the smallest eddies.

The standard k-ε model uses the following transport equations. The transport equation for mean turbulent kinetic energy (k):

\[
\frac{\partial}{\partial t} (\rho k) - \nabla \cdot (\rho k \mathbf{U}) = \nabla \cdot [\nu \nabla k] + \frac{2}{\kappa} \mathbf{S}_{ij} \mathbf{S}_{ij} - \varepsilon 
\]  

and the transport equation for the rate of dissipation of mean kinetic energy (ε):

\[
\frac{\partial}{\partial t} (\rho \varepsilon) - \nabla \cdot (\rho \varepsilon \mathbf{U}) = \nabla \cdot [\nu \nabla \varepsilon] + C_1 \frac{\varepsilon}{k} \frac{\nabla k}{k} - C_2 \frac{\rho \varepsilon^2}{k} 
\]  

Equations (6) and (7) can be explained as:

- **Local rate of change of** k/
- **Transport by** convection
- **Transport by** diffusion
- **Rate of production of** k/
- **Rate of destruction of** k/

The previous model presents some disadvantages. It is not satisfactory in the near-wall boundary with adverse pressure gradients [1]. For these cases k-ω model can be used.

Near the wall, a new variable is used: the turbulence frequency \( \omega = \frac{\varepsilon}{k} \). The difference with the k-ε model is that the second equation models the dissipation related to the frequency of the large eddies. In the equation (7) of the k-ε model, \( \varepsilon \) is replaced with \( \varepsilon = k \cdot \omega \) which gives the transport equation of the turbulence frequency \( \omega \) as below:
\[
\frac{\partial U_i}{\partial t} + \text{div} \{ U \text{grad} U \} - S_{ij} \cdot \left[ -\frac{2}{3} \frac{\partial U_i}{\partial x_j} - \frac{1}{2} \sum_{k=1}^{2} \frac{\partial k}{\partial x_i} \frac{\partial k}{\partial x_k} \right] = 0
\]

It can be seen that a last term appears. It is the cross-diffusion term and comes from the \( \varepsilon = k \cdot \omega \) transformation of the diffusion term in the \( \varepsilon \) -equation [1].

The \( k-\omega \) SST model uses a \( k-\varepsilon \) model in the far field and a \( k-\omega \) model near the wall [1].

The different values of the constants in the equations (6), (7) and (8) are set to the standard values in StarCCM+.

2.2. Train model and mesh study

The model studied is a generic train model at scale 1:50 which is called the Aerodynamic Train Model (ATM). It has 2 cars, 1 inter-car gap and simplified bogies.

The computational domain is a parallelepiped with a length of 5.347m, a width of 1.09m and a height of 0.585m. The dimensions are the same as in [11] and are chosen to mimic an experimental setup in a water towing tank. The train is 1m long and is assumed to have a cross-section area of 0.0049m\(^2\). The computational domain also contains a platform.

The characteristic length of the train is chosen to be the hydraulic diameter which is in full scale 3m, which means that at 1:50 the characteristic length is 3/50=0.06m. This value is used in the turbulent model.

Boundary conditions

Different boundary conditions are used: wall for all the train, platform and far field, inflow for the inlet and pressure outlet for the outflow.

At the inlet, Dirichlet conditions are used, which means that all the three velocity components and two turbulent quantities are prescribed. The values can be found in table 1:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity</td>
<td>15 m/s</td>
</tr>
<tr>
<td>Hydraulic diameter</td>
<td>0.06 m</td>
</tr>
<tr>
<td>Turbulent length scale</td>
<td>0.006 m</td>
</tr>
<tr>
<td>Turbulence intensity</td>
<td>0.30%</td>
</tr>
</tbody>
</table>
At the wall, no-slip has been applied, which means that the tangential velocity of the flow at the wall is zero. For the wall, an all-$y+$ wall treatment has been used which leads to the fact that a Neumann boundary condition is used for the turbulent kinetic energy: $\frac{\partial k}{\partial n} = 0$ at the wall.

In StarCCM+, the wall treatment is the different assumptions used to model the flow near the wall. These assumptions depend on the chosen model. These assumptions change depending on the wall distance which is called $y+$ and is given as [12]:

$$y^+ = \frac{y u^*}{\nu}$$

where $y$ is the distance from the cell centroid to the wall, $u^*$ is the reference velocity and $\nu$ is the kinematic viscosity. Three types of wall treatment are given by StarCCM+ [12]:

- the high-$y+$ wall treatment that uses the wall-function-type approach in which one considers that the near-wall cells lie within the logarithmic region of the boundary layer.

- the low-$y+$ wall treatment is used only for low Reynolds number. It assumes that the viscous sublayer is properly resolved.

- The all-$y+$ wall treatment is a combination of the two treatments above. It uses the high-$y+$ wall treatment for coarse meshes and the low-$y+$ wall treatment for fine meshes.

In the simulation, the all-$y+$ wall treatment has been applied since it is available with the k-$\omega$ SST model and it is the recommended one [12].

The ground and the platform are moving wall with the speed at 15m/s to simulate the movement relative to the train.

**Grid definition**

The model of the mesh used is a trim-hexaedral model with a prism layer mesher available in StarCCM+. In figure 2, a view of the mesh on the train is shown. It is seen that the mesh is finer for the tail of the train.
On figure 3 can be seen the six different volumetric controls created for the simulation: one for the head, one for the body, four for the tail and the wake.

Three different grids have been generated and used for comparison, Coarse Mesh (CM), Medium Mesh (MM) and Fine Mesh (FM).
The details of the grid are here describe for the medium mesh since it is the one which is later used for the simulations. For the medium mesh the base size is 0.02m, in the far field the relative minimum size is 20% of the base size and the relative target size is 80% of the base size. The cell size is bigger in the far field since it is not here that the turbulence needs to be resolved with a fine grid. Six different volumes have been created in order to get finer grid in the important areas i.e. at the front of the train, along the train and in the wake. The cells of the platform are also finer than in the rest of the volume since a probe is placed there in order to study the slipstream.

Here it can be found a summary of the different grid parameters for the far field for the case 2:

<table>
<thead>
<tr>
<th>Base size (depending on the case)</th>
<th>0.02 m</th>
</tr>
</thead>
<tbody>
<tr>
<td>Number of prism layer</td>
<td>5</td>
</tr>
<tr>
<td>Prism layer stretching</td>
<td>1.2</td>
</tr>
<tr>
<td>Prism layer thickness</td>
<td>25%</td>
</tr>
<tr>
<td>Growth rate</td>
<td>slow (four equal sized cell layers/transition)</td>
</tr>
</tbody>
</table>

*Table 2: Far field parameters*

These values are not always the same depending on the volumetric control. The 6 volumetric controls can be defined as the following: head, body, tail1, tail2, tail3 and tail4. The “tail” volumetric controls are ranked from the smallest to the biggest. At these 6 volumetric controls can be added the platform which has specific parameters for its surface.

For the different volumetric controls the relative size of the cells are greatly smaller than for the rest of the grid, also the number of prism layers and the prism layer thickness have changed.

<table>
<thead>
<tr>
<th>Volumetric control</th>
<th>Relative size (percentage of base)</th>
<th>Number of prism layer</th>
<th>Prism layer thickness</th>
</tr>
</thead>
<tbody>
<tr>
<td>Head</td>
<td>5%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Body</td>
<td>10%</td>
<td></td>
<td></td>
</tr>
<tr>
<td>Tail1</td>
<td>5%</td>
<td>7</td>
<td>20%</td>
</tr>
<tr>
<td>Tail2</td>
<td>10%</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Master Thesis – Florian Guillou – KTH
Table 3: Volumetric controls parameters

<table>
<thead>
<tr>
<th>Tail3</th>
<th>20%</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tail4</td>
<td>20%</td>
</tr>
</tbody>
</table>

For the platform the relative minimum size has been set at 15\% of the base size while the target size has been set at 60\% of the base size.

The values are the same for the three grids except for the base size which is 0.015m for the fine mesh and 0.03m for the coarse mesh. This leads to the following numbers of cells: for the coarse mesh, the grid contains 3 000 000 cells, 8 500 000 cells for the medium mesh and 18 000 000 cells for the fine mesh.

A steady turbulent flow using a Menter SST k-\omega model using the Reynolds–Averaged Navier–Stokes equation (RANS) is used for the grid comparison.

In order to verify the grid, the results of the different simulations have been compared to previous results of velocity plots obtained in [6]. The drag and lift coefficients have been calculated to check their coherence and the wall y+ value has been checked in order to verify that it was not too high. The CM, the MM and the FM velocities have been compared for different lines in the flow of the grid. The first is perpendicular to the flow at a height of 0.024m and 1.1m behind the train. The second is parallel to the flow and in the symmetry line of the train at a height of 0.024m and the last one is parallel to the flow at 0.06m away from the center of the train on the platform and at a height of 0.1m.

### 2.3. Drag and lift coefficients study

In order to check the good behavior of the simulation for the grids, the drag and lift coefficients are good parameters to calculate. If the calculated coefficients do not change between grids, the result is assumed to be grid independent.

Drag and lift coefficients are defined as below:
\[ C_D = \frac{D}{\frac{1}{2} \rho V^2 S} \]  

(10)

\[ C_L = \frac{L}{\frac{1}{2} \rho V^2 S} \]  

(11)

where \( S \) is the reference area defining as the area of the cross-section of the train body which is set at 12.35 m\(^2\) for the full-size ATM, \( D \) is the drag force, \( L \) is the lift force and \( V \) is the free stream velocity.

Different scale models have been used when doing the Reynolds number dependency study. The parameters of the grid is the same for all scales, except the base size which is 0.04 m for 1:25 scale, 0.1 m for 1:10 scale and 1 m for full-size scale.

The behavior of these two coefficients depending on the Reynolds number has not been studied before for this geometry. Different Reynolds numbers were used to compare the different coefficients and if they are changing with it. To do so, the speeds have been changed and the size of the train also. The different cases are summarized in the table 4:

<table>
<thead>
<tr>
<th>Case number</th>
<th>Scale</th>
<th>Velocity</th>
<th>Reynolds number</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>scaled 1:50</td>
<td>10 m/s</td>
<td>40 000</td>
</tr>
<tr>
<td>2</td>
<td>scaled 1:50</td>
<td>15 m/s</td>
<td>60 000</td>
</tr>
<tr>
<td>3</td>
<td>scaled 1:50</td>
<td>20 m/s</td>
<td>80 000</td>
</tr>
<tr>
<td>4</td>
<td>scaled 1:50</td>
<td>25 m/s</td>
<td>100 000</td>
</tr>
<tr>
<td>5</td>
<td>scaled 1:25</td>
<td>15 m/s</td>
<td>120 000</td>
</tr>
<tr>
<td>6</td>
<td>scaled 1:25</td>
<td>30 m/s</td>
<td>240 000</td>
</tr>
<tr>
<td>7</td>
<td>scaled 1:10</td>
<td>20 m/s</td>
<td>400 000</td>
</tr>
<tr>
<td>8</td>
<td>scaled 1:10</td>
<td>30 m/s</td>
<td>600 000</td>
</tr>
</tbody>
</table>
2.4. Velocity profile study

One of the goals of this study is to analyze the velocity profile in the wake of the high-speed train. Since the flow is turbulent and unsteady in the wake, the velocity behavior in the wake is very chaotic.

For this study the results are given for the case 2. The simulation is unsteady RANS. Since the flow is unsteady, the velocity profile depends on time and some time steps are needed in order to have proper results and converging residuals, that is why the computational time is set at 1 second. The time step is set at 1.33 ms for the case 2. Here the results are given for the last iteration so at 1s.

2.5. TSI study

This study allows to check if the aerodynamic loads on passengers on a platform does not exceed the norm. It is related to the slipstream which can destabilize objects or people on a platform. According to the Official Journal of the European Union: “A full length train, running in the open air at a reference speed $U=200\text{km/h}$ shall not cause the air speed to exceed value $U_{2\sigma}=15.5\text{m/s}$ at a height of 1.2m above the platform and at a distance of 3.0m from the track centre, during the whole train passage (including the wake)”. The velocity $U_{2\sigma}$ is defined as: “The value $U_{2\sigma}$ is the upper bound of the $2\sigma$ confidence interval of the maximum resultant induced air speeds in the $x$-$y$ platform plane. It shall be based on at least 20 separate measurements and under similar test conditions with ambient wind speeds of less than or equal to $2\text{m/s}$”. That is:

$$U_{2\sigma}=\underline{\overline{U_{2\sigma}}}(12)$$
where
\( \bar{U} \) is the mean value of all air speed measurements
\( \sigma \) is the standard deviation

In order to get the instantaneous velocity on different points on the specific line defined by the TSI requirement, a line of probes of 5 meters (1 meter before the front part of the train and 3 meters after it) has been created in the grid at the precise distance and height specified above. The different probes are spaced by 0.01m, the total amount of probes is hence 500. Since the train is running at 15m/s, the data have been collected every 0.01/15=0.67ms.

The unsteady simulation duration has been set at 1s which is enough time so that 20 probes have moved through the domain. Due to transient in the beginning of the solution, only the data after 0.1s is saved.

The data needed to be export in order to treat them since StarCCM+ can not do it. They were treated via Matlab. A Matlab code has been written in order to plot the velocity profile of different time departure which correspond of different probes along the line in a ground fixed coordinate system.

35 time departures have been taken into account (so more than 20 measurements), all from the beginning of the line probe so 1m before the front part of the train. The velocity is normalized by \( U_n = 1 - \frac{U}{U_\infty} \).

### 2.6. **Q-Plot**

The Q-factor is used to emphasize the vortices. Here it is especially the vortices in the wake that are interesting to illustrate. The Q-factor is defined as:

\[
Q = \Box_y \Box_y - S_y S_y \]  \tag{13}

where \( \Box_y = \frac{1}{2} \frac{\partial u_i}{\partial x_j} - \frac{\partial u_j}{\partial x_i} \) and \( S_y = \frac{1}{2} \frac{\partial u_i}{\partial x_j} \frac{\partial u_j}{\partial x_i} \) are the antisymmetrical and symmetrical part of the velocity gradient respectively.
By looking at the equations, it can be seen that a high positive value of $Q$ identifies regions where the vorticity magnitude dominates the strain rate. This means that the $Q$ criterion visualizes vortex cores.

In the results part, a $Q$-isosurface has been plotted for a value of 500 for $Q$ which was enough to distinguish vortices phenomena.

### 2.7. Phenomena frequency study

In order to study the frequency of the phenomena occurring in the wake of the train, a probes line has been created in the $y=0$ plane from the rear part of the train to 1 meter after at an altitude of 0.024m (same than for the TSI). The probes line contains 10 probes (1 probe each 10 cm). This is done for case number 2.

The velocity results from one of the ten probes have been treated thanks to a Fast Fourier Transform algorithm (FFT) in Matlab (the code is in annexe). FFT extracts the frequencies of a signal.

The different frequency will be used to calculate the Strouhal number $St$ defining by:

$$St = \frac{fL}{U}$$

(14)

where $f$ is the frequency of vortex shedding, $L$ the characteristic length (here the hydraulic diameter) and $U$ the velocity of the flow. Then, it will be possible to compare the Strouhal number with the Strouhal number calculated in DDES simulation.
3. Results and discussion

3.1. Mesh study

In this paragraph the results for the comparison between the coarse, medium and fine grids are shown. Three velocities plots can be seen in different planes of the computational domain: the first is perpendicular to the flow at a height of 0.024m and 1.1m behind the train. The second is parallel to the flow and in the symmetry line of the train at a height of 0.024m and the last one is parallel to the flow at 0.06m away from the center of the train on the platform and at a height of 0.1m.

The different comparisons for the different meshes are showed in figures 4, 5 and 6.

![Velocity plot comparison in the wake](image)

*Figure 2: Velocity plot comparison in the wake*
In the different figures, it can be seen that the results for the medium and the fine mesh are very similar while the results for the coarse mesh differs from the two others.

The shapes of the different curves are quite the same and only a shift with the coarse mesh can be seen. The values of the peaks are also less important for the coarse mesh. It can come from the fact that the cells are bigger in the case of the coarse mesh and so it reduces the accuracy of the results. And other explanation is that the results can move a bit between iterations due to the unsteady behavior of the wake and so the vortices which can create the shift between the curves.

For the rest of the study, the medium mesh has been kept since it gives the same results than for the fine mesh and with a lower number of cells the time of computation is shorter.
### 3.2. Drag and lift coefficient study

In this section are presented the Reynolds dependency of the drag and lift coefficients study for nine different Reynolds numbers. Also a comparison of the repartition of the drag between viscous and pressure drag between two Reynolds number is shown.

The different results obtained for the nine different simulations are presented in table 5.

<table>
<thead>
<tr>
<th>Case number</th>
<th>Lift coefficient</th>
<th>Drag coefficient</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>0.0254</td>
<td>0.4500</td>
</tr>
<tr>
<td>2</td>
<td>0.0486</td>
<td>0.4527</td>
</tr>
<tr>
<td>3</td>
<td>0.0559</td>
<td>0.4408</td>
</tr>
<tr>
<td>4</td>
<td>0.0590</td>
<td>0.4181</td>
</tr>
<tr>
<td>5</td>
<td>0.0632</td>
<td>0.4077</td>
</tr>
<tr>
<td>6</td>
<td>0.1098</td>
<td>0.4269</td>
</tr>
<tr>
<td>7</td>
<td>0.1080</td>
<td>0.4122</td>
</tr>
<tr>
<td>8</td>
<td>0.1087</td>
<td>0.4147</td>
</tr>
<tr>
<td>9</td>
<td>0.0980</td>
<td>0.3669</td>
</tr>
</tbody>
</table>

*Table 5: Lift and drag coefficients results*
Here are presented the results for all the cases in a semi-log graph:

First, it can be seen that the drag coefficient is roughly decreasing with the Reynolds number from 0.45 at $Re=40\,000$ to 0.4147 at $Re=600\,000$. Since the model is a generic train, it is difficult to compare these results with a same kind of high-speed train. However by comparing the results with the ones found in [9] and repeated in [4] or the ones in an experimental study in a wind tunnel [10], it is possible to see that the results are in the same range of drag coefficient (between 0.3 and 0.5).

The fact that some drag coefficients do not follow tendency that the drag coefficient decreases as the Reynolds number increases can come from the fact that the grid is not exactly the same. Since the size of the geometry has been increases, the size of the base size cell must be increased too. Which can leads to some differences reflected in the cell numbers in the grid, those could varied with a difference of 10 000 cells for example. It can be also the reality.
In contrary, the lift coefficient increases with the Reynolds number from 0.0254 at \( \text{Re}=40\,000 \) to 0.1 at \( \text{Re}=600\,000 \).

It is good to remark that the curve contains only 9 cases and maybe some refinement around precise Reynolds number can be interesting to adjust the tendency of the curve. Moreover, the Reynolds number for a flow around a high-speed is generally around \( 10^7 \) while here only one simulation has been done at this range of value.

These results give a first idea of the evolution of the drag and lift coefficients with the Reynolds number for this geometry.

To better understand the reason for the decrease in drag, it is relevant to know the repartition of the drag between pressure drag and viscous drag. This comparison has been done for two Reynolds number: 60 000 (case 2) and 11 100 000 (case 9). The results are showed in figure 8.

![Figure 6: Viscous and pressure drag coefficients repartition](image)

First it is obvious to see that the repartition of the drag between pressure drag and viscous drag is greatly different depending on the Reynolds number. For \( \text{Re}=60\,000 \), the pressure drag is 42% of the total drag while the viscous drag is 58% of the total drag. It is the inverse for \( \text{Re}=11\,100\,000 \), 73% of the total drag comes from the pressure drag while the viscous drag represents 27% of the total drag. The fact that
the viscous drag coefficient reduces that much seems really logical. By assuming that the Reynolds number is the ratio of the inertial forces over the viscous forces, since the Reynolds number is increasing the inertial forces become more and more predominant than the viscous forces and so on for the viscous drag. Moreover for empirical comparison using the relation known as the 1/7 Power Law from Von Karmann giving an approximation of the skin friction coefficient which is related to the viscous drag:

\[ C_f = \frac{0.0583}{R_e^{0.2}} \]  

(15)

it is obvious to see that the skin friction reduces as the Reynolds number increases.

The pressure drag coefficient remains roughly the same for the two cases since it is 0.21 for the case number 2 and 0.24 for the case number 9. It seems logical since the pressure drag is related to the geometry of the train which does not change, only the grid has changed which can provide the difference.

It is important to notice that the viscous drag coefficient can seem quite low for a high-speed train for the second case, this can come from the fact that the train is quite short (only two cars) and for a longer train, which is mostly the case for high-speed train, the viscous drag will be higher.

### 3.3. Velocity profile

The next step is to look at the flow field and in this part different velocity plot for the case number are shown as well as a comparison between RANS and DDES results.
Here the results of the velocity profile for the case 2 (Re=60 000) for an unsteady simulation are presented.

It is still clear that two vortices are visualized at the rear part of the train and create a periodic scheme along the wake. It is clearly clear on figure 10.
The different parts of slipstream can be visualized in figures 9 and 10: the pressure pulse at the very front which characterizes the flow around a train which was region 1, the boundary layer which increases along the train which was region 2, the two counter-rotating when the flow separates at the rear part of the train which was region 3 and the disturbed flow in the far wake which was region 4.

It can be observed that in the wake the velocity profile is not symmetrical which is due due to the presence of the platform. The turbulent flow is also visible thanks to characteristic chaotic phenomena. It is possible to see repetition of pattern in the wake which seems to have specific frequency which explains that it is interesting to do a FFT study in order to get the frequencies of that phenomena.

It is worth mentioning that additional velocity plots have been extracted during the course of this work in order to see that there was no strange behavior in the solution. Some parts of the train, especially the inter-car gaps, were found to cause some problems, since some spurious results where visible on velocity plots. To conclude, this part of the model requires very fine and precise grid in order to have accurate results.
Here is presented a comparison between the DDES and RANS results for the velocity plot and a Reynolds number of 60 000.
It can be clearly seen similarities between the two results on figure 11 and 12. The different regions of the slipstream are visible on both plots with the head pressure plot, the boundary layer, the two counter-rotating vortices at the rear part and the disturbed flow in the wake. For DDES the flow in the wake is also unsymmetrical which is due to the presence of the platform. The velocity plot for the DDES is really more detailed than for the RANS simulation which is the main difference between the two plots.

Here two pressure plots are shown. They correspond to a pressure plot realized for DDES and RANS in a plane perpendicular to the flow situated 0.3 m behind the train.

As for the velocity plots, the similarities between the two plots are obvious. The center of the two counter-rotating vortices are situated at the same places. However, the distinction of the vortex on the left is difficult for RANS since it is very small and not detailed. The values are approximately the same even if for RANS the maximum value is bigger than for DDES with 7.5 Pa and 5.5 Pa respectively. Also, the DDES gives again more detailed results. It has to be noticed that the contour levels are not the same for the two plots which can give some difficulty to interpret completely the two graphs.

**3.4. TSI study**

In this section the results for the slipstream study and the TSI requirement are
presented. The plots for the instantaneous velocities and the filtered velocities for different probes are shown. Also a comparison between the DDES and RANS results is done.

In the figure 15, the instantaneous velocities extracted in ground fixed probes are shown. Figure 15 shows the two pressure pulses at the front and rear parts of the train when it is passing. Then, the chaotic behavior of the flow in the wake is obvious. For each measurement, the velocity profile is changing. The peak of velocity is different for all samples and appears in the wake between 25 m and 75 m (full size) depending of the sample.

In order to calculate the $U_{2\sigma}$, the previous data must be filtered using a 1s window.
moving average filter according to TSI. In model scale this corresponds to a window at

0.004s. The filtered results are showed in figure 16:

Taking the maximum for each sample in figure 16, \( U_{2\sigma} \) can be calculated using equation (12).

After multiplication of the normalized \( U_{2\sigma} \) by 200 km/h, the \( U_{2\sigma} \) has been calculated as 8.1 m/s which is below the 15.5m/s. The train is then respecting the norm of the European Union for the aerodynamic loads on passengers on a platform.

It can be seen below a comparison between the DDES and RANS simulation for the filtered velocities in red and instantaneous velocities in black.
Figure 13: Instantaneous and filtered velocities versus reference time for RANS
The two plots seem to be very similar with higher peaks for DDES but those are less wide than for RANS. The results obtained for $U_{2\sigma}$ for DDES is 8.6 m/s which also fulfills the requirement and it is really closed to the RANS result with a difference of 6%.

### 3.5. Q-Plot

In this section a Q–factor plot for the case 2 is plotted. A comparison between two Q–plot for the DDES and RANS models is also done.

The Q–criterion visualizes vortex cores. It is plotted for the unsteady RANS computation in figure 19 which is a Q–isosurface plot for Q=500.

It can be seen on the picture the vortices in the wake of the train. They are rotating in an opposite direction which is visible with the two colors representing in red the positive values of the velocity in the $y$–direction and in blue the negative values of
the velocity in the y-direction. It is also visible that the vortices are growing with the distance from the rear part of the train.

I can be seen below a comparison between DDES and RANS for a isosurface Q-plot for $Q=125\,000$.

---

The same conclusion than for the velocity plots can be done. It is clearly seen that the flow is separated at the rear part of train in two counter-rotating vortices and the results for DDES is really more detailed than for RANS.

### 3.6. Phenomena frequency study

In this section is developed the aerodynamic phenomena frequency study. First, a FFT algorithm is applied to time series of the velocity at a probe placed in the wake
of the train. Then a Strouhal number is calculated using the main frequency of that

signal and is compared with the DDES result.

The first peak of the 0 Hz frequency has been deleted from the graphic since it represents the mean value of the signal and is not interesting. The first peak occurs at a frequency of 25 Hz, the second at 45 Hz, the third at 64.5 Hz and the fourth one at 75 Hz. Probably the fourth frequency is a harmonic of the first peak. The higher ordered peaks are the harmonics of the first frequencies.

The Strouhal number will be calculated for the first frequency since it is the most important frequency of the signal.

\[
S_t = \frac{25 \times 0.06}{15} = 0.1
\]

This value has been compared with the one calculated with DDES [11] which was 0.0855. The two values are reasonably closed, since they differ with approximately 15%. This shows that the RANS have captured the same physical flow phenomenon as the DDES.
4. Conclusion

The main part of this master thesis was to study the wake structures of a High-Speed train by simulating the flux and the turbulence via a CFD software StarCCM+. This model has been applied to a generic train model scaled 1:50 called Aerodynamic Train Model (ATM). The method used was the Averaged Navier–Stokes equation model coupling with a $k-\omega$ SST model for the turbulence.

The aim goal of the master thesis was to study the slipstream which is an aerodynamic phenomenon caused by the viscous air which is dragged when a train is passing and which can destabilize people or objects on a platform near the train. To avoid this problem, the train must fulfill some requirement such as the Technical Specification of Interoperability which maintains that “A full length train, running in the open air at a reference speed $U=200\text{km/h}$ shall not cause the air speed to exceed value $U_{2\sigma} =15.5\text{m/s}$ at a height of 1.2m above the platform and at a distance of 3.0m from the track centre, during the whole train passage (including the wake)”.

The first part of the thesis consisted in getting a reliable grid which has been done by comparing a coarse, medium and fine meshes using a steady turbulent flow. The results of the mesh comparison show a good dependence between the three grids and the medium mesh has been kept for the rest of the study since the resolution was enough for the simulation.

Different studies have been realized with this mesh and some of them have been compared to previous DDES study on the same model. The result of the TSI study is that the $U_{2\sigma}$ of the ATM is 8.1 m/s which is below the limit and so the ATM fulfills this requirement. The result of the DDES study gave 8.6 m/s which is then very closed to the result of the RANS simulation.

One of the other studies is a drag and lift coefficient analyze. The flow has been computed for different Reynolds numbers by changing the size and/or the velocity in order to see the difference for the drag and lift coefficients depending on this number.
It has been shown that the drag coefficient is roughly decreasing as the Reynolds number increases while it is the contrary for the lift coefficient, which is due to a decrease in friction drag.

The velocity profile for a Reynolds number of 60 000 has been plotted and compared with the plot of the DDES study. On this plot, it is seen the head pressure pulse at the front part of the train as well as a turbulent flow in the wake with a repetitive pattern. The wake is also non-symmetric due to the presence of the platform. The comparison with the DDES plot shows a good similarity in the general behavior of the wake even though the DDES plot is much more precise in the details of that wake.

The Q-factor, which is a good quantity to visualize the vortices, showed the presence of two counter-rotating vortices in the wake of the train which is the same conclusion by looking at the result of the DDES study even though the DDES plot contains smaller turbulent scales.

Finally, a Fast Fourier Transform algorithm has been applied to instantaneous velocity results in the wake of the train in order to get the frequency of the aerodynamic phenomena in that wake. The main frequency is 25 Hz and corresponds to a Strouhal number of 0.1, quite closed to the results obtained with DDES which is 0.085.

To summarize, the results obtained agree well with the ones of the DDES study even though those results are more precise. The grid for RANS contains less cell : 8 500 000 cells for 20 000 000 for the DDES. A difference which is quite important and can reduce greatly the computation time event though the RANS does not contain the small scales resolved by the DDES. Depending on the accuracy wanted, the RANS remain a good tool for this model.

More works can be done on this subject. Effectively, it could be interesting to do TSI requirement for people working near the tracks. It can be interesting to complete the drag and lift coefficients study notably around the local minimum value around Re=100 000. It can be also interesting to get a grid with less cell by searching a more optimized solution.
5. References


Decomposition – 2012


[8] Starccm+ V.6 Help documents


6. Appendix

Matlab program 1 : FFT

dir=’/home/caa/fguillou/Documents/MATLAB/Wake_probe/’;
k=1;
A(:,k)=csvread(’/home/caa/fguillou/Documents/MATLAB/Wake_probe/geo0.csv’,1,0,[1 0 20 0]);
for i=300:769
    fname=[dir ’point0.’ num2str(i) ’.csv’];
k=k+1;
    A(:,k)=csvread(fname,1,1,[1 1 20 1]);
end
B=sortrows(A,1);
g1=B(3,2:471);
g1=transpose(g1);
T=0.0013;
Fs=1/T;
L=470;
t=(0:L-1)*T;
NFFT = 2^nextpow2(L); % Next power of 2 from length of y
Y = fft(g1,NFFT)/L;
f = Fs/2*linspace(0,1,NFFT/2+1);
% Plot single-sided amplitude spectrum.
set(gca,'FontSize',15)
plot(f(13:257),2*abs(Y(13:NFFT/2+1)))
title('Single-Sided Amplitude Spectrum of y(t)')
xlabel('Frequency (Hz)')
ylabel('|Y(\theta)|')

Matlab program 2 : TSI study

dir='/scratch/fguillou/StarCCM/Thesis/EssaiD_3/Velocities/';
k=1;
A(:,k)=csvread('/scratch/fguillou/StarCCM/Thesis/EssaiD_3/Velocities/geo0.csv',1,0,[1 0 501 0]);
point=1;
ziter=501-point;
times=[50 75 100 125 150 175 200 225 250 275 300 325 350 375 400 425 450 475 500 525 550 575
600 625 650 675 700 725 750 775 800 825 850 875 900];
for i=0:1499
    fname=[dir 'step0.' num2str(i) '.csv'];
    k=k+1;
    A(:,k)=csvread(fname,1,1,[1 1 501 1]);
end
B=sortrows(A,1);
for s=1:35
    for z=0:ziter
        C(z+1,s)=B(point+z,z+times(s));
    end
end
for l=1:35
    C(:,l)=smooth(C(:,l),100);
end
C=(15-C)/15;
for l=1:35
    Max(l)=max(C(:,l));
end
dev=std(Max);
average=mean(Max);
(average+2*dev)*200/3.6
x=0:0.01:5;
set(gca,'FontSize',15)
plot(x,C)
xlabel('x (m)')
ylabel('1-U/Uinf')

Matlab program 3: Cl and Cd plot

X=[40000 60000 80000 100000 120000 240000 400000 600000 1.11E7];
CD=[0.45 0.4527 0.4408 0.4181 0.4077 0.4269 0.4122 0.4147 0.3669];
CL=[0.0254 0.0486 0.0559 0.059 0.0632 0.1098 0.108 0.1087 0.098];
figure(1)
subplot(2,1,1)
set(gca,'FontSize',15)
semilogx(X,CD,'b+-')
xlabel('Re')
ylabel('Drag coefficient')

subplot(2,1,2)
set(gca,'FontSize',15)
semilogx(X,CL,'b+-')
xlabel('Re')
ylabel('')

X=[40000 60000 80000 100000 120000 240000 400000 600000];
CD=[0.45 0.4527 0.4408 0.4181 0.4077 0.4269 0.4122 0.4147];
CL=[0.0254 0.0486 0.0559 0.059 0.0632 0.1098 0.108 0.1087];

figure(2)

subplot(2,1,1)
set(gca,'FontSize',15)
plot(X,CD,'b+-')
plot(X,CD,'b+-')
xlabel('Re')

ylabel('Drag coefficient')

subplot(2,1,2)

set(gca,'FontSize',15)

plot(X,CL,'b+-')

xlabel('Re')

ylabel('Lift coefficient')