UNTVERSITY OF NAVARRA
SCHOOL OF ENGINEERING
SAN SEBASTIÁN

TECNUN
Universidad de Navarra

STUDY OF CROSSWIND EFFECTS ON THE DYNAMICS AND
AERODYNAMICS OF HIGH-SPEED TRAINS

DISSERTATION
submitted for the Degree of Doctor of Philosophy of the University of
Navarra by

IGNACIO SESMA GOTOR

under the supervision of
Dr. Jordi Viñolas Prat and
Dr. Gorka Sánchez Larraona

San Sebastián, September 2013
A mi madre y mi hermana,
(. .) Because the people who are crazy enough to think they can change the world, are the ones who do

Apple commercial
Acabar esta tesis supone para mí el fin del trayecto. Familiares allegados saben que el haber podido dedicar unos años de mi recién estrenada vida profesional al mundo ferroviario supone haber cumplido un objetivo que tenía cuando me sentaba en un pupitre del colegio. Es la hora de hacer trasbordo y cambiar de medio de transporte, pero no sin antes agradecer a quienes me habéis ayudado a alcanzar la estación de destino en las mejores condiciones.

Me gustaría agradecer en primer lugar a Tecnun-Universidad de Navarra y a CEIT por haberme ofrecido la posibilidad de realizar esta tesis doctoral.

Mi agradecimiento a mis directores de tesis. Mi sincero agradecimiento a Jordi por haber apostado por esta tesis cuando la situación invitaba a lo contrario. Gracias por habermme transmitido tus conocimientos y habermme dado lecciones más importantes que las técnicas, las cuales estoy seguro me serán de gran ayuda en el futuro. Gorka, gracias por tu dedicación y por habermme instruido en el mundo de la aerodinámica. Sin tu ayuda la mitad de este libro probablemente nunca hubiera sido escrito. Me quiero acordar también de Alejandro Rivas por abrirme las puertas del Departamento de Fluidos y de Germán Giménez, siempre disponible para ayudar, un ejemplo en lo profesional y en lo personal.

Gracias a mis compañeros del laboratorio de ferrocarriles, a mi querido amigo de nacionalidad chilena Rodrigo. A Iñaki y Aina con quien compartí tantos días de trabajo en el cubículo del laboratorio. A Miguel por estrenar su PhD corrigiendo esta tesis. A Albi quien me enseñó a poner un tren sobre las vías en SIMPACK por primera vez. A Javi e Imanol porque además de ser grandes profesionales, con su personalidad hacen un equipo del Departamento de Mecánica de CEIT. Gracias también a Manolo por solucionar siempre con amabilidad mis problemas con Windows y su caprichoso modo de funcionar. También me quiero acordar de Nere, Luis, Edu, Jorge Juan, Xabi y Pablo porque cada uno de forma diferente me habéis ayudado durante este tiempo.

Mi más profundo agradecimiento a Wendy Baldwin. Tu ayuda y tu personal dedicación han conseguido que maneje el inglés con cierta soltura.

Mi especial gratitud a Diego y Miguel por estos cuatro años en los que hemos hecho de un piso en Garibay, nuestra casa. Gracias por vuestra comprensión después de los tantos días difíciles que hemos pasado. Gracias por vuestra amistad y por ayudarme a ser mejor persona.
Y mi familia. Se dice que cuando alguien empieza una tesis, lo hace el interesado y su familia. Echando la vista atrás, el dicho guarda bastante verdad. Mi agradecimiento a mis padres. En particular, siempre estaré en deuda con mi madre y mi hermana por el apoyo durante estos años. Gracias por mejorar y soportar mi a veces insoportable carácter. Gracias por haberme dejado equivocar aun cuando vuestros consejos decían lo contrario. Sobre todo, gracias a mi madre por el esfuerzo realizado para que pudiera estudiar en la Universidad de Navarra. Disfrutad de este éxito porque es tanto vuestro, como mío. Por último, mi agradecimiento a mi tía adoptiva, Cristina, con quien he compartido tantas alegrías.
# Table of Contents

AGRADECIMIENTOS I

TABLE OF CONTENTS III

LIST OF FIGURES VII

LIST OF TABLES XV

RESUMEN XVII

ABSTRACT XIX

CHAPTER 1: INTRODUCTION 1

1.1 What makes crosswinds over high-speed trains a new object of study 2

1.2 A multidisciplinary subject 4

1.3 Defining ‘safe operation’ 6

1.4 Currently open questions on the design of fences to protect high-speed trains 8

1.5 State of the art 10

1.5.1 Aerodynamics 11

1.5.2 Dynamics 15

1.6 Aim of the thesis 18

1.7 Structure of the thesis 19

CHAPTER 2: CURRENT REGULATION 21

2.1 European Standard EN 14067-6:2010 22

2.1.1 Determination of aerodynamic coefficients 22

2.1.2 The wind gust model: Chinese Hat wind scenario 25

2.1.3 Simplified models 29

2.2 Ground and track configurations 35

2.2.1 Single track and ballast rail 36

2.2.2 Flat ground with gap 36

2.2.3 Double track ballast and rails 37

2.2.4 Standard embankment of a 6 m height 37

2.3 Other regulations to guarantee safety 39

2.3.1 A comparison of some important requirements 40

2.3.2 Other approaches to the embankment scenario 42
## Table of Contents

### CHAPTER 3: VEHICLE MODELS 47

3.1 Multibody models to study vehicle dynamics 47  
   3.1.1 Full multibody model 48  
   3.1.2 Simplified two-dimensional model 50  

3.2 Models to study vehicle aerodynamics 54  
   3.2.1 A 2D model. Computational model and mock-up of the vehicle. 55  
   3.2.2 3D models 63

### CHAPTER 4: VEHICLE DYNAMICS IN A CROSSWIND CONTEXT 75

4.1 Standard EN 14067-6:2010 76  

4.2 Comparison between vehicle models: response to wind loads 77  

4.3 Guaranteeing safety with a 2D model 80  
   4.3.1 Wheel unloading calculation. Restrictions of the 2D model 81  
   4.3.2 Calculating CWCs with both models 85  
   4.3.3 Calculating CWCs. A parametric study 86  

4.4 Vehicles circulating on a curved track 88  
   4.4.1 Influence of some suspension components on safety and comfort 95  

4.5 Conclusions 103

### CHAPTER 5: CFD APPROACHES TO STUDY AERODYNAMICS 105

5.1 A eulerian approach: URANS with FVM 106  
   5.1.1 Turbulence modelling: URANS 107  
   5.1.2 The Finite Volume Method 113  

5.2 A lagrangian approach: LES with LBM 116  
   5.2.1 A previous step prior to LBM: Lattice Gas Automata schemes 117  
   5.2.2 The Lattice Boltzmann method 119  
   5.2.3 Turbulence modelling in LBM 121

5.3 Validation of a CFD code based on LBM to simulate crosswinds over trains 124  
   5.3.1 Flat ground with 235 mm gap 126  
   5.3.2 Double track ballast and rail 133  
   5.3.3 A comparison of wheel unloading calculations 139

5.4 Conclusions 141

### CHAPTER 6: EFFECT OF FENCES ON VEHICLE AERODYNAMICS 143

6.1 A 2D study of the effects of fences on a vehicle standing on a bridge 143  
   6.1.1 Effects on vehicle coefficients and flow 145  
   6.1.2 Reynolds dependency of the aerodynamic coefficients 152
<table>
<thead>
<tr>
<th>6.2</th>
<th>A brief assessment of fences in the 3D space</th>
<th>156</th>
</tr>
</thead>
<tbody>
<tr>
<td>6.2.1</td>
<td>Conclusions</td>
<td>163</td>
</tr>
<tr>
<td></td>
<td>CHAPTER 7: CONCLUSIONS AND FUTURE WORK</td>
<td>165</td>
</tr>
<tr>
<td>7.1</td>
<td>Conclusions</td>
<td>165</td>
</tr>
<tr>
<td>7.2</td>
<td>Future work</td>
<td>166</td>
</tr>
<tr>
<td></td>
<td>APPENDIX A: PUBLICATIONS</td>
<td>169</td>
</tr>
<tr>
<td></td>
<td>BIBLIOGRAPHY</td>
<td>171</td>
</tr>
</tbody>
</table>
FIGURE 1: Weight and speed trends of Shinkansen vehicles (up) and ICE vehicles (down). Weight in the Shinkansen family is measured with empty vehicles whereas in the ICE family weight is measured with fully loaded vehicles.

FIGURE 2: Flow chart of a crosswind computation that would solve both dynamics and aerodynamics together.

FIGURE 3: Schema of a CWC defining the stable and instable areas of operation. $v_w$ stands for wind speed and $v_v$ stands for vehicle speed.


FIGURE 5: Parapet installed at the Uetsu line in Japan (right). Parapet installed in TGV Méditerranée line at the Viaduc des Angles in France (left).

FIGURE 6: Aerodynamic coefficients of ICE 3 standing on flat ground with 235 gap.

FIGURE 7: Isosurface of the instantaneous static pressure with value -0.4 Pa showing the instantaneous wake structures taken from (Hemida and Krajnovic, 2010). Case of yaw angle 90 degrees (top) and case of yaw angle 35 degrees (bottom).

FIGURE 8: Wind distribution example as given in EN 14067-6:2010.

FIGURE 9: Coordinate system for the derivation of the wind model. $X,Y$ is the inertial coordinate system for the definition of the main flow direction $w$. $\beta_w$ is the angle between wind and track. $u_{turb}$ are the turbulent disturbances parallel to the main flow direction. $v_w$ are the turbulent disturbances perpendicular to the main flow direction.

FIGURE 10: Definition of the three-mass model according to EN 14067-6:2010.

FIGURE 11: The five-mass model according to EN 14067-6:2010. The bodies are $WS_i$ (wheelset), $BG_i$ (bogie) and $CB$ (carbody). The stiffnesses are $C_{pij}$ (primary suspension), $C_{sij}$ (secondary suspension) and $ARB_i$ (anti-roll bar) where the subindex $i$ stands for position of the element and $j$ stands for the side of the vehicle and. Finally, $Q_{ij}$ are the vertical forces at the contact.

FIGURE 12: Placement of the contact point in the five-mass model.

FIGURE 13: Single track and ballast scenario.

FIGURE 14: Configuration of flat ground with gap of 235 mm.

FIGURE 15: Configuration of double ballast and rail. Dimensions are given in millimetres.
FIGURE 16: Configuration of a 6 m standard embankment. Dimensions are given in millimetres .............................................................. 38

FIGURE 17: Components of wind near and on top of the embankment .................. 39

FIGURE 18: Multibody model of a generic high-speed train ................................ 49

FIGURE 19: Stiffness characteristic of the bumpstops ........................................ 49

FIGURE 20: Schematic representation of the two-dimensional model. Description of the masses, suspension elements, Q-forces and aerodynamic loads ....................... 51

FIGURE 21: Comparison of the sway test of both models .................................... 54

FIGURE 22: Relative error of the sway test .......................................................... 54

FIGURE 23: Cross-section of the vehicle standing on a bridge along with the dimensions of the vehicle (H stands for height and W for width) and the bridge. The parameters of the fences, height (hf) and eave length (le) are also given. The wind loads are drag force (D), lift force (L) and roll moment (Mv). The moment around the leeward rail is M .................. 56

FIGURE 24: Non-scaled representation of the computational domain. Dimensions are in terms of S that is the height of the vehicle plus the bridge. Boundary conditions are specified in grey color ................................................................. 57

FIGURE 25: Mock-up of the vehicle (up) and mock-up of the vehicle standing on the bridge without fences (down) ..................................................... 60

FIGURE 26: Comparison between the wind tunnel tests (WT) and the computational results (CFD) for the given cases ............................................................... 63

FIGURE 27: Shape of the ICE 3 model available at EN 14067-6:2010 and detail of the bogie ............................................................ 65

FIGURE 28: Example of a lattice structure with an octree of 8 levels ....................... 67

FIGURE 29: Comparison of the flow at the leeward side of the vehicle computed in terms of the Enhanced wall function (right) and the Non-equilibrium wall function (left). Yaw angle: 30 degrees .................................................. 68

FIGURE 30: Non-scaled representation of the CFD domain, front view (up) and top view (down). Dimensions are given in terms of the length (Lw = 40 m) and the height (H = 4 m) of the train model. The boundary conditions are in grey for the top and the laterals of the domain. The boundary conditions of the ground is in black ................. 70

FIGURE 31: Non-scaled representation of the CFD domain, front view (up) and top view (down). Dimensions are given in terms of the length (Lw = 40 m) and the height (H = 4 m) of the train model. The boundary conditions are in grey for the top and the laterals of the domain. The boundary conditions of the ground are in black ............ 72
FIGURE 32: Definition of the lattice sizes of both the vehicle and the track model... 73
FIGURE 33: Lateral deformation of the bumpstop. Comparison between the 2D model and the multibody model............................................................. 77
FIGURE 34: Rotation angle of the torsion bar. Comparison between the 2D model and the multibody model............................................................. 78
FIGURE 35: Vertical displacement of the primary suspension. Comparison between the 2D model and the multibody model............................................................. 79
FIGURE 36: Lateral displacement of carbody’s CoG. Comparison between the 2D model and the multibody model.......................... 80
FIGURE 37: Wheel unloading comparison between the full model and the simplified model ........................................................................................................ 81
FIGURE 38: Schema of the contact points location and Q-forces calculation, for onepoint contact and multipoint contact ................................................. 83
FIGURE 39: Comparison of CWC using the simplified and the multibody model and two different load cases............................................................. 86
FIGURE 40: Difference in results of CWCs when cases 1 to 4 are compared to the Reference case (see TABLE 18)............................................................. 87
FIGURE 41: Example of one wind scenario and track cant used for the simulations in curved track............................................................. 89
FIGURE 42: Wheel unloading ratio of the vehicle running on a curved track under different crosswind conditions ............................................................. 90
FIGURE 43: Effect of the wind velocity on the non-compensated accelerations measured at the center of the carbody............................................................. 91
FIGURE 44: Effect of the wind velocity on the non-compensated accelerations measured at the front-end of the carbody............................................................. 92
FIGURE 45: Displacement of the contact point on the tread under different wind load conditions. Measures were taken on the left wheel of wheelset one ........................................................................................................ 93
FIGURE 46: Measures of the flange contact on the left wheel of wheelset one under different crosswind conditions ............................................................. 93
FIGURE 47: Curved track. Lateral deformation of the bumpstop for the given crosswind velocities ........................................................................................................ 94
FIGURE 48: Stiffness characteristic of the bumpstop under three hypotheses. Case of standard configuration (blocked at 60 mm), case of the minimum allowed lateral deformation (blocked at 45 mm) and the case of the maximum allowed lateral deformation (blocked at 75 mm) ........................................................................................................ 96
FIGURE 49: Variation of the maximum value of the wheel unloading ratio as function of the maximum lateral deformation of the bumpstop .................................. 97
FIGURE 50: Increase of the lateral displacement of the carbody’s CoG as function of the maximum lateral deformation of the bumpstop ................................................ 98
FIGURE 51: Influence of the lateral deformation of the bumpstop on the non-compensated acceleration measured on the front of the carbody. Curved track ....... 99
FIGURE 52: Influence of the anti-yaw damper on the non-compensated acceleration measured on the leading bogie. Curved track ............................................. 100
FIGURE 53: Wheel unloading increase as function of the anti-roll stiffness .......... 101
FIGURE 54: Increase of the lateral displacement of the carbody’s COG as function of the anti-roll stiffness .................................................................................. 101
FIGURE 55: Values of the non-compensated accelerations as function of the anti-roll bar’s stiffness .................................................................................. 102
FIGURE 56: Influence of the anti-roll stiffness on the non-compensated acceleration measured at the center of the carbody. Curved track ........................................ 103
FIGURE 57: Schematic representation of the energy cascade and the approach to solve fluid dynamics of the different CFD models ................................................. 106
FIGURE 58: Example of a decomposition of a generic flow variable, φ .................. 108
FIGURE 59: Control volume and notation for discretization. Cell based approach in an unstructured mesh .......................................................................................... 114
FIGURE 60: Simplest model of LGA lattice with d = 2 and b = 4 .......................... 117
FIGURE 61: Interpolating functions for the Non-Equilibrium wall functions (Shih et al., 1999). ............................................................................................................. 124
FIGURE 62: Cutting planes to show the flow pattern ......................................... 126
FIGURE 63: Comparison of the aerodynamic coefficients of a ICE 3 between the results from wind tunnel tests provided by the standard and Schober et al, 2010, and the computed coefficients with CFD. Ground scenario: flat ground with 235 mm gap ... 127
FIGURE 64: Comparison of the $C_{mx,lee}$ and the error of the coefficients from FIGURE 63 between the results from wind tunnel tests provided by the standard and Schober et al, 2010, and the computed coefficients with CFD. Ground scenario: flat ground with 235 mm gap ........................................................................ 128
FIGURE 65: Three-dimensional view of a iso-surface of the mean pressure field colored with values of the vorticity [$s^{-1}$] for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap ................................................. 129
FIGURE 66: Contours of the mean pressure field [Pa] at the positions showed in FIGURE 62 for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap ................................................................. 130

FIGURE 67: Contours of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap ................................................................. 131

FIGURE 68: Vectors of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap ................................................................. 131

FIGURE 69: Windward, leeward and front views of the contours of the mean pressure field [Pa] at the vehicle’s surface for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap ................................................................. 132

FIGURE 70: Instant value of the pressure at the vehicle surface calculated with XFlow v. 89.03 (left) and with XFlow version 90.00 (right) ................................................................. 133

FIGURE 71: Comparison of the aerodynamic coefficients of a ICE 3 between the results from wind tunnel tests provided by the standard and the computed coefficients with CFD. Ground scenario: double track ballast and rail ................................................................. 135

FIGURE 72: Comparison of the $C_{mx,lee}$ and the error of the coefficients from FIGURE 70 between the results from wind tunnel tests provided by the standard and the computed coefficients with CFD. Ground scenario: double track ballast and rail .... 135

FIGURE 73: Three-dimensional view of a iso-surface of the mean pressure field colored with values of the vorticity [s$^{-1}$] for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail ................................................................. 136

FIGURE 74: Contours of the mean pressure field [Pa] at the positions showed in FIGURE 62 for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail ................................................................. 137

FIGURE 75: Contours of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail ................................................................. 138

FIGURE 76: Vectors of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail ................................................................. 138

FIGURE 77: Windward, leeward and front views of the contours of the mean pressure field [Pa] at the vehicle’s surface for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail ................................................................. 139
FIGURE 78: Comparison of the wheel unloading ratio (up) and its epsilon (down) using both the experimental and the computed coefficients in the flat ground with gap scenario .................................................. 140

FIGURE 79: Comparison of the wheel unloading ratio (up) and its epsilon (down) using both the experimental and the computed coefficients in the double track ballast and rail scenario .......................................................................................... 141

FIGURE 80: Time signals of the calculated coefficients for the case of fence $h_f = 2250$ mm with eave 750 mm (up) and a panoramic view of the scenario showing the turbulent viscosity (kg/m·s) (down). Calculations carried out at $Re = 12 \times 10^6$ ........ 146

FIGURE 81: Variation of the time period of the coefficients time signals in terms of percentage by taking as reference the ‘No Fence’ scenario. Calculations carried out at $Re = 12 \times 10^6$ ........................................................................................................ 147

FIGURE 82: Two-dimensional aerodynamic coefficients of the vehicle calculated at $Re = 12 \times 10^6$ .............................................................................................................................................. 148

FIGURE 83: Streamlines coloured with velocity in meters per second when $Re = 12 \times 10^6$. The scenario has a fence without eave and a height of 2250 mm .................. 149

FIGURE 84: Contours of mean pressure [Pa] (up) and mean velocity [m/s] (down) when $Re = 12 \times 10^6$. Three scenarios are compared: no fence (left), $h_f = 1250$ mm (center), $h_f = 2750$ mm (right) ........................................................................................................ 150

FIGURE 85: Contours of mean pressure [Pa] (up) and mean velocity [m/s] (down) when $Re = 12 \times 10^6$. Three scenarios are compared of the fence $h_f = 2250$ mm. No eave (left), $l_e = 500$ mm (center), $l_e = 750$ mm (right) ......................................................................................... 151

FIGURE 86: Two-dimensional aerodynamic coefficient of moment around the leeward rail of the vehicle, $C_{mv}$, calculated at $Re = 12 \times 10^6$ ........................................................................................................ 152

FIGURE 87: Variation of the time period of the coefficients time signals in terms of percentage by taking as reference the ‘No Fence’ scenario. Calculations carried out at $Re = 1 \times 10^5$ ........................................................................................................ 153

FIGURE 88: Strouhal number calculated in all the scenarios and for the two Re ...... 154

FIGURE 89: Reynolds dependency of the aerodynamic coefficients $C_d$, $C_l$, $C_{mv}$. Difference between tests at $Re = 1 \times 10^5$ and $Re = 12 \times 10^6$ .................................................. 155

FIGURE 90: Mean velocity field and discretization of the domain in the surroundings of the fence $hf = 1750$ mm (right) and $hf = 2750$ mm (left) .................................................................................. 156

FIGURE 91: Aerodynamic coefficients of the unprotected vehicle against the cases of vehicle sheltered with fences ............................................................................................... 157

FIGURE 92: Contours of mean pressure on the vehicle surface [Pa]. Three scenarios are compared: No Fence (up), $h_f = 1750$ mm (center) and $h_f = 2750$ mm (down) .... 158
FIGURE 93: Effect of fences on the $C_{Mx,lee}$ coefficient ......................................................... 159

FIGURE 94: Comparison of the moment around the leeward rail computed with the simplified model (gray color) and the 3D model (red color) .............................................. 160

FIGURE 95: Contours of mean pressure [Pa] at the positions showed in FIGURE 62. Three scenarios are compared: No Fence (up), $h_f = 1750$ mm (center) and $h_f = 2750$ mm (down) ......................................................................................................................... 161

FIGURE 96: Contours of mean velocity [m/s] at the positions showed in FIGURE 62. Three scenarios are compared: No Fence (up), $h_f = 1750$ mm (center) and $h_f = 2750$ mm (down) ........................................................................................................................................ 162

FIGURE 97: Velocity vectors [m/s] at the positions showed in FIGURE 62. Three scenarios are compared: No Fence (up), $h_f = 1750$ mm (center) and $h_f = 2750$ mm (down) ........................................................................................................................................ 163
### list of tables

<table>
<thead>
<tr>
<th>Table Number</th>
<th>Table Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Functions for a temporal wind distribution</td>
<td>25</td>
</tr>
<tr>
<td>2</td>
<td>Parameters of the Chinese Hat that are fixed by EN 14067-6:2010</td>
<td>26</td>
</tr>
<tr>
<td>3</td>
<td>Method factor $f_m$ for UIC track gauge for different types of vehicles</td>
<td>31</td>
</tr>
<tr>
<td>4</td>
<td>Results from the calculation of $f_{emb}$ following the Eurocode steps for</td>
<td>45</td>
</tr>
<tr>
<td></td>
<td>embankments of different heights and slope 0.67 taken from Herb et al., 2007</td>
<td></td>
</tr>
<tr>
<td>5</td>
<td>Mass and inertial parameters of the multibody model</td>
<td>48</td>
</tr>
<tr>
<td>6</td>
<td>Stiffness characteristic of the bumpstop. Values of positive terms</td>
<td>49</td>
</tr>
<tr>
<td>7</td>
<td>Parameters of the vehicle's suspension of the multibody model. $k$, $k_x$ and</td>
<td>50</td>
</tr>
<tr>
<td></td>
<td>$k_z$ are stiffness coefficients. $c_x$ is a damping coefficient</td>
<td></td>
</tr>
<tr>
<td>8</td>
<td>Masses, inertias and coordinates of the CoG used in the simplified model</td>
<td>51</td>
</tr>
<tr>
<td>9</td>
<td>Suspension parameters of the two-dimensional model</td>
<td>52</td>
</tr>
<tr>
<td>10</td>
<td>Coordinates of the suspension elements of the simplified model</td>
<td>53</td>
</tr>
<tr>
<td>11</td>
<td>Comparison of the vibration modes between the models of the vehicle ...</td>
<td>53</td>
</tr>
<tr>
<td>12</td>
<td>Lattice sizes for the configuration of the ground scenarios used to validate</td>
<td>71</td>
</tr>
<tr>
<td></td>
<td>the code based on the Lattice-Boltzmann Method</td>
<td></td>
</tr>
<tr>
<td>13</td>
<td>Lattice sizes for the configuration of the simulations to assess the</td>
<td>73</td>
</tr>
<tr>
<td></td>
<td>sheltering effect of fences in the 3D space</td>
<td></td>
</tr>
<tr>
<td>14</td>
<td>Calculation of the wheel unloading factor through different methods and</td>
<td>82</td>
</tr>
<tr>
<td></td>
<td>with both models.</td>
<td></td>
</tr>
<tr>
<td>15</td>
<td>Comparison of the Q-forces at the contact calculated with onepoint and</td>
<td>84</td>
</tr>
<tr>
<td></td>
<td>multipoint</td>
<td></td>
</tr>
<tr>
<td>16</td>
<td>Comparison of the displacement of the contact point calculated with</td>
<td>84</td>
</tr>
<tr>
<td></td>
<td>onepoint and multipoint</td>
<td></td>
</tr>
<tr>
<td>17</td>
<td>Comparison of the wheel unloading including contact types and number of</td>
<td>85</td>
</tr>
<tr>
<td></td>
<td>contact points. For multipoint, it shows the difference between the number</td>
<td></td>
</tr>
<tr>
<td></td>
<td>of areas from which the ration is calculated</td>
<td></td>
</tr>
<tr>
<td>18</td>
<td>Options considered for comparison to the 'Reference case' in CWC calculations.</td>
<td>86</td>
</tr>
<tr>
<td></td>
<td>Results appear in FIGURE 40</td>
<td></td>
</tr>
<tr>
<td>19</td>
<td>Track layout of the curved track</td>
<td>88</td>
</tr>
</tbody>
</table>
La investigación del efecto del viento lateral sobre la circulación de los vehículos ferroviarios comenzó a ser reseñable durante los años 90 con el desarrollo de los trenes de alta velocidad. El proyecto europeo TRANSERAERO (Schulte-Werning et al., 2002) supuso el punto de partida, ya que consiguió captar la atención de la comunidad científica y los fabricantes de material rodante. El viento lateral puede generar un problema de seguridad y es también uno de los factores clave que pueden impedir el continuo incremento de velocidad demandado al transporte ferroviario.

El estudio del efecto del viento lateral requiere de una visión global debido a que se debe analizar tanto la dinámica como la aerodinámica del vehículo. Es un problema acoplado que lo convierte en multidisciplinar, y por tanto en la tesis se aborda el problema desde las dos perspectivas. El bloque centrado en la dinámica del vehículo estudia el alcance de los modelos bidimensionales para extraer conclusiones en relación con la seguridad y el confort del vehículo. La descarga de rueda es el factor empleado por las normas en el contexto del viento cruzado para garantizar la seguridad del vehículo. Las conclusiones muestran que un modelo simplificado es capaz de proporcionar factores de descarga de rueda con pequeños márgenes de error. Además, como el viento lateral puede resultar un problema también desde el punto de vista del confort, mediante modelos multibody completos del vehículo se comprobó que la circulación en curva con viento lateral resulta ser un escenario con unas condiciones de circulación particularmente adversas. Dada una velocidad de viento lo suficientemente elevada, la aparición de un marcado movimiento de lazo puede obligar a reducir la velocidad del vehículo si se quiere mantener la calidad de la marcha.

En la tesis se detallan trabajos previos del campo de la aerodinámica en los que se estudia detenidamente el flujo tridimensional en torno a los vehículos de alta velocidad en un escenario de viento lateral. El efecto de la infraestructura es, sin embargo, importante y cualquier modificación en torno al vehículo altera el flujo. A este respecto, en esta tesis se estudia el efecto de los parapetos protectores (barreras) y cómo modifican las cargas del viento sobre el vehículo. Se han empleado primero simulaciones 2D para llevar a cabo un estudio paramétrico del diseño de la valla protectora, y con el fin de verificar la utilidad de modelos simplificados. El presente trabajo muestra que los modelos 2D son válidos para encontrar un grupo de diseños con los que simular escenarios 3D en detalle.

En relación con el análisis del efecto protector de las barreras en 3D, el objetivo fue la obtención de los coeficientes aerodinámicos del vehículo utilizando algunos diseños de las vallas utilizadas en el estudio bidimensional. La propuesta que aquí se recoge
consiste en emplear un código CFD (Computational Fluid Dynamics) que no requiere
mallado y está basado en el Lattice-Boltzmann Method (LBM). La validación realizada
en el contexto de esta tesis refleja que este tipo de códigos proporcionarán resultados
con un error razonable en el rango de ángulos de ataque de interés.
abstract

The research on the crosswind field on the circulation of railway vehicles became noteworthy in the 90s with the development of high-speed trains. The European project TRANSERA (Schulte-Werning et al., 2002) was the starting point since it was able to attract the attention of both the scientific community and the rolling stock manufacturers. Crosswinds are a safety matter and one of the key factors that are able to stop the continuous speed increase requested to the rail transportation.

The study of the effect of crosswinds on vehicles problem requires a global vision since it involves both the dynamics and the aerodynamics of the vehicle. It is a coupled problem that requires a multidisciplinary approach and therefore the thesis addresses the problem from both perspectives. The sections centered on the vehicle dynamics firstly study 2D models and their benefit to extract conclusions in regard to the vehicle safety and comfort. The wheel unloading ratio is the factor used by standards to guarantee the safeness of the vehicle in the crosswind context. Conclusions show that a simplified model is able to provide wheel unloading factors with small rates of error. Moreover, crosswinds can also become a problem from the point of view of the vehicle's comfort. By using full multibody models, we checked that the case of circulation on a curved track under lateral wind sets especially adverse conditions that can constraint the vehicle to reduce the travel speed in cases of high wind speeds. We observed that the vehicle described a yawing movement that made it to become instable, which reduced significantly the ride quality.

The thesis describes previous aerodynamic research on the three-dimensional flow around high-speed trains in a crosswind scenario. However, the effect of the infrastructure is important and any modification at the surroundings of the vehicle alters the flow. At this respect, the thesis studies the sheltering effect of wind fences. Firstly, 2D computations have been used to carry out a parametric study of the fence design and to validate the value of simplified models. This work shows that 2D models are valid to find a set of fence designs to carry more detailed simulations in the 3D space.

The goal of measuring the sheltering effect of wind-breaking devices in the 3D space was to obtain the aerodynamic coefficients of the vehicle using some of the fence designs that were used in the 2D model. The proposal herein presented is based on the use of a meshless CFD (Computational Fluid Dynamics) code that is based on the Lattice-Boltzmann Method (LBM). Results prove that these codes provide results with a reasonable margin of error at the range of yaw angles of interest for this investigation.
Chapter 1

The effect of crosswinds over trains has become an important topic and relevant to the scientific community because it is mainly a safety matter. Vehicles speed has increased in the last years and the trend of designers has been to lighten vehicles. The goal of increasing velocity is to make rail transportation competitive against other means of transport; therefore, it is vital to face the problem in order to guarantee the future of this mean of transportation. Weight reduction methods to reduce the dead-weight per passenger are solutions of special interest, i.e. by developing lighter rolling stock. In addition, high-speed lines are designed with large number of bridges in order to cross valleys, and many embankments are needed to uniform the terrain on which the tracks are built. This increases wind velocity as the distance to the surface is larger; thus, under the same wind conditions, the train will withstand larger wind loads on a bridge than on flat terrain. Consequently, the risk of a train being overturned by crosswinds is increased due to the combination of three factors: an increase of the vehicles speed, an increase of the wind velocity that reaches the vehicles and a reduction of weight. A reduction of the weight decreases the stabilizing momentum, while an increase of both the vehicle and wind speed produces larger overturning moment.

This section raises the question of why should the topic of crosswinds be analyzed, and it also presents the main research lines that are currently open. The goals of this thesis, as well as the structure of the book, are expressly proposed at the end of the chapter.
Chapter 1: introduction

1.1 What makes crosswinds over high-speed trains a new object of study

The study of crosswinds over high-speed trains began in the late 90’s, but it was in the last decade when the crosswind problem has deserved a fully attention of the scientific community, rolling stock manufacturers and rail administrators. Wind was always present and trains could run before lateral wind was considered a risk. Therefore, why crosswinds have raised as a problem that was not in the past?

In recent years, rolling stock manufacturers pursued to develop new faster vehicles to connect distant cities in the shortest time possible, and increasing the number of seats per car, too. Traditionally, the mean of transport that competed with railways was road vehicles, and if we focus on the transport of passengers, such mean were buses. Flying was the only option to cover large distances since trains were slow due to both infrastructure and vehicle limitations. In this scenario, a person who wanted to cover short and medium distances would have chosen between travelling by car or by train. Trains were relegated to a secondary place as roads and cars kept improving. Rail transportation was reduced somewhat to freight because tracks were not getting the investment required to be an alternative to road transportation. Authorities pushed road and air transportation, and under such strategy, trains lost the people’s interest. Nevertheless, roads got saturated as cars became accessible to the mid classes needing to move people by other mean of transport, which brought back rail transportation after years of declination.

Environmental awareness also grew progressively. Cars clearly have a negative impact on the environment because of their fuel-based propulsion that made electric trains to become a real alternative since they do not locally contaminate. Lastly, the railway industry developed and it turned out to be key and profitable for the economy in some countries, mostly European.

In their way to become a real alternative, the railway industry researched in technological developments that improved passenger’s comfort and travelling speed. The latter is what matters to crosswinds because an increase of the vehicle’s speed also creates higher aerodynamic forces. The goal of the industry was to make high-speed vehicles able to run over 300 km/h to compete not only against road transportation but also against the transport by air. Actually, trends show that vehicles have speeded up an average of 75 km/h in the last 20 years (FIGURE 1). Such improvement was partially accomplished by designing lighter vehicles, with chasses and decorative elements made in new materials that reduced the overall weight of trains. FIGURE 1 shows that vehicles are lighter today than they were in the past despite of having increased its velocity, and incorporated new elements that improved the passenger’s comfort.
Chapter 1: introduction

Obviously, faster vehicles require better tracks so national governments built new tracks ad-hoc for high-speed trains. They were outlined with as many kilometers in tangent track as possible, and with very large radius for curves. For example, a track on which vehicles run at 330 km/h must include curves with a minimum radius of 6500 m (source: ADIF). Designs included a great number of bridges and embankments to fulfill these requirements but it also increased the wind loads that trains should withstand.

All these phenomena working together, made high-speed vehicles to withstand larger wind loads, mainly due to the increase of the travel speed and to the higher wind velocity that reaches the track. Moreover, trains are today more vulnerable to crosswinds since the balancing force, which comes only from the vehicle's mass, is lower since vehicles are lighter.

FIGURE 1: Weight and speed trends of Shinkansen vehicles (up) and ICE vehicles (down). Weight in the Shinkansen family is measured with empty vehicles whereas in the ICE family weight is measured with fully loaded vehicles.
1.2 A multidisciplinary subject

Aerodynamic engineers seek to design an aerodynamic shape at early concept stages to minimize the air resistance, usually called ‘drag’. They define the external geometry of the end car, checking whether the drag value is appropriate with wind tunnel tests and/or simulations. This step only concerns aerodynamics since it does not require taking into account that trains are also dynamic systems. One of the major problems that manufacturers encounter is that an optimum external shape to reduce the aerodynamic drag making the train energetically efficient has the trade-off of worsening the train behaviour against crosswinds. The aerodynamics of the train determines the loads created by winds but the vehicle dynamics are in charge of being able to withstand those wind forces.

Studying crosswinds is complex because one must consider train dynamics as well as train aerodynamics. The aerodynamic part of the problem is about the interaction between the airflow and the vehicle whereas dynamics refers to the response of the vehicle to the wind loads. A wind gust hitting a train from the side takes the vehicle to a dynamic condition that requires a self-dedicated analysis. How should be the problem outlined? The border between aerodynamics and dynamics is not clear on each side of the problem influences the other, a fact that handicaps doing a proper analysis. Extracting any conclusion by only considering the aerodynamics may lead to reach incomplete conclusions since the airflow by itself does not capture the full problem. Likewise, dynamics cannot be detached from aerodynamics to do an easier study either. Technically, it happens like this because the crosswind topic is what is called ‘a coupled problem’. In such circumstance, the best approach would consist on solving both dynamics and aerodynamics at a time, i.e. by connecting a code that computes the vehicle dynamics to another computer program able to solve aerodynamics. The other alternative consists on solving the whole problem in the computational fluid dynamics software (CFD) by introducing the train dynamics within the flow problem. FIGURE 2 shows a hypothetical step-by-step guide:
The mentioned methodology would supposedly be the best representation of the actual phenomenon yet some drawbacks make it to be still a line under research. The time to compute the whole problem simultaneously increases drastically for various reasons. The wind flow must be solved in the time domain since the bodies position change at each instant of time. The latter also implies using deformable meshes, which also contributes to increase the computational effort. The complexity of the model should be also considered. The key is to verify if the coupled model gives more accurate results or its inherent complexity adds new uncertainties. By coupling everything in one model, some simplifications are frequently assumed to make the computation more affordable. In this context, a proper analysis of the reliability of the coupled model must be done by checking its advantages in terms of accurateness and affordability.

At the moment, the study of the train aerodynamics and dynamics is done in two-step calculation, assuming that this is at present the engineering approach that has proved to have a sufficient safe margin. Such approximation allows us to create two accurate models of the train, one for the aerodynamic study and another for the dynamic analysis. The standard step-by-step procedure includes firstly the calculation of the aerodynamic coefficients in wind tunnel tests or with Computational Fluid Dynamics (CFD) simulations. Typically, train models are static and they are oriented to wind so as the yaw angle is artificially obtained. What we get from this step could be either the aerodynamic coefficients’ value or the time signal of the wind forces to be introduced in the dynamic model. Once the aerodynamic problem provides the data for the simulation of wind forces, they are exported to the dynamic model to solve
the vehicle dynamics. At this step, we can measure aspects like the displacements of the main bodies, study the suspension characteristics or the wheel/rail contact. This procedure does not reflect though the interaction between the dynamic system and the wind loads since the aerodynamic forces were computed in a previous step and the model was fixed to the ground.

1.3 Defining ‘safe operation’

Standards guarantee the safeness of the vehicle in terms of the wheel unloading ratio. The European Standard EN14067-6:2010 is thoroughly described in Chapter 2, whose principle is that vehicles can withstand wind loads until a 0.9 wheel unloading ratio is reached. The characteristic wind curves (CWC) of the vehicle (see Figure 3) define graphically this criterion by paring the speed of the most-sensitive-to-crosswind car with the wind velocity that leads to 0.9 of wheel unloading. It must be pointed out that the standard states that reaching a wheel unloading ratio above 0.9 does not mean that the vehicle is going to overturn but only to be operating in risk of overturning. In this way, the term ‘vehicle in stable operation’ used in the standard does not refer to the dynamic behaviour of the vehicle, meaning only that the wheel unloading of the vehicle is below the given limit. The calculation of the CWCs can include the effect of uncompensated accelerations and they can be elaborated for several yaw angles.

FIGURE 3: Schema of a CWC defining the stable and instable areas of operation. $v_w$ stands for wind speed and $v_{tr}$ stands for vehicle speed.

The assessment of the CWC that characterize the vehicle against crosswinds has two well-differentiated stages. Firstly, either wind tunnel tests or CFD simulations provides the aerodynamic coefficients of the vehicle. The second stage is for building
a representative dynamic model of the vehicle that can be a simple quasi-static model or a full multibody model. In this way, the standard reflects that the crosswinds study is multidisciplinary. Furthermore, the process of calculating CWCs in two independent stages shows the classic approach to the matter even when aerodynamics and dynamics are coupled in a real situation.

Rail administrators, e.g. the Spanish rail administrator ADIF, require the CWCs of all the vehicles running in their tracks to guarantee a safe operation. The procedure consists in installing anemometers along the line to measure the wind speed, watching with particular attention those locations that are known for usually having high wind velocities. By knowing the wind velocity and the vehicle running on the track, they are able to limit the train velocity depending on this meteorological factor. FIGURE 4 shows the speed limits of the vehicles owned by the railway operator RENFE by 2009, showing that there are vehicles especially sensitive to wind, such as series S130, and other trains less affected, i.e. classes S100 and S102. The latter trains should only reduce the travel speed below 300 km/h when the wind speed is higher than 100 km/h, a situation that seldom takes place.

![FIGURE 4: Maximum speed limit of RENFE vehicles as function of the wind speed value. Source: ADIF (2009)](image-url)
1.4 Currently open questions on the design of fences to protect high-speed trains

Currently, there is a clear need of reducing the impact of wind on high-speed vehicles due to the reasons that have been exposed in Section 1.1. Manufacturers can optimize the behaviour of trains against crosswinds but there are a large number of influential parameters that depend on some aspects coming from the civil engineering and the topography of the track surroundings. Certain lines are more exposed to crosswinds than others; hence, corrective actions try to reduce locally the wind speed, e.g. including wind-breaking devices that protect vehicles from wind.

It may obvious that adding parapets near the track will reduce the wind speed that reaches the vehicles, and as FIGURE 5 shows some lines already have tracks sheltered with parapets. Both examples are elevated tracks corresponding to a bridge and a viaduct, which is not a coincidence, because Section 1.1 stated that the track is more exposed to wind on these locations.

![FIGURE 5: Parapet installed at the Uetsu line in Japan (right). Parapet installed in TGV Méditerranée line at the Viaduc des Angles in France (left)](image)

In general terms, three are the main parameters that define a wind fence:

- The height of the protection.
- Eaves, as they can be built on the top of fences to deviate the flow.
- Porosity. Fences can be either solid or porous, letting some wind pass through them.

Beyond the implications in relation with the vehicles, including parapets on the infrastructure implies to reconsider certain aspects of the civil engineering. By adding a windbreak device, some problems arise just from the fact that parapets must absorb the force induced from wind and they cannot swing. Under severe wind conditions, the parapet loaded by wind must withstand the forces and transmit them
Chapter 1: introduction

to the support. If the track is on the ground, either with or without embankment, the
transmission and installation involves no particular issues. It does not happen in the
same way when parapets are installed on viaducts or bridges. Being the fences
installed on these elements of the infrastructure, wind forces are not transmitted
directly to the ground. In a wind scenario, bridges and viaducts must be able to
withstand forces coming from wind, but with parapets installed on them, such force
increases considerably. Thus, the infrastructure should be able to resist the higher
forces that parapets transmit.

There are some open questions related to parapets regarding how parapets influence
the dynamics and aerodynamics of trains. We already know that substantial changes
on the vehicle aerodynamics influence somewhat the vehicle dynamics. By installing a
parapet near the track, the flow around a high-speed train will be altered. There are
parameters of the fence geometry -such as the height- that will definitely introduce
substantial changes into the flow structure. At this point, questions like the followings
arise: Do fences always reduce the impact of wind and thus increase safety? What is
the design for an optimal fence? How should the ends of fences be designed? Rail
administrators are today sceptical about the inclusion of parapets to protect vehicles
as these questions are still under research.

Fences reduce wind loads but high fences can create suction forces that might be as
large as those directly induced by crosswinds (Barcala and Meseguer, 2007). The value
of the fence height that makes suction forces to appear also depends on the specific
characteristics of vehicles. Therefore, one of the main obstacles on this kind of
studies is the difficulty to generalize a problem that is not already easy to solve.
Although, there are standard fences that can work in most of the situations, finding
the optimal design needs of specific studies to confirm the hypothesis since safety
can be compromised.

The calculation of a scenario to measure the sheltering effect of fences is usually
done by placing the train model on the track and between the fences with no relative
movement. Thereby, they are mostly static tests either in a wind tunnel or with CFD
simulations where the train stands still and the flow evolves to a steady state. Even if
fences guarantee the safety levels imposed by the rail administrator, the open
question of what happens when the train enters and exits the sheltered track has no
answer yet. To answer this question, one should consider the train in movement
along the scenario. In terms of CFD calculations based on the eulerian approach such
as ANSYS Fluent (ANSYS Inc., 2010), moving models require to include an adapting-
mesh approach since the mesh should be adapted for the new position of the moving
geometry. However, the time needed to solve accurately these kind of problems is
still non-affordable with current computers in most cases. Wind tunnels are not a
good tool either since experimental tests with moving models are not accurate
enough because creating the conditions is not possible in most occasions.
Moving models are not in the scope of this thesis, yet we did CFD calculations with a code based on the lagrangian approach (see Section 5.2) that may ease the simulation of these scenarios. CFD codes based on the Lattice-Boltzmann Method (LBM) are meshless; therefore, their main advantage is that the setup of the problem does not require the pre-process step of creating a mesh for the scenario. The computational time required to obtain accurate results is also very high, nevertheless the time that engineers should dedicate to prepare the calculations is supposedly lower in this kind of CFD codes. However, LBM codes are still under research and they need to be firstly validated for each application (see Section 5.3).

1.5 **State of the art**

Researching on the crosswind effect on high-speed trains involves the study of both the aerodynamics and the dynamics of the car. In fact, it is not possible to make progress on this field without making progress on both faces of the problem. We already presented the idea of coupling aerodynamics with dynamics, which would be the ideal way of analysing the crosswind effect by solving the aerodynamics and the vehicle dynamics together. However, current studies focus either on vehicle dynamics or aerodynamics. Gathering information about this topic can be cumbersome since there is so many open questions that the collection of research projects, publications and standards may end in a sort of unconnected material. The key lies on relating how an increase of the knowledge about aerodynamics makes progress in the understanding of dynamics, and vice versa.

'TRANSAERO – A European Initiative on Transient Aerodynamics for Railway System Optimisation' (Schulte-Werning et al., 2002) supposed a kick-off on the modern research of crosswinds over high-speed trains. The project took place in 1999 and by that time, the European Union was already encouraging manufacturers and administrators to increase travelling speeds. To accomplish such goal, new infrastructures were built and new light trains with state-of-the-art technology reached maximum speeds. In such context, studies awarded about possible new problems and crosswinds were one of them. The project covered not only side wind effects, but also trains passing effects and pressure wave effects. Studying crosswind effects was done by doing some on-track tests, by measuring aerodynamic coefficients in wind tunnel experiments and by simulating the flow with CFD within the possibilities of computers at that time.

A selection of the available publications covering aerodynamics and dynamics of vehicles under crosswinds aims to clarify the starting point of the research done during this thesis. The state of the art review has two parts: one section is devoted to collect those publications that focus on aerodynamics whereas another refers to the bibliography regarding dynamics of vehicles in the crosswind context.
1.5.1 Aerodynamics

There are three state-of-the-art reviews that are very helpful to be aware of what have been done, what is the current situation and what are the current challenges. Baker et al., 2009 does a general overview, including wind tunnel experiments, CFD simulations and even dynamics of the vehicle. Another review is dedicated to the nature of flow since it focuses on the aerodynamic behaviour of vehicles under crosswinds (Baker, 2010). Finally, (Raghunathan et al., 2002) describes some issues that have come up in recent years as a consequence of having increased the speed of vehicles. This work is not focused only in crosswinds but it may helps to have a general perspective of the general problems of vehicle aerodynamics.

A proper study of aerodynamics is essential to represent accurately wind loads for the simulations with multibody models. Aerodynamics is needed to characterize the vehicle and also to let us know at which degree the train is sensible to crosswinds. One crucial characteristic of the flow around trains is the inherent turbulent nature. This issue increases the requirements of wind tunnels to do experiments, and constraint CFD studies to make use of turbulent models in the case the chosen is the computational. Chapter 5 is dedicated to the explanation of the current models, which can be classified as RANS and URANS models if all the scales of turbulence are modeled or LES if large scales are solved and only small scales are modeled.

Historically, wind tunnel tests were the main topic in the research of aerodynamics. Experimental tests are proven to give valid results of aerodynamic coefficients at a reasonable cost. Although certain methodologies increase the information derived from wind tunnel tests (Sanquer et al., 2004), these experiments usually give limited information about the flow (structure, flow parameters...) and on the surface of the coach. An alternative has risen in the last decade with CFD because early codes were not fully operational and valid alternatives (Gaylard, 1993). The scientific community has developed during the recent years knowledge on this field being helped by the continuous advance of computers. This approach opens a new world of possibilities since CFD allow us to know what is the magnitude of almost every parameter and to study the flow structure. However, turbulence models introduce uncertainty and there is a big dependency on which model is chosen since exact solutions are still not feasible. As current standards reflect, CFD simulations still requires to be validated in some manner with experimental tests.

Although published works with numerical studies are just a few, researches pushed CFD to make it being at the same level of wind tunnel tests. Most of the current research covers wind tunnel tests in which the vehicle stands on complex infrastructure such as embankments or bridges. Some works also tried to represent the relative movement between car and ground without losing accuracy in wind tunnel tests. Considerable number of authors does research on the mathematics
behind the modeling of wind gusts. Lastly, some others develop unsteady models of wind to be considered in simulations instead of deterministic approaches. However, such works are not included in this revision since they are not a point of this thesis.

C.J. Baker is possibly the most important researcher on the field, having published works since the late 80s. In a paper from 1985, he already pointed out that wind flow on an embankment speeds up, and that effect is not easy to reproduce in wind tunnel tests (Baker, 1985). In fact, such difficulties make the issue to be still opened (see Section 2.2.4) and the methodology exposed in his work is today a reference. Since then, contributions helping to solve singularities of different infrastructures are being published. Furthermore, infrastructure does not equally disturb the flow around every kind of vehicles since the carbody geometry also plays an important role on how embankments or bridges modify the aerodynamic coefficients (Suzuki et al., 2003). In reference to embankments, Cheli et al., 2010 concluded that at low angles of attack (below 40º), if a coefficient is defined by adopting as reference velocity the wind speed measured over the track, it is equivalent to the one measured on the flat ground. As literature proves, this conclusion can be extrapolated to other types of vehicles (Schober et al., 2008).

In Baker, 1986 and Humphreys and Baker, 1992 results of the first tests carried out in wind tunnel with moving models are presented. Trying to represent the relative movement between vehicle and ground is not straightforward. The effect of atmospheric turbulence is hard to reproduce and may modify somewhat the measurements on static models, especially at low yaw angles. Thus, results show that the drag force coefficient was lower in moving tests but the lift coefficient did not change. More recent tests demonstrated that if tests are correctly set up, coefficients from moving models do not change considerably in comparison with still models (Bocciolone et al., 2008).

This thesis covers the analysis of the sheltering effect of wind fences on high-speed trains. Research has been done since TRANSAERO project took place but even today, wind tunnel tests are the main manner to increase knowledge. Tests cover the inclusion of fences in most of infrastructures: flat ground, embankments, viaducts or bridges. Results from Bocciolone et al., 2008 and Cheli et al., 2011 shows that fences can reduce drastically the risk of overturning. The procedure to reach any conclusion is as follows. A still model is placed between fences and its aerodynamic coefficients are measured in that situation. Then, authors compare the situation between a vehicle being protected by fences with the scenario of unsheltered vehicle. Mostly, a few parameters of fences are compared since parametric studies in wind tunnels are costly. Usually, a parameter among height, solid or porous is modified to check the influence of the parameter. In Barcala and Meseguer, 2007 a big number of parameters are modified; several heights as well as inclination or length of eaves are tested. It was possible to accomplish such analysis by reducing the model to 2D and
assuming the limitations of two-dimensional studies. Conclusions can be drawn in terms of relative effect of the fence parameters but the flow around the vehicle cannot be studied since for this purpose a 3D study is required. In Avila-Sanchez et al., 2010 a similar analysis is carried out but the approach of this work is to study the effect of fences on the catenary. In this case, the assumption of a two dimensional flow is valid because both the bridge and the catenary can be considered as infinite bodies.

The following references studied how to aerodynamically characterize the cross-section of bridges. They were of interest since one chapter of the thesis studied the effect of fences on the cross-section of a train standing on a bridge. They were useful to properly understand the vortex shedding phenomenon that appears when a bluff body is inside a flow and the great dependency on the Reynolds number on 2D studies. Schewe, 2001 studied with experiments what is called ‘more-or-less bluff bodies’ that are shapes with rounded edges. Larose and D’Auteuil, 2006 presented an analogous study of bluff bodies with sharpened edges and also from wind tunnel tests. Schewe and Larsen, 1998 experimentally analysed how to test cross-sections of bridges. Similarly, Mannini et al., 2010 simulated in the 2D space a bridge in terms of URANS computations.

Flow structure is of interest because there is a direct relation between the flow and the pressure value on the vehicle surface, hence a relation with the aerodynamic coefficients of the vehicle. Flow structure is well defined in a paper by Copley, 1987 which shows that the flow around high-speed trains have two well differentiated regimes depending on the yaw angle despite having an aerodynamic geometry. Chiu and Squire, 1992 complemented the research done by Copley focusing on high yaw angles instead. In Gawthorpe, 1994 a good explanation of the flow structure is presented. The authors describe the flow at low yaw angles (below 45º), at the transitions phase (between 45º and 60º) and at high yaw angles (above 60º). Such effect can be observed because aerodynamic coefficients have a common trend that is shared by most of high-speed trains. A linear increase is typical at low yaw angles, followed by transition where a maximum is reached, from which the value decreases to a minimum at high angles of attack (see FIGURE 6).

CFD calculations of crosswinds over trains are usually done with codes based on the Finite Volume Method (FVM) such as Fluent (ANSYS Inc., 2010), being Diedrichs, 2003 a good example of pros and cons of using this technique. Text covers most of the process to be done in order to have a good setup of the computational model. The type and the quality of mesh, turbulence models, numerical schemes and different codes are compared in terms of the values of the aerodynamic coefficients. The goal was to validate RANS simulations with experimental tests to prove the possibility of carrying out aerodynamic analysis with CFD. Usually, RANS simulations give higher values of side force and smaller values of lift force. Hence, the equivalent roll
moment around the leeward rail in comparison with experimental tests keeps almost unchanged (Diedrichs, 2008). Similar to this works is the one in Diedrichs et al., 2007 where the authors represented the speeding factor of embankments with numerical simulations.

Regional trains are not under the scope of this thesis. However the analysis in Diedrichs, 2010 is of interest to notice the changes on the behavior of the flow structure around the train and the impact of these differences on the trends of the aerodynamic coefficients.

Another good example of CFD calculations using LES simulations can be found in Hemida and Krajnovic, 2010. Simulations that use these turbulence models are proved to be more accurate than RANS computations since LES only model the small scales of the flow field whereas large scales are solved. The influence of nose shape was herein investigated showing that LES models also reproduced the changes on the flow field as function of the yaw angle.

Another approach to solve fluid dynamics consists on solving the Boltzmann equation in terms of the so-called Lattice Boltzmann Method (LBM). EN 14067-6:2010 allowed manufacturers to calculate the wind forces over trains in terms of LBM and is also present in Baker et al., 2009. However, the study of railway aerodynamics in terms of the LBM is not spread in the scientific community, and only a few works are available in the literature. In Wang et al., 2008 the authors presented results, though not contrasted, from simulations using a model of a high-speed train to show the capabilities of the LBM. Other fields have also tested the LBM as a viable alternative.
to simulate fluid dynamics obtaining reasonable good results. An example of a study from the automotive field can be found in Fares, 2006, where simulations with the Ahmed reference body are done to validate the LBM in the field of external aerodynamics. Results show good agreement with experimental and other computational studies at the two slant angles (25° and 35°) that were simulated. In Holman et al., 2012, the software used in this thesis is validated against drag calculations with the Ahmed body. Finally, the flow field around the superstructure of a frigate can also be calculated in terms of the LBM, showing that computations agree adequately with experiments (Sym, 2008).

FIGURE 7: Isosurface of the instantaneous static pressure with value -0.4 Pa showing the instantaneous wake structures taken from (Hemida and Krajnović, 2010). Case of yaw angle 90 degrees (top) and case of yaw angle 35 degrees (bottom).

1.5.2 Dynamics

Topics covering vehicle dynamics under crosswinds can be classified as: alternative estimations of overturning risk, developing of simplified models, and analysis of the vehicle parameters that have a direct impact on the vehicle behaviour. TRANS AERO helped to set the basics of the vehicle overturning process. Wind loads turn the carbody and this moves the vehicle into an instable position. If the vehicle masses are significantly displaced from the equilibrium position, wheel-rail vertical forces change, decreasing in the wheels located in the windward side.
Crosswind is usually an external factor that can be added to a set of different phenomena that may appear during the running of a vehicle. The authors in Andersson et al., 2004 classified the main causes that could make reaching the wheel unloading limit at the leading bogie. Crosswinds are the main cause of wheel unloading with 63 per cent of relative importance, followed by dynamic forces coming from track, i.e. due to track irregularities (15 per cent). Lateral acceleration contributes with as much as 14 per cent and finally suspension deflection with 8 per cent. Nevertheless, the authors use GENSYS multibody software during the risk evaluation process in order to relate wind velocity with cant deficiency and vehicle speed values. The authors already apply the wheel unloading criterion to guarantee that the vehicle is not under risk of overturning, addressing that the results obtained from the multibody simulations are hardly checked with on-track tests.

Some authors focused their research on finding adequate vehicle and track models that were able to reproduce the behavior of the car under side wind. A demonstration of getting reliable values of Q-forces with simpler vehicle models is presented in the work by Diedrichs et al., 2004. The goal of this work was to check whether simpler quasi-static models can provide wheel unloading ratios as accurate as those calculated by using full multibody models. They also considered curved tracks by introducing the equivalent uncompensated lateral accelerations in the simplified models and by making the full model run in curved track. Three types of vehicle were checked: conventional vehicles in which only the leading car is modeled, Jacobs’-bogies-based vehicles, and semi-trailers. Results are given in terms of Q-forces and wind curves, for tangent and curved track. The best agreement was reached for the conventional and Jacob’s bogie vehicles where the difference between models was almost zero. The semi trailer model showed higher but still depreciable differences between models. The highest permissible wind speed varies around 3 m/s and values of Q-forces from 4 to 7 per cent, which seemed to be good results. Carrarini, 2008 focused instead on the parameters to be included in the car models to simulate crosswind effects on the vehicles. He stated that models with linear suspension elements and without model of wheel/rail contact were appropriate to calculate CWC curves, and even to analyze the transitory response to wind loads. The work of Xu and Ding, 2006 is slightly different since their model aims to represent the interaction between the vehicle and the track. This research is essentially interesting due to the author’s track model. Two parallel rails of a track were modeled as two continuous beams supported by a discrete-elastic foundation of three layers with sleepers and ballasts. The complex track model was compared with a standard model of rigid track showing that lateral and vertical displacements of the carbody were identical. Similarly, a complex methodology trying to reproduce the interaction between vehicle, wind and the bridge on which the vehicle runs is explained in Li et al., 2005.
Other authors such as Alfi et al., 2010 starts the exposition assuming that crosswinds are already safety matter, focusing instead on how to guarantee the safety margins. In this case, the authors developed an active control of airspring secondary suspensions in order to improve ride comfort and safety. The secondary suspension is a set of key elements in the crosswind context and the airsprings parameters determine the vehicle reaction to wind loads. Although the crosswinds analysis is not a fundamental part of the paper, the case of vehicle under crosswinds is useful for the authors for proving that their system increase safety margins. By including an active suspension, wheel unloading decreases despite having bigger oscillations when wind loads the car.

Thomas did a great contribution to the state of the art of vehicle dynamics under crosswinds. Research presented in Thomas et al., 2010 compared different models of gusts in order to contrast which one is less conservative and which one produces highest values of wheel unloading. Values of the vertical position of carbody's center of gravity (CoG), the lateral stiffness of secondary suspension and the maximum deformation of bumpstops were modified in order to analyze the sensitivity to these parameters. Authors demonstrated that a simple trapezoidal-like load produces higher values of wheel unloading than the standardized TSI-like gust, and that the worst scenario is a step shape wind load. The sensitivity analysis helped to state that increasing the bumpstop deformation could increase vehicle instability. Under these conditions, the value of the carbody's CoG is a key parameter because it directly modifies the car restoring moment. In Thomas et al., 2010 tests on track were compared with 2D and 3D models of the real car. They concluded, as was previously stated in other works, that it is difficult to carry out tests on track with high values of wind speed, but at the same time, they demonstrated that reproducing such tests with computational models is reachable. Since track tests are rather expensive, another possibility is to build a test bench and try to reproduce wind gusts with full-scale measurements (Thomas et al., 2011). Looking at these results, one can conclude that the representation of wind loads in a bench is complex. Furthermore, simulations and measurements did not agree quite well when values of wheel unloading are analyzed.

Baker et al. presented an interesting work with the purpose of simulating long scenarios of unsteady wind (Baker et al., 2011). Crosswinds are in most occasions a comfort issue more than safety issue because once the standards are fulfilled; there is a guaranty that those vehicles can run without being in risk of overturning. This work imitates the comfort tests procedure in which a several kilometers long track is simulated. This kind of tests are mostly interesting when there is data available from real track irregularities and the instabilities introduced by the track can be superposed to wind loads.
To a lesser extent, works like Wetzel and Proppe, 2010 and Carrarini, 2007 presented alternatives to the deterministic approaches that are in the standards. Their methodologies give as a result Probabilistic Wind Curves (PCWC) that is a more accurate approach than the widespread approach of CWCs. The idea behind PCWCs is to give an estimation of the probability that an event of certain wind and velocity speed takes place, considering in the calculation some of the uncertainties of the vehicle’s parameters or the uncertainties inherent to the calculation of the aerodynamic coefficients.

1.6 Aim of the thesis

This thesis aims to answer some of the open questions related to the effect of crosswinds on currently high-speed trains. In order to fulfil our goal and based on the reasons previously exposed, the present work needed to focus in both dynamics and aerodynamics of the problem. Regarding vehicle dynamics, this thesis seeks:

- To study the scope of simplified models to analyze some aspects of the vehicle dynamics in a crosswinds context.
- To assess whether crosswinds can become a problem of comfort in addition to the already known safety matter.

The other face of the crosswind problem is the aerodynamics of high-speed vehicles. In this field, numerous works have been published in recent years that study the characteristics of the flow around trains. However, there is still a need of research to know how wind-breaking devices influence this already known flow. This is the reason of having been focused on:

- To analyze the effect of fences on high-speed trains in terms on two-dimensional CFD simulations. To answer the question of what conclusions can be extracted from simplified models that can be extrapolated to a three-dimensional domain.
- To validate a meshless CFD code based on the Lattice-Boltzmann theory by contrasting results from simulations with experiments in wind tunnel tests available in the European Standard EN 14067:6-2010.
- Lastly, to measure the effect of wind fences in the 3D space in terms of a CFD code based on the Lattice-Boltzmann method. These simulations will verify the results provided by the 2D model and will quantify the effect of parapets in terms of the wheel unloading ratio.
1.7 Structure of the thesis

This thesis is structured in seven main chapters covering both the dynamics and the aerodynamics of vehicles in the crosswind context.

Chapter 2 presents the main sections of the European Standard EN 14067:6-2010 aiming to help in the understanding of the most important clauses. It also deals with a comparison of the recommendations from the different standards that apply currently in Europe.

The models for the dynamic and aerodynamic studies are collected in Chapter 3. The verifications and validations of the simplified models are already discussed at this section of the thesis.

Chapter 4 deals with the analysis of the vehicle dynamics in the crosswind context. Results firstly discuss the scope of 2D models to study vehicle dynamics and to guarantee safety. It also analyses vehicle dynamics running on a curved track while they are being subjected to crosswinds loads to discuss whether crosswinds can also become a comfort issue.

The study of vehicle aerodynamics starts at Chapter 5. The presentation of the equations of the continuum-based CFD theory opens the topic, as well as the equations from the Lattice-Boltzmann Method (LBM). The latter is the theoretical base of the meshless CFD code that was used in the thesis. Since these CFD codes need to be firstly, this chapter also presents our validation in terms of EN 14067:6-2010.

Chapter 6 analyses thoroughly the effect of wind-breaking devices on high-speed trains. Firstly, a 2D model of a vehicle standing on a bridge approaches the problem by studying the main parameters of the fence design. Next, the chapter presents results with simulations in the 3D space with a few fence models used in the 2D model. This allowed us to be able to make a comparison with the computed results from the two-dimensional model.

Finally, the last part of the thesis presents the conclusions reached in the framework of this investigation and outlines some research lines that could be undertaken. Moreover, the author’s publications and the bibliography can be found at the end.
Chapter 2  

current regulation

The European Standard EN 14067-6:2010 could be the starting point for doing research on the crosswind topic. The standard is a source with information describing aspects such as CFD simulations, wind tunnel tests, dynamic models of vehicles and clauses for guaranteeing a safe operation. The current regulation reveals the complexity and the coupled nature of the crosswind problem. A standard covering together topics as the vehicle dynamics and aerodynamics is not common, but the current approach was needed to standardise the full procedure.

This European regulation has been changing during the last years to adopt recommendations suggested by the members of the committee in charge of the standard formulation. In fact, we handle up to three drafts of EN 14067-6:2010 in the time we researched on this topic, being prEN 14067-6:2007 the first draft available to the public.

Herein, the explanation of EN 14067-6:2010 highlights the most relevant sections including some clarifications to help with the standard comprehension. This section is not an interpretation of the standard since the information gathered here exactly reproduces the directives within the document.

Several standards cover the crosswind topic guaranteeing a safe operation. A comparison is available at the end of the section to clarify the position of each one in regard to the most important points of the vehicle validation. Even though the crucial recommendations do not vary between standards some of the procedures change between them. Currently, there are national guidelines such as DB Ril 80704 in Germany, or GM/RC2542 and GM/RT2142 in the United Kingdom. Moreover, the Technical Specification for Interoperability (TSI) of high-speed rolling stock applies for vehicles running across borders in Europe. Lastly, EN 14067:6-2010 started ruling in 2010 for new railway vehicles operating in Europe.
2.1 European Standard EN 14067-6:2010

This European Standard is part of the series ‘Railway applications – Aerodynamics’, which in addition to cover the crosswind assessment, the series establishes requirements and procedures for aerodynamics in open tracks and tunnels. The full title is ‘EN 14067-6 Railway Applications - Aerodynamics - Part 6: Requirements and test procedures for crosswind assessment’ and it applies to those passenger vehicles with velocity up to 360 km/h and freight vehicles with velocity up to 160 km/h.

2.1.1 Determination of aerodynamic coefficients

Any calculation begins with the measurement of wind forces or the determination of the vehicle’s aerodynamic coefficients. Therefore, the standard firstly deals with the two possible approaches that allow us to measure wind loads on vehicles: CFD simulations and wind tunnel tests. The chief purpose of applying CFD is ‘to determine the set of aerodynamic loads appropriate for determining the mechanical stability of a critical vehicle’. Additionally, the goal of wind tunnel tests barely differs from CFD simulations, thus the standard explains a similar concept in other words. It states that wind tunnel tests ‘enable the determination of the vehicle aerodynamic force and moment coefficients, including the rolling moment coefficient about the lee rail’.

In regard to CFD simulations, the standard describes the requirements to properly set up the problem and to correctly represent the vehicle. Due to the same reasons, instrumentation requirements, train model setup and other aspects of wind tunnel tests are briefly described.

Note that dynamic simulations can be performed without having calculated any aerodynamic coefficient since the standard provide enough data to carry on generic studies with the reference vehicles. Annex C and Annex E of the standard include the coefficients of ICE 3, TGV Duplex and ETR 500 in different ground configurations (see Section 2.2). Once the aerodynamic coefficients are got somehow, the wind loads can be calculated and the vehicle dynamics studied.

a) CFD simulations

CFD simulations are useful since creating large computational domains can significantly reduce the blockage effect. Furthermore, calculations with high values of the Reynolds number can be carried out easier than in wind tunnel tests.
CFD simulations require experimental tests to firstly validate the computational model. Annex C presents the aerodynamic coefficients of the three reference vehicles (ICE 3, TGV Duplex and ETR 500) on the single track and ballast ground as benchmark values (see Section 2.2.1). The goal is to replicate the experimental tests checking whether the force state in the CFD is close to experimental tests and within the limits recommended by the standard. The reference value is \( C_{M,lee} \) which represents the wind induced moment acting around the line of contact between wheels and the lee rail. The standard validates those simulations that keep \( \varepsilon_{\text{max}} \) below 0.15 and \( \varepsilon_{\text{mean}} \) below 0.1.

\[
\max \left( \frac{C_{M,\text{lee}} - C_{M,\text{lee},\text{rms}}}{C_{M,\text{lee},\text{rms}}} \right) < \varepsilon_{\text{max}} \tag{2.1}
\]

\[
\text{mean} \left( \frac{C_{M,\text{lee}} - C_{M,\text{lee},\text{rms}}}{C_{M,\text{lee},\text{rms}}} \right) < \varepsilon_{\text{mean}} \tag{2.2}
\]

Simulations can be performed in terms of the continuum theory governed by the momentum equations, the so-called Navier-Stokes equations, or in terms of the kinetic gas theory, the so-called Lattice-Boltzman equation. Direct Numerical Simulations (DNS) are the only that provide the exact numerical solution of the problem; however, the required computational resources make the method non-affordable. Large Eddy Simulations (LES) and Detached Eddy Simulations (DES) approximate the solutions taking into account the inherent unsteadiness of the flow. Reynolds-Averaged Navier Stokes (RANS) simulations are steady and may be only used in those cases where the flow is 'evident'. Further information about turbulence models can be found in Chapter 5.

The whole model should be scaled using a unique ratio to achieve geometry similarity with the actual vehicle. The position of the car that is object of study defines the number of vehicles that must be modeled. If the leading car is the vehicle to test, half a downstream vehicle should be included at least though the standard recommends modeling a full downstream car. If the tested vehicle is an intermediate car, one full model of the car upstream should be included in addition to the downstream body.

As long as the model represents accurately the real vehicle, the aerodynamic coefficients will provide accurate information. The standard recommends paying special attention to smooth and curved surfaces since any transformation to polygonal surfaces will create non-real flow detachments. Simplifications can be assumed in the train under body, the bogies, the pantograph and the wheels. The under body only requires the 'general shape' and bogies can be simplified by only including the geometry that creates the blockage at the region. The pantograph can
be excluded from the model and the wheel-rail contact may be simplified by flattening the wheels base.

Mesh quality is only given in terms of the dimensionless wall distance $y'$. Limit values are given for RANS simulations, keeping the $y'$ value close to 1 for low-Reynolds simulations and between 30 and 150 for high-Reynolds simulations. The only rule applying in LES and DES simulations is referenced to the mesh independency of the results. The difference of $C_{Mx,le}$ between the three refinements that the standard asks to carry out should not exceed 3 per cent. The largest value of roll moment around the leeward rail among the three results has to be chosen relying on a safety criterion.

In regard to the boundary conditions of the computational domain, the standard just recommends setting an inlet and outlet condition able to reproduce wind tunnel tests conditions. The standard concerns about the free flow profile at the inlet boundary. The standard only states that the conditions should be ‘appropriate for the train configuration’ at the outlet, the top and lateral boundaries. The ground can be either stationary or moving, and the train walls shall be treated as no-slip walls.

**b) Wind tunnel experiments**

The recommendations and the procedure to measure the aerodynamic coefficients in wind tunnel tests are partially shared with CFD simulations. This is the case of the information regarding the model setup and accuracy, and the benchmark criteria. EN 14067-6:2010 points out some specific details of wind tunnel testing but a thoroughly description of the right way for carrying out experiments can be found in the normative EN 14067-2:2003.

Keeping the blockage ratio as small as possible is one of the biggest concerns when wind tunnel tests are carried out. The blockage ratio shall be measured at a yaw angle of 30 degrees taking into account both the train model and the infrastructure. Corrections are needed in cases with blockage ratios between 5 per cent and 15 per cent, being aware that these corrections will decrease the value of the aerodynamic coefficients. Hence, no applying any blockage corrections from Annex B is considered a conservative criterion.

The standard describes, though briefly, some other aspects like the Mach number, which should not be higher than 0.3, the boundary layer or the instrumentation requirements.
2.1.2 The wind gust model: Chinese Hat wind scenario

The standard describes a gust model to include in simulations with multibody models. The full wind scenario (see FIGURE 8) includes a linear rise (from \( t_1 \) to \( t_2 \)) to the base level of wind \( (U_{\text{mean}}) \) when the train is loaded in steady state (from \( t_2 \) to \( t_3 \)). The rise of the wind speed corresponds to the Chinese Hat scenario that represents the wind gust (from \( t_3 \) to \( t_4 \)). Between \( t_4 \) and \( t_5 \) the wind speed decreases again to the previous base level also following the Chinese Hat function. Then, another time period with constant wind loads takes place between \( t_5 \) and \( t_6 \). Lastly, a linear decrease to zero wind speed finishes the scenario (from \( t_6 \) to \( t_7 \)). TABLE 1 contains the formulation in order to recreate this wind scenario.

![Wind distribution example as given in EN 14067-6:2010](image)

**FIGURE 8:** Wind distribution example as given in EN 14067-6:2010

<table>
<thead>
<tr>
<th>Interval</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>([t_0:t_1])</td>
<td>(u(t) = 0)</td>
</tr>
<tr>
<td>([t_1:t_2])</td>
<td>(u(t) = \frac{U_{\text{mean}}}{t_2-t_1}(t-t_1))</td>
</tr>
<tr>
<td>([t_2:t_3])</td>
<td>(u(t) = U_{\text{mean}})</td>
</tr>
<tr>
<td>([t_3:t_4])</td>
<td>(u(t) = \text{mirrored Chinese Hat})</td>
</tr>
<tr>
<td>([t_4:t_5])</td>
<td>(u(t) = \text{Chinese Hat})</td>
</tr>
<tr>
<td>([t_5:t_6])</td>
<td>(u(t) = U_{\text{mean}})</td>
</tr>
<tr>
<td>([t_6:t_7])</td>
<td>(u(t) = \frac{U_{\text{mean}}}{t_7-t_6}(t-t_6))</td>
</tr>
</tbody>
</table>

**TABLE 1:** Functions for a temporal wind distribution
The mathematical description of the Chinese Hat refers only to the decaying section of the exponential function; the gust is completed with the increase segment by mirroring the function with respect to the vertical axis of coordinates.

The standard gives a full description of the mathematical model in Annex I but Section 5.4.4.2 of the standard also includes a short version of the formulation. The only difference between both sections is that the short version already assumes the wind hitting the vehicle perpendicularly ($\beta_w = 90^\circ$). Herein, the full version of the Chinese Hat is described specifying the case of the 90-degree wind angle by the end of the section.

The method describes a spatial distribution since the wind gust is fixed in space, hence the transformation to calculate the temporal distribution is only possible when the train speed is constant.

![Coordinate system for the derivation of the wind model](image)

FIGURE 9: Coordinate system for the derivation of the wind model. X,Y is the inertial coordinate system for the definition of the main flow direction w. $\beta_w$ is the angle between wind and track. $u_{\text{turb}}$ are the turbulent disturbances parallel to the main flow direction. $v_{\text{turb}}$ are the turbulent disturbances perpendicular to the main flow direction.

Some of the parameters fixed by the method are:

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference height of the site ($z$)</td>
<td>4 m</td>
</tr>
<tr>
<td>Normalized gust amplitude ($\tilde{A}$)</td>
<td>2.84</td>
</tr>
<tr>
<td>Roughness length of sites representative of interoperable lines</td>
<td>0.07 m</td>
</tr>
<tr>
<td>Probability of a gust duration $T$ for a given amplitude</td>
<td>0.5</td>
</tr>
</tbody>
</table>

TABLE 2: Parameters of the Chinese Hat that are fixed by EN 14067-6:2010
The wind speed along the track and perpendicular to the vehicle is calculated by:

$$v_w = U_{\text{mean}} + 2.84 \cdot \sigma_u \cdot C$$ \hspace{1cm} (2.3)

where $U_{\text{mean}}$ is the mean wind speed, $\sigma_u$ is the standard deviation and $C$ the coherence function.

The calculation of the mean wind speed and the standard deviation is straightforward. We calculate the value of $U_{\text{mean}}$ by dividing the maximum wind speed ($U_{\text{max}}$) by the gust factor, $G$, (see Equation (2.5)). The gust factor is calculated from the turbulence intensity ($I$) and the normalized gust amplitude, $\tilde{A}$. The value of the maximum wind speed is an input to the problem and the reader chooses it. Therefore, the standard deviation can be obtained through Equation (2.6).

$$G = 1 + \tilde{A} \cdot I = 1.284 + 0.3446 = 1.6946$$ \hspace{1cm} (2.4)

$$U_{\text{mean}} = \frac{U_{\text{max}}}{G}$$ \hspace{1cm} (2.5)

$$\sigma_u = l \cdot U_{\text{mean}} = 0.2446 \cdot U_{\text{mean}}$$ \hspace{1cm} (2.6)

The coherence function, $C$, is the last factor of Equation (2.3) that lasts to be calculated but is also the most complex:

$$C = e^{-2.5 \cdot \left( \frac{n \cdot L_{uy}}{U_{\text{mean}}} \right)^2}$$ \hspace{1cm} (2.7)

Firstly, we use the power spectral density (PSD) of the turbulence of the longitudinal component ($u$) in Equation (2.8) to calculate the time constant of the gust ($T$).

$$S_u(n) = \frac{4 \cdot f \cdot \sigma_u^2}{\left(1 + 70.7 \cdot f^2\right)^2} \cdot \frac{1}{n}$$ \hspace{1cm} (2.8)

where $n$ limits the frequency range of the data, either measured or calculated, between $n_1 = 1/300$ Hz and $n_2 = 1$ Hz. $f$ stands for the normalized frequency using the characteristic length ($L_u^*$) as in Equation (2.9). This length can be explained as the spatial wavelength of the gust in the $w$ direction and the value is 96.0395 m. This value is obtained in accordance to TSI, which established a reference height $z = 4$ m and roughness $z_0 = 0.07$ m.

$$f = \frac{n \cdot L_{uy}}{U_{\text{mean}}}$$ \hspace{1cm} (2.9)

The mean time constant can be got by integrating the PSD between the $n_1$ and $n_2$ limits presented in the previous paragraph:
\[ T = \frac{1}{2} \left[ \left( \frac{\int_{n_1}^{n_2} S(n) \, dn}{\int_{n_1}^{n_2} S(n) \, dn} \right)^{0.5} \right] \] (2.10)

Finally, the time constant of the gust \( T \), which is the maximum duration of the gust, is calculated with Equation (2.11).

\[ T = 4.1825 \cdot T \] (2.11)

Now, we are ready to calculate the non-dimensional wind velocity variations in the longitudinal direction \( (u_x) \) and in the lateral direction \( (u_y) \) of the component \( u \).

\[ u_x(\hat{x}) = f \cdot \hat{x} \cdot \cos(\beta_w) \cdot \frac{1}{U_{\text{mean}}} \] (2.12)

\[ u_y(\hat{x}) = f \cdot \hat{x} \cdot \sin(\beta_w) \cdot \frac{1}{U_{\text{mean}}} \] (2.13)

where the \( \hat{x} \) term in the equations is a function of the 'distance along the track towards the position of the maximum amplitude of the gust' that can be easily calculated by:

\[ \hat{x} = v \cdot (t - t_{\max}) \] (2.14)

being \( t_{\max} \) the instant of time at which the maximum of the Chinese Hat takes place. This means that \( t_{\max} \) should be chosen at the beginning of the calculation for being an input to the problem. Moreover, the factor \( f [s^{-1}] \) is:

\[ f = \frac{1}{2 \cdot T} \] (2.15)

Finally, we are ready to calculate the coherence function via linear interpolation since the \( u_x \) and \( u_y \) terms can now be introduced in Equation (2.3).

The specific case in which the wind is perpendicular to the track can be easily resumed here. If \( \beta_w = 90^\circ \), the value of \( u_x \) is zero and the coherence function is simplified to:

\[ C(\beta_w = 90^\circ) = \frac{1}{\sigma_u} \] (2.16)

Therefore, the wind speed normal in to the vehicle Equation (2.3) is simplified to:

\[ v(\beta_w = 90^\circ) = U_{\text{mean}} + 2.84 \cdot \sigma_u \cdot C(\beta_w = 90^\circ) \] (2.17)
A low-pass filter consisting in a moving spatial average, based on a window size equal to the vehicle length and step smaller than 0.5 m, will filter the wind gust before being introduced in the full wind scenario. The value of $U_{\text{max}}$ without being filtered is the wind velocity taken as reference to build the CWC. Another important point is the determination of $t_3$ and $t_5$ (see FIGURE 8), where the exponential function meets a function with the constant value $U_{\text{mean}}$. The standard assumes that the minimum value of the Chinese Hat and $U_{\text{mean}}$ are equal when the difference between these values is 1 per cent.

The standard eases the check of the Chinese Hat giving a full record of a calculation example before and after applying the window filter in Annex I, part 2 of the standard.

Vehicle designs different from to the standards such as the vehicles with Jacob bogies, articulated trains or any vehicle with transmission of the roll moment between cars do not require any special model of the Chinese Hat. Only one time-history of wind will be applied to both vehicles. Furthermore, only a single gust is considered with a time delay whose value depends on the train speed and the longitudinal distance between the geometrical centers. During the gust of wind, each car will be submitted to either the mean force or the gust. The window-type filter does change since the filtering process for each car should be done separately taking into account each one of the car lengths.

### 2.1.3 Simplified models

A dynamic model of the vehicle is needed to assess the wheel unloading factor of vehicles in a crosswind context. The standard considers several levels of complexity, which are from the simplest to the most complex model:

- A three-mass model.
- A five-mass model.
- A full multibody model.

#### a) A three-mass model

It is the simplest model proposed in the standard which considers it 'less precise but conservative because of built in margins'. The main advantage of this model is the simplicity of the modeling process; however, some suspension elements cannot be included since they have strictly a three-dimensional configuration.

There are some restrictions regarding the vehicle type that reduce the applicability of this model. The text specifies that 'the method is suitable only for vehicles with two
sets of running gear or two single axis' that indirectly excludes vehicles with Jacob bogies and articulated trains. Tilting trains can be reduced to three masses providing that the tilt effect is included in the mass distribution, the suspension characteristics and the CoG movements.

**FIGURE 10**: Definition of the three-mass model according to EN 14067-6:2010

**FIGURE 10** shows a schematic representation of the model, being \( m_0 \) the unsprung masses (usually the mass of the four wheelsets), \( m_1 \) the primary suspended masses (usually the mass of the two bogies) and \( m_2 \) the secondary suspended masses (usually the mass of the carbody). Furthermore, \( y_1 \) is the lateral displacement of the primary suspended masses’ CoG and \( y_2 \) is the lateral displacement of the secondary suspended masses’ CoG. The height of each mass is also shown in the figure, being \( z_{CoG,0} \) the height of the unsprung masses, \( z_{CoG,1} \) of the primary suspended masses and \( z_{CoG,2} \) of the secondary suspended masses. Finally, \( 2b_a = 1.5 \) m is the lateral contact spacing for track gauges of 1435 mm.

The characteristic wind speed that makes the wheel unloading ratio to reach 90 per cent is determined by calculating the equilibrium moment at the position of the leeward rail. Although the three-mass model can be easily introduced into a multibody software to be simulated, the standard describes an iterative method that may be enough.
Chapter 2: current regulation

The main equation is:

\[ \sum M = f_{20} \frac{1}{f_m} M_m + M_{CoG} + M_l = M_{x,lee} = 0 \]  \hspace{1cm} (2.18)

\( f_{20} \) represents the wheel unloading and its value is 0.9. \( f_m \) is a ‘method factor’ and ‘considers uncertainties in the method’. The value of such factor is given only for track gauge 1 435 mm and the value depends on the vehicle type, as TABLE 3 shows:

<table>
<thead>
<tr>
<th>( f_m )</th>
<th>Passenger vehicles</th>
<th>Freight wagons</th>
<th>Locomotives</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.2</td>
<td>1.15</td>
<td>1.2</td>
<td></td>
</tr>
</tbody>
</table>

TABLE 3: Method factor \( f_m \) for UIC track gauge for different types of vehicles

\( M_m \) is the restoring moment due to vehicle’s masses that is calculated as follows:

\[ \sum M_m = m \cdot g \cdot b \]  \hspace{1cm} (2.19)

being \( m \) the mass of the vehicle that is calculated as the addition of each independent mass, and \( g \) the gravity acceleration.

The moment due to the lateral movement of the center of gravity of the suspended masses (\( M_{CoG} \)) is defined as:

\[ M_{CoG} = \left( m \cdot y_1 + m \cdot y_2 \right) \cdot g \]  \hspace{1cm} (2.20)

\( M_l \) is the moment due to uncompensated lateral acceleration that uses the standard formula for the calculation of \( a_q \) for a curve of radius \( R_c \) and track cant \( h \):

\[ M_l = m \cdot a_q \cdot z_{CoG} \]  \hspace{1cm} (2.21)

\[ a_q = \frac{v^2}{R_c} - \frac{g \cdot h}{2 \cdot b} \]  \hspace{1cm} (2.22)

The aerodynamic moment induced by the wind is \( M_{x,lee} \). In order to proceed with the calculation, the aerodynamic coefficient \( C_{M_{x,lee}}(\beta) \) needs to be got from any CFD simulation or experimental test.

\[ M_{x,lee} = \frac{1}{2} \rho \cdot v_s^2 \cdot A_0 \cdot d_0 \cdot C_{M_{x,lee}}(\beta) \]  \hspace{1cm} (2.23)

where \( A_0 \) is the normalized area that is 10 m\(^2\) and \( d_0 \) the normalized length, which is 3 m. The relative wind speed is \( v_a \) which needs to be obtained in terms of Equation (2.24). Lastly, the yaw angle, \( \beta \), is got from Equation (2.25).
Chapter 2: current regulation

\[ v_a^2 = (v_w \cdot \cos \beta_w + v_y)^2 + (v_w \cdot \sin \beta_w)^2 \]  \hspace{1cm} (2.24)

\[ \beta = \arctan \left( \frac{v_w \cdot \sin \beta_w}{v_w + v_y \cdot \cos \beta_w} \right) \]  \hspace{1cm} (2.25)

b) A five-mass model. Advanced quasi-static method

This model is more complex that the three-mass model presented in the previous section though simpler than a full multibody model. The five-mass model is the simplest approach in the 3D space and it is mainly intended to perform quasi-static calculations. The five-mass models are proved to be conservative since the characteristic wind speeds obtained from calculations using this model are not higher than those that are got from full multibody models. The standard limits using this model to the same types of vehicles that were possible to been reduced to a three-mass model. Tilting trains can be studied through this method but introducing appropriately the tilt effect of the train.

The concept consists in dividing the vehicle into five masses connecting them with stiffness elements but without any damper (see Figure 11). The masses that represent the bogies and the carbody have five degrees of freedom each. Displacements along axis Y and Z, and turns around axis X, Y and Z are allowed. Both masses representing the wheelsets are fixed and cannot move or turn.

The five masses of the model need to be correctly parameterized which means that the data regarding the mass property and the CoG position should be accurately introduced. The model can include linear as well as non-linear stiffness characteristics, therefore lateral and vertical stops can also be considered.

Since the model is three-dimensional, the aerodynamic side and lift forces \((F_y, F_z)\) and all the aerodynamic moments should be introduced into the model. The drag force is not considered in any crosswind calculation no matter the model approach. The effect of the uncompensated acceleration, \(a_u\), can also be introduced as an additional lateral force.

\[ F_i = \frac{1}{2} A_i \cdot \rho \cdot v_i^2 \cdot C_{A_i}(\beta), i = y, z \]  \hspace{1cm} (2.26)

\[ M_i = \frac{1}{2} A_i \cdot d_i \cdot \rho \cdot v_i^2 \cdot C_{A_i}(\beta), i = x, y, z \]  \hspace{1cm} (2.27)

The relative wind velocity should be calculated according to Equation (2.24) and the yaw angle according to Equation (2.25).
FIGURE 11: The five-mass model according to EN 14067-6:2010. The bodies are WS (wheelset), BG (bogie) and CB (carbody). The stiffnesses are $C_{pij}$ (primary suspension), $C_{sij}$ (secondary suspension) and ARB (anti-roll bar) where the subindex $i$ stands for position of the element and $j$ stands for the side of the vehicle and finally, $Q_{ij}$ are the vertical forces at the contact.

One advantage of this model over the three-mass model is that it lets one take into account the lateral displacement of the contact point. The standard establishes the maximum displacement of the point, $y_B$, which is quantified to be 28 mm. Therefore, $Q$-forces are not equidistantly introduced but repositioned according to FIGURE 12.

The lateral contact spacing for a UIC standard track gauge is 1 500 mm:
Therefore, \( b_{A,\text{min}} \) is calculated with Equation (2.29) since the contact point of the wheel located at leeward is displaced 28 mm:

\[
b_{A,\text{min}} = b_A - y_b
\]

The standard recommends comparing the sums of the wheelset loads in each bogie and the model suspension coefficient, \( s \), in order to validate the model. The maximum allowed difference between measurements and simulations in the case of the wheelset loads is 1 per cent. In regard to the suspension coefficient, the standard states that the simulation results ‘shall lie within the tolerances of the measured values’. Finally, the standard specifies that the bumpstop shall be correctly modelled including the non-linear stiffness characteristic.

The model can be solved either by using a multibody software or by explicit formulation. In the authors’ opinion, the derivation of the system of equations (available in the Annex H of the standard) can be a great deal whereas creating the model with a multibody software is rather simple. Annex H of the standard also includes two examples of CWC calculations in order to check whether the model is properly set up. These examples are also used to validate the five-mass model when the difference between the calculated CWC and the result proposed in the standard has a maximum difference of ± 0.5 m/s measured at \( \beta_w = 90^\circ \).

c) A multibody model. Time-dependent simulations

This main goal of this model is to perform time-dependent simulations, mainly to assess CWC curves by introducing the Chinese Hat wind scenario that was presented in Section 2.1.2.

The most critical vehicle should be modeled in the worst operational scenario which typically is the case of empty car. However, the standard adds that in the cases with restraints at the coupling between adjacent vehicles, the contiguous cars should be modeled too. In regard to the mass property, the standard specifies that ‘a check shall be made that an even distribution of passengers is not more critical than an empty vehicle’.

The election of the track gauge, the rail profile and the rail inclination depends on the operation of the vehicle. For the assessment of interoperable vehicles, the standard establishes a UIC gauge, rail profile 60E2, new wheel profile and the worst case against of 1/20 and 1/40 rail inclinations. The calculations require the specific parameters of the track on which the train will run for any other vehicles.
The modeling requirements of the vehicle are rather flexible and the standard only states the minimum requirements. Any device that may have effect on the overturning of the vehicle should be included, in addition to the relevant parts of the vehicle (bogies, wheelsets, carbody, ...) and the suspension elements (stiffnesses, dampers, bumpstops, ...). The wheel/rail contact model should be able to feature 'nonlinear contact geometry and creep force/creep relation'.

The model needs to pass the validation process that matches with the one of the five-mass model that is fully described in Section 2.1.3b).

Wind forces and moments are described in Equation (2.26) and (2.27). The integration method must be able to calculate the step corresponding to the maximum value of the wind gust. In order to meet this requirement, the standard recommends that the output step size of the calculation shall be lower than 1/30 s.

The standard presents a broader formula for the wheel unloading calculation which takes into account that Q-forces distributes evenly between the front and the rear wheelset of a bogie:

\[
\frac{\Delta Q}{Q_0} = 1 - \frac{Q_i}{2Q_0} - \frac{Q_j}{2Q_0}
\]  

(2.30)

where \(Q_0\) are the Q-forces for the empty, unloaded vehicle and without any excitation. The Q-forces of the unloaded wheel of the first wheelset in the bogie are \(Q_i\), and \(Q_j\) are the Q-forces of the unloaded wheel of the second wheelset in the bogie.

The time signal of the wheel unloading ratio should be post-processed with a low-pass filter to avoid high frequency distortions at the contact. The standard suggests a 4th order Butterworth filter with an upper corner frequency of 2 Hz; however, any similar filter could also be used.

### 2.2 Ground and track configurations

The section of the standard describing ground configurations is somewhat a list of several topologies that represent most of the possible grounds. Other platforms such as viaducts or bridges are excluded due to the unfeasibility to establish any standardized geometry. EN 14067-6:2010 gathers several ground scenarios that are also presented in other standards currently available in Europe.

The significance of the ground scenario comes from the fact that they have a direct influence on the flow around the vehicle, hence the aerodynamic coefficients of the vehicle substantially change. In order to carry out wind tunnel tests or simulations one infrastructure model needs firstly to be chosen among the available alternatives.
2.2.1 Single track and ballast rail

The CEN standard proposes the scenario represented in FIGURE 13 as reference unlike the previous proposals of the TSI, which are the flat ground with gap (Section 2.2.2) and the 6 m embankment (Section 2.2.4). Therefore, experimental tests on wind tunnel based using this scenario are required to compute the vehicle’s CWC for validation purposes. Thus, other grounds available in the standard are optional configurations without any compulsory effect.

The aerodynamic coefficients of the vehicle in other scenarios can be estimated from the results in this scenario. This scenario is also the benchmark reference for CFD calculations or wind tunnel experiments.

![FIGURE 13: Single track and ballast scenario](image)

2.2.2 Flat ground with gap

This is the simplest configuration since the representation of the track and ballast is not included. This configuration consists on elevating the train to leave a gap of 235 mm (see FIGURE 14), measured from the floor to the lowest point of wheels. The clearance is approximately the distance measured from the position of top of the rail (T.O.R) to the ground. This ground scenario is taken from the TSI, which establishes this configuration as a requirement for the validation of vehicles.
2.2.3 Double track ballast and rails

This ground scenario with two tracks includes both directions of travel, which adds two load cases: the windward and the leeward track. The dimensions of the geometry are given in FIGURE 15 without including the tolerances allowed by the standard. The vehicle should be placed on the rails without leaving any gap between the track and the vehicle model.

2.2.4 Standard embankment of a 6 m height

The ground configuration consists on a 6-meter high standard embankment (see FIGURE 16) that includes the representation of the double track and ballast of FIGURE 15 on the top of the model. The standard make special emphasis on the slope of the embankment that has to be 2/3 since the value of the wind speed at the top of the embankment is directly influenced by this parameter.
The alternative to perform tests in a wind tunnel is to approach the coefficients on the embankment from data from other scenarios. However, one must be aware that there is currently a discrepancy on the way of estimating the overspeed effect. The open question is about how to calculate the wind speed at the top of the embankment taking into account that the own infrastructure accelerates the wind flow. In order to save costs, EN 14067-6:2010 suggests to use the single track ballast and rail coefficients for the 6 m embankment following the so-called Baker’s hypothesis. The proposal consists in modifying the wind speed and the wind angle by scaling them with an overspeeding factor.

Baker, 1985 exposed that only the component normal to the embankment gets accelerated over the embankment. Thus, the component parallel to the embankment remains unchanged and the local wind angle on the top of the embankment (β) changes.

Following the notation presented in FIGURE 17, the wind velocity on top of the embankment will be:

$$V_{emb} = \sqrt{\left(\frac{v}{v} + U \cdot \cos \beta\right)^2 + \left(U \cdot \sin \beta\right)^2}$$

(2.31)

and the wind angle on top of the embankment is:

$$\beta = \arctan \left( \frac{f_{emb} \cdot U \cdot \sin \beta}{v + U \cdot \cos \beta} \right)$$

(2.32)
where $f_{emb}$ is the overspeeding factor. EN 14067-6:2010 refers to ‘an adequate source’ such as EN 1991-1-4 (CEN, 2005), which is the part of the Eurocode dealing with wind actions, however further information can be found in Section 2.3.2.

![Diagram of components of wind near and on top of the embankment](image)

**FIGURE 17:** Components of wind near and on top of the embankment

### 2.3 Other regulations to guarantee safety

The overview of EN 14067-6:2010 tried to make easier the reading of the text and to explain those aspects that were not clear enough. The final purpose of the normative is rather clear: to come out with a simple criterion that guarantees the safety of the vehicle. This criterion, the CWC of the vehicle, is built by calculating the pairs of wind and vehicle speed that lead to the wheel unloading ratio limit. Once the CWC of the vehicle is built, following a method from those presented in Section 2.1, the following question arises: what is the level of accordance with the other co-existing standards?

The CEN standard is not the only normative that provides information to perform crosswind studies. The following standards also regulate the behavior of the vehicle against crosswinds:

- The DB guideline Ril 80704 (DB, 2006) which is a requirement by the German railway authority EBA for German rolling stock assessment.
- The RSSB standard GM/RC2542 (RSSB, 2009). This standard gives different recommendations for trains running with standard cant deficiency and for trains circulating with larger values of cant deficiency. In order to make the
Chapter 2: current regulation

explanation of the topic clear, the following text only refers to the standard values of cant deficiency.

Vehicles should follow the TSI standard when trains are going to travel between countries in Europe (interoperate). On the contrary, the CEN standard applies to all trains since it is a requirement to pass the certification process in any country in the European Union. Lastly, the DB guideline and RSSB standard are local standards just for trains circulating in Germany and in the UK, respectively. Usually, TSI standards are quite generalist and they do not get into detail, however this is not the case of the standard related to crosswinds. Since the CEN standard is relatively new, the TSI filled the absence of any standard dedicated to crosswinds by going into detail, i.e. by formulating the Chinese Hat scenario (see Section 2.1.2) or by listing the ground scenarios in order to carry out the wind tunnel tests (see Section 2.2). In fact, most of this CEN standard is a reformulation of the guidance already present in the TSI including a few updates.

The CEN standard that has recently been developed focuses on common definitions and descriptions of the aerodynamic phenomena and measurement procedures. Due to the application to all types of rail traffic standards have not converged to one method per phenomenon but allows variations that arise from national requirements. The European project AeroTRAIN, which is part of the TrioTRAIN cluster of projects, deals with key railway interoperability issues. The objective of these projects is to propose an innovative methodology that will ease rail vehicle certification process in Europe to become a faster, cheaper and better process for all involved stakeholders. The overall goal of the project AeroTRAIN is to promote interoperable rail traffic in Europe by reducing costs and time of certification and closing ‘open points’ in the TSI's.

### 2.3.1 A comparison of some important requirements

In the Annex A of the EN 14067-6:2010 there are a couple of tables that compare the requirements of these three standards in reference to certain topics of the crosswind assessment, during both aerodynamics and dynamics stages. This fact shows that limits are arbitrary most of the times, and more than one approach is valid to guarantee safety. The purpose of this section is to compare these different approaches, emphasizing those themes in which there is a slight disagreement between the standards.

Firstly, we need to know which is the type of vehicle at which the standard applies, since it appears that each standard has a different application range. TSI applies to those high-speed trains with velocity greater than 250 km/h, DB RII to all passenger trains with speed higher than 140 km/h and RSSB to all passenger and freight trains.
Chapter 2: current regulation

CEN asks to all the passenger vehicles with velocity up to 360 km/h and freight vehicles with velocity up to 160 km/h to fulfill the requirements within the standard.

Each of the ground scenarios presented in Section 2.2 would require the calculation of a specific CWC curve since the aerodynamic coefficients of the vehicle change. However, testing vehicles in a wind tunnel at all the ground scenarios is not affordable due to costs and time restrictions. Standards overcome this limitation by defining the minimum scenarios at which the vehicles should be tested. However, standards disagree on the definition of the ground scenarios to experiment for building the vehicle's CWC. TSI asks to obtain the CWC curves in the flat ground with 235 mm gap (see Section 2.2.2) and in the 6 m embankment (see Section 2.2.4). DB Ril only asks to measure the coefficients in the simplest scenario, which is the flat ground with 235 mm gap, and unexpectedly the RSSB standard does not even mention this topic. The approach of the CEN standard consists on defining a new ground scenario that is not present in any other standard that is the single track ballast and rail (see Section 2.2.1).

The approaches to wind tunnel experiments and CFD simulations are mostly different, founding the biggest discrepancies in the specifications of the blockage factor, the Reynolds number and the symmetry check.

Blockage establishes indirectly either the size of the model or the wind tunnel dimensions. The bigger the blockage, the higher will be the values of the coefficients and the disparity with the true coefficients; nevertheless, it was found that having bigger coefficients leads to conservative results. In this regard, TSI allows a ratio lower than 10 per cent. DB Ril allows up to 15 per cent but introducing corrections in the bracket between 5 and 15, which is also a CEN statement. RSSB combines CEN and DB specifications, since the administration allows having blockages ratios up to 15, but applying the corresponding corrections.

Reynolds number has a relative importance since the measurements should be independent of the flow. RSSB and CEN agrees to set the inferior limit in 250 000, which is increased up to 600 000 in DB Ril. Surprisingly, TSI does not include a criterion in relation to this aspect.

The most specific standard concerning symmetry check is DB Ril. The German normative asks to check the differences in $C_{Fy}$, $C_{Fx}$ and $C_{Mx}$ and make sure that they are smaller than 5 per cent or 0.1 at several yaw angles: $0^\circ$, $20^\circ$, $30^\circ$ and $40^\circ$. TSI just says that the check is 'needed' and RSSB does not mention this topic. CEN is quite ambiguous since it states that 'check measurements should be performed with both positive and negative yaw angles' without specifying which angles should be checked.

In general terms, there is a better agreement in the dynamical stage between the CEN, the TSI and the DB Ril standard as the RSSB does not have any recommendation.
for this respect. Regarding the allowed vehicle models, the three standards agree that the time-dependent simulations with full multibody models are the best approach to represent the real behavior of trains under crosswinds. The three of them require that the Chinese Hat wind scenario should be introduced in the multibody models. Simplified models are allowed by the CEN and DB Ril standards in order to perform quasi-static simulations by introducing steady wind loads. Nevertheless, DB Ril only considers as valid a five-mass model of the vehicle whereas the CEN standard states that a simplified three-mass model is able to provide accurate enough results.

The checklist to verify the model of the CEN is taken from the TSI which makes it to be identical. DB Ril could be considered to be more flexible in regard to this point since it does not requires to check both the suspension coefficient and the position of the center of gravity.

2.3.2 Other approaches to the embankment scenario

Authors and standards have proposed alternatives to obtain the aerodynamic coefficients of the vehicle approaching the problem in different ways. The calculation the aerodynamic coefficients of the vehicle on this configuration could be confusing since TSI and EN 14067-6:2010 do not agree in which methodology should be followed. In addition, TSI asks for the vehicle’s CWC on this ground scenario as part of the validation process, needing to measure the vehicle’s aerodynamic coefficients on the embankment.

a) TSI and DB Ril proposals

The current version of the TSI asks for the measurement of the coefficients of the vehicle on a 6 m embankment through wind tunnel tests, a methodology that is also followed by SNFC. If the coefficients are measured with the vehicle on top of the embankment, the following formulae apply. This approach also considers valid the theory by Baker, in which only the speed component perpendicular to the embankment is accelerated changing the wind angle on top of the embankment.

\[
\begin{align*}
V_s(t) &= \sqrt{V_v^2 + V_w(t) \cos \beta(t)^2 + (C(t) \cdot V_w(t) \sin \beta(t))^2} \\
\beta(t) &= \arctan \left( \frac{C(t) \cdot V_w(t) \sin \beta(t)}{V_v + V_w(t) \cos \beta(t)} \right) \\
C(t) &= \frac{C_S - 1 + G(t)}{C_S - G(t)}
\end{align*}
\] (2.33, 2.34, 2.35)
where $C_{SV} = 1.2416$ for the windward case and $C_{SV} = 1.1705$ for the leeward case. $G(t)$ is the gust factor that is calculated by diving the instantaneous wind speed of the Chinese Hat by the mean wind speed.

The Deutsche Bahn in the Guidline Ril 80704 (DB, 2006) proposes to use a very similar approach to the one adopted in the CEN standard. The methodology and the justification between both standard matches, being the reference ground that sets the difference. As the German standard only requires doing tests with the flat ground, this scenario must provide the coefficients for the embankment. The method to calculate the new wind speed does not change, thus the methodology in Section 2.2.4 still applies.

b) The DEUFRAKO proposal

The participants of the European project DEUFRAKO, Aerodynamics in open air (AOA, 2008) developed a formula that modifies the aerodynamic loads of the flat ground to the embankment scenario by scaling the wind speed. This formulation is more complex and it is based in parameters such as the geometry of the embankment, the width of the train and the position on the embankment. One of the main advantages of the method is the possibility of considering the vehicle on both the windward and the leeward track.

c) Another alternative. Scaling the aerodynamic coefficients

The aerodynamic coefficients are the parameters that should be scaled following other methodologies. The method consists on working out Equation (2.31) and Equation (2.32) as in Schober et al., 2010.

Firstly, the train speed is set to zero since there is no relative movement between the vehicle and the wind tunnel floor. In such way, Equation (2.31) and Equation (2.32) are reduced to:

$$\frac{V_{emb}}{U} = \sqrt{(\cos\beta_m)^2 + f_{emb}^2 (\sin\beta_m)^2}$$  \hspace{1cm} (2.36)

$$f_{emb} = \frac{\tan\beta_m}{\tan\beta_m}$$  \hspace{1cm} (2.37)

In this case, the method assumes that both the wind speed and the wind angle do not change from the Flat Ground (FG) configuration to the embankment:

$$\beta_m = \beta$$  \hspace{1cm} (2.38)
Equation (2.36) can be rewritten using $\beta$ as an argument:

$$\frac{U}{V_{emb}} = \sqrt{(\cos^2 \beta) + f_{emb}^2 (\sin^2 \beta)}$$

Equation (2.39) can be rewritten using $\beta$ as an argument:

$$U = V_{emb}$$

If the assumptions in Equation (2.38) and (2.39) are valid, we are assuming that the wind loads the flat ground are the same on the embankment. Therefore, we can calculate the aerodynamic coefficients on the embankment by applying this relation into the forces and moments equations:

$$C_{i,emb}(\beta) = \frac{C_{i,FG}(\beta)}{\cos^2 \beta + f_{emb}^2 \sin^2 \beta}$$

According to Baker’s hypothesis, the method is mostly valid only for $\beta \geq 30^\circ$, which reduces the applicability of the method. The reason is that the critical wind angles in the crosswinds over high-speed trains topic are always in the range between 0 degrees and 30 degrees (Orellano and Schober, 2006).

d) The calculation of the $f_{emb}$ factor

The $f_{emb}$ factor can be got in many different ways, being as many approaches to calculate the embankment factor as there was to estimate the overspeeding effect of wind. Although there are different approaches such as the one presented in Diedrichs et al., 2007 the most extended is the one based in the Eurocode (CEN, 2005), which is followed by the German guideline and suggested by the EN standard.

TABLE 4 is taken from Herb et al., 2007 in which the authors calculated the $f_{emb}$ for different embankment heights $(r_c)$ for the slope value (0.67) set in the CEN standard and the TSI.

Diedrichs et al., 2007 presented a formulae based on the potential flow theory around a cylinder. The embankment factor could be calculated following Equation (2.42) whenever the wind is perpendicular to the embankment. Equation (2.43) is more general and valid to calculate $f_{emb}$ at the whole range of wind angles.

$$f_{emb,90^\circ} = 1 + \left(\frac{r_c}{z}\right)$$

$$f_{emb}(\beta) = \sqrt{\cos^2 \beta_{emb} + f_{emb,90^\circ}^2 \sin^2 \beta_{emb}}$$
where \( r_0 \) is the height of the embankment and \( z \) is the elevation of the top of the train measured from the top of the embankment.

<table>
<thead>
<tr>
<th>( r_0 ) [m]</th>
<th>( f_{emb} )</th>
<th>( r_0 ) [m]</th>
<th>( f_{emb} )</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>1.07</td>
<td>18</td>
<td>1.46</td>
</tr>
<tr>
<td>4</td>
<td>1.14</td>
<td>20</td>
<td>1.48</td>
</tr>
<tr>
<td>6</td>
<td>1.23</td>
<td>22</td>
<td>1.50</td>
</tr>
<tr>
<td>8</td>
<td>1.29</td>
<td>24</td>
<td>1.51</td>
</tr>
<tr>
<td>10</td>
<td>1.34</td>
<td>26</td>
<td>1.52</td>
</tr>
<tr>
<td>12</td>
<td>1.39</td>
<td>28</td>
<td>1.53</td>
</tr>
<tr>
<td>14</td>
<td>1.42</td>
<td>30</td>
<td>1.53</td>
</tr>
<tr>
<td>16</td>
<td>1.44</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

*TABLE 4: Results from the calculation of \( f_{emb} \) following the Eurocode steps for embankments of different heights and slope 0.67 taken from Herb et al., 2007*
Chapter 3

vehicle models

This section presents the vehicle models that were used to study dynamics and aerodynamics of high-speed trains under crosswinds, including the verifications and/or the validations that were carried out.

Two dynamic models of the same vehicle were needed to analyse the vehicle dynamics on the crosswind context, and to check whether 2D models are valid tools to calculate CWCs.

The study of the vehicle aerodynamics was divided in two stages. Firstly, we used a two-dimensional model of a high-speed vehicle to do a parametric study of the fence design, comparing results from experimental tests and numerical simulations. Then, we performed numerical simulations in 3D, for which we used the ICE 3 model provided by the standard EN 14067:6:2010. The sections that cover the aerodynamic models contain the description of the vehicle model and the configuration of the computational domain to carry the CFD calculations.

3.1 Multibody models to study vehicle dynamics

The parameters have been estimated in order to have a ‘generic high speed vehicle’, but do not exactly correspond to any existing train. The model represented a standard train, with two independent bogies per car and distributed traction along the train. Furthermore, there was no roll coupling between adjacent cars in this type of vehicles, only the leading car was modeled for being known as the most critical vehicle.

Two different models were used to study the problem: a simplified three-mass model and a complete multibody model of the same vehicle. The 2D model was a simplification of the full model, which included a complete representation of the suspension and the interaction between wheel and rail.
Both models were built and numerically solved with the commercial multibody software SIMPACK (SIMPACK AG, 2008). Most of the suspension components were modeled as compact elements to make the model simpler. If any component was modeled as Point-to-Point element (PtP), a clarification will be made.

### 3.1.1 Full multibody model

The full vehicle model had 23 bodies and 44 degrees of freedom. The bodies masses and inertias are listed in TABLE 5 and the full set of suspension parameters is available at TABLE 7. The aerodynamic forces have been calculated using the coefficients provided by the standard annex for an ICE 3 train, which are represented in FIGURE 18. The carbody, bogies and wheelsets each had six degrees of freedom. The primary suspension consisted of vertical and lateral linear springs and dampers. The secondary suspension was built up in vertical, lateral and longitudinal directions; linear and non-linear springs and dampers were taken into account. In addition, torsion springs were added to the full model. The model included the standard S1002 wheel profile and UIC60 with an inclination of 1:40 rail profile. The wheel-rail forces were calculated by means of Kalker’s FASTSIM algorithm.

<table>
<thead>
<tr>
<th>Body</th>
<th>Mass [kg]</th>
<th>Ix [kg·m²]</th>
<th>Iy [kg·m²]</th>
<th>Iz [kg·m²]</th>
<th>CoG’s height [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wheelsets</td>
<td>1 800</td>
<td>1 000</td>
<td>100</td>
<td>1 000</td>
<td>− 0.46</td>
</tr>
<tr>
<td>Bogies</td>
<td>3 500</td>
<td>560</td>
<td>315</td>
<td>1 715</td>
<td>− 0.5</td>
</tr>
<tr>
<td>Carbody</td>
<td>53 500</td>
<td>70 000</td>
<td>72 · Mass</td>
<td>72 · Mass</td>
<td>− 1.4</td>
</tr>
</tbody>
</table>

Distance between wheelsets of the same bogie: 2.5 m
Distance between bogies centers: 17.375 m

TABLE 5: Mass and inertial parameters of the multibody model

Bumpstops had a non-linear stiffness characteristic (see FIGURE 19) created from discrete points and a linear interpolation. TABLE 6 provides the positive terms since the stiffness characteristic is symmetric with respect the origin of coordinates.
Chapter 3: vehicle models

FIGURE 18: Multibody model of a generic high-speed train.

FIGURE 19: Stiffness characteristic of the bumpstops

<table>
<thead>
<tr>
<th>Deformation [mm]</th>
<th>Stiffness [N/m]</th>
<th>Deformation [mm]</th>
<th>Stiffness [N/m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>0</td>
<td>45</td>
<td>6 870</td>
</tr>
<tr>
<td>25</td>
<td>0</td>
<td>50</td>
<td>11 580</td>
</tr>
<tr>
<td>30</td>
<td>600</td>
<td>55</td>
<td>17 170</td>
</tr>
<tr>
<td>35</td>
<td>1 760</td>
<td>60</td>
<td>29 200</td>
</tr>
<tr>
<td>40</td>
<td>3 730</td>
<td>65</td>
<td>230 000</td>
</tr>
</tbody>
</table>

TABLE 6: Stiffness characteristic of the bumpstop. Values of positive terms
### 3.1.2 Simplified two-dimensional model

#### a) Description

Although this model was also a ‘multibody model’ because simulations were carried out with multi-body code, in this thesis we called it a ‘simplified three-mass model’ as it was simplified to 2D. The vehicle model had three masses (the carbody, another mass acting as both bogie frames, and another mass representing all the wheelsets of the vehicle) and three degrees of freedom (lateral, vertical and roll) as
result of a simplification from the multi-body vehicle model, which was explained in the Section 3.1.1.

![Schematic representation of the two-dimensional model. Description of the masses, suspension elements, Q-forces and aerodynamic loads](image)

The vehicle had three masses. One mass stood for the carbody of the vehicle, another mass represented both bogies, and finally a mass represented the four wheelsets of the vehicle. Each of these masses had three degrees of freedom (vertical, lateral and roll), whose values and coordinates of the center of gravity are shown in TABLE 8.

<table>
<thead>
<tr>
<th>Body</th>
<th>Mass [kg]</th>
<th>Ix [kg·m²]</th>
<th>Y-coordinate of CoG [m]</th>
<th>Z-coordinate of CoG [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Wheelsets</td>
<td>7 200</td>
<td>4 000</td>
<td>0</td>
<td>−1.4</td>
</tr>
<tr>
<td>Bogies</td>
<td>7 000</td>
<td>1 120</td>
<td>0</td>
<td>−0.5</td>
</tr>
<tr>
<td>Carbody</td>
<td>53 500</td>
<td>70 000</td>
<td>0</td>
<td>−0.46</td>
</tr>
</tbody>
</table>

TABLE 8: Masses, inertias and coordinates of the CoG used in the simplified model

The suspension elements included in the model were limited to the minimum number. All of them are in the Y-Z plane. The secondary suspension, which acts between the carbody and the bogie frame, was composed of linear springs in both vertical and lateral directions. It also had a linear anti-roll stiffness that transfers momentum around the X axis. In addition, nonlinear lateral bumpstops were taken into account and were modeled as progressive reaction forces, though they were not included in FIGURE 20. The primary suspension, which connects the bogie frames and the wheelsets, was modeled with linear springs in both lateral and vertical directions.
Contact force between the wheel and the rail was modeled with a linear spring which acts only in the compression direction; stiffness and damping terms will be taken from the multi-body model so the total contact force computation is the same in both studies. It is important to note that the same value for lateral and vertical contact stiffness is introduced to the model with the purpose of having the same rigidity in all directions; the same thing applies to damping. The contact stiffness was considered only for the purpose of representing the same joint between the wheelset and the ground of the full model. Another possibility is to consider an infinite stiffness at the contact. However, considering the same stiffness of the multi-body model doesn’t involve any difficulty, and it represents the full model more faithfully.

<table>
<thead>
<tr>
<th>Suspension element</th>
<th>$k_y$ [N/m]</th>
<th>$k_z$ [N/m]</th>
<th>$c_y$ [N/m·s]</th>
<th>$c_z$ [N/m·s]</th>
<th>$k_α$ [N/rad·m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary</td>
<td>$5 \times 10^7$</td>
<td>$4.8 \times 10^5$</td>
<td>$9 \times 10^3$</td>
<td>$2.8 \times 10^4$</td>
<td></td>
</tr>
<tr>
<td>Secondary</td>
<td>$4.8 \times 10^5$</td>
<td>$7 \times 10^5$</td>
<td>$6 \times 10^4$</td>
<td>$4 \times 10^4$</td>
<td>$3 \times 10^6$</td>
</tr>
<tr>
<td>Anti-roll</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>W/R contact</td>
<td>$2 \times 10^9$</td>
<td>$2 \times 10^9$</td>
<td>$1 \times 10^5$</td>
<td>$1 \times 10^5$</td>
<td></td>
</tr>
</tbody>
</table>

TABLE 9: Suspension parameters of the two-dimensional model

Damping in the wheel/rail contact and primary and secondary suspension were also taken into account. Stiffness and damper values for all the suspension elements (modeled as compact elements) are shown in TABLE 9 and the coordinates of the compact elements in TABLE 10.

<table>
<thead>
<tr>
<th>Suspension element</th>
<th>Y-coordinate [m]</th>
<th>Z-coordinate [m]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary</td>
<td>± 1</td>
<td>− 0.56</td>
</tr>
<tr>
<td>Secondary</td>
<td>± 1</td>
<td>− 0.8</td>
</tr>
<tr>
<td>Anti-roll</td>
<td>0</td>
<td>− 0.3</td>
</tr>
<tr>
<td>Bumpstop</td>
<td>0</td>
<td>− 0.55</td>
</tr>
<tr>
<td>W/R contact</td>
<td>0.75</td>
<td>0</td>
</tr>
</tbody>
</table>

TABLE 10: Coordinates of the suspension elements of the simplified model

b) Verification of the model

In order to verify that the simplified model parameters are correctly set, a comparison was made regarding vibration modes and sway behaviour. This also demonstrates that the 2D movements of the vehicle as a reaction to aerodynamic loads are equivalent for both models.
TABLE 11 show a comparison between the two models that have already been shown in sections 3.1.1 and 3.1.2. Obviously, only the modes which are in the Y-Z plane can be compared.

Results show that the modes that play a role in the characteristic movement of the vehicle when subjected to crosswinds are similar: natural damping as well as resonance frequencies agree.

<table>
<thead>
<tr>
<th>Mode</th>
<th>Multibody model</th>
<th>Simplified model</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Damping</td>
<td>Freq. [Hz]</td>
</tr>
<tr>
<td>Inferior sway of the carbody</td>
<td>0.18</td>
<td>0.49</td>
</tr>
<tr>
<td>Vertical of the carbody</td>
<td>0.12</td>
<td>0.75</td>
</tr>
<tr>
<td>Superior sway of the carbody</td>
<td>0.15</td>
<td>1.23</td>
</tr>
</tbody>
</table>

TABLE 11: Comparison of the vibration modes between the models of the vehicle

Sway simulation was carried out in order to verify whether the simplified model reacts like the multi-body model does when it is exposed to external forces, which involve turns in the bodies. This is a consequence of how crosswinds influence the dynamic of the vehicle: aerodynamic forces primarily cause the vehicle to turn its carbody, so the way it reacts to that kind of disturbance should be verified. The sway test coefficient was calculated in terms of Equation (3.1):

\[ s = \frac{\theta_{\text{carbody}} - \theta_{\text{mechan}}}{\theta_{\text{carbody}}} \]  (3.1)

Results of the comparison can be observed in FIGURE 21 and FIGURE 22: Relative error of the sway test. The simulation consists of the calculation of the sway coefficient for different values of superelavation, from 25 mm to 225 mm, with an increment of 5 mm; other values were interpolated. Although the curves diverge for superelavation values greater than 200 mm, the difference between them does not reach 1 per cent.

If the sway test coefficients agree for different cants, then the value of the rotating degree of freedom around the X axis of both models also agree as deduced from the sway test coefficient formula.

It can be concluded that both models achieve similar results and the simulations with the simplest model could provide sufficient precision.
Chapter 3: vehicle models

3.2 Models to study vehicle aerodynamics

The study of vehicle aerodynamics requires the external shape of the vehicle. The model should represent the actual geometry of the vehicle since the level of accuracy has a great influence on the representation of the flow problem.

Firstly, a simplified two-dimensional approach analyzed the scope of two-dimensional simulations when a parametric study of the fence design were needed. The 2D model represented the cross-sectional area of a vehicle standing on a bridge so that CFD could be also tested in the simulation of complex phenomena. Secondly, a three-dimensional representation of an ICE 3 train was used to validate a CFD method.
based on the Lattice-Boltzman Method, and to study the effect of fences in the 3D space.

The computer used to perform the aerodynamic studies, both the two-dimensional and three-dimensional, had 96 GB of RAM memory and 4 processor with 12 cores each.

3.2.1 A 2D model. Computational model and mock-up of the vehicle.

The vehicle was the RENFE Class 120 from CAF. The vehicle was differently modelled for doing tests in the wind tunnel and simulations with CFD. Each version had different characteristics so detailed information regarding simplifications and the related descriptions of the simulations and the tests are specified in self-dedicated sections.

a) Geometry of the computational model and domain discretization

The high-speed vehicle used in this study was the RENFE Class 120 from the Spanish train manufacturer CAF. After being pre-processed in CAD software, the real cross-sectional shape of a passenger coach and a typical section of a bridge were introduced into the computational model in real scale (see FIGURE 23).

Since the aim of the paper was to analyze the influence of solid fences, it was necessary to parameterize them by height, \( h_f \), and the length of the eave, \( l_e \). Four fence heights were tested: \( h_f = 1250 \text{ mm}, 1750 \text{ mm}, 2250 \text{ mm} \) and \( 2750 \text{ mm} \). Three eave lengths were also considered: no eave, \( l_e = 500 \text{ mm} \) and \( 750 \text{ mm} \). In addition to the scenario where no fence is installed, a total of twelve different fence designs were considered.

The train was located at the critical side, i.e. the windward side, of a double track bridge since that is the side where the train is most exposed to wind. At the leeward side, the shielding effect of the fences increases and so wind loads over the vehicle are lower (Barcala and Meseguer, 2007).

The aerodynamic loads per unit length, drag force \( (D) \), lift force \( (L) \) and roll moment \( (M_0) \) are calculated at the center of the carbody (see FIGURE 23). \( M_0 \) is the moment around the leeward rail, which measures the turnover of the vehicle caused by crosswinds loads. It assesses the efficiency of the parapets against overturning in one coefficient since it takes into account the contribution of all the aerodynamic loads. The vehicle will be at risk of overturning when the restoring moment due to the vehicle's mass does not keep the vehicle in equilibrium.
Chapter 3: vehicle models

FIGURE 23: Cross-section of the vehicle standing on a bridge along with the dimensions of the vehicle (H stands for height and W for width) and the bridge. The parameters of the fences, height (h_f) and eave length (l_e) are also given. The wind loads are drag force (D), lift force (L) and roll moment (M_v). The moment around the leeward rail is M_v.

The dimensionless aerodynamic force coefficients, $C_d$ and $C_l$, and the moment coefficient, $C_{m0}$, are calculated according to the standard fluid dynamics formulae. For these coefficients, the train's width (W) and height (H) are used:

$$C_d = \frac{D}{0.5 \cdot \rho \cdot v^2 \cdot H}$$  
(3.2)

$$C_l = \frac{L}{0.5 \cdot \rho \cdot v^2 \cdot H}$$  
(3.3)

$$C_{m0} = \frac{M_v}{0.5 \cdot \rho \cdot v^2 \cdot H \cdot W}$$  
(3.4)

where $v_\infty$ is the free flow air velocity and $\rho$ the air density.

The Reynolds number ($Re = U_\infty \cdot L_c / \nu$) was calculated with the kinematic viscosity of the fluid, $\nu$ (see Section 2.1.3) and by considering as the characteristic length, $L_c$, the distance from the top of the coach to the bridge base (8.5 m). The Strouhal number, which is a dimensionless number to describe the oscillations of the flow, was used to analyze the vortex shedding. It is defined as $St = f \cdot L_c / v_\infty$, where $f$ is the frequency of the vortex shedding.

The moment coefficient around the leeward rail ($M_v$) is useful to assess whether the fence is effective since it takes into account all the forces. Fence efficiency could be
evaluated just by analyzing the trend of this moment. It is calculated doing a force and moment equilibrium at that point:

\[
C_m = \frac{1}{2} C_d \frac{H}{W} + \frac{1}{2} C_l \frac{b_0}{W} - C_{m0}
\]  

(3.5)

where \( b_0 \) is the distance between contact points (1.5 m). An analysis of the equation shows that the effect of the roll moment is relatively small.

The computational domain was two-dimensional and represented by a rectangle (Figure 24). To reproduce wind tunnel tests, a uniform profile was imposed at the inlet boundary. Depending on the simulation type, the velocity value can be 0.22 m/s \((\text{Re} = 1 \times 10^5)\) to reproduce wind tunnel tests or 30 m/s \((\text{Re} = 12 \times 10^6)\) to represent strong wind conditions. The turbulence intensity was 3 per cent and the turbulence viscosity ratio had a value of 5. The latter is the ratio between the turbulent viscosity, which is a property of the flow, and the viscosity of the fluid. At the outlet, a constant pressure boundary was adopted. Finally, for the top and bottom faces a symmetry condition was selected. The external borders of the domain are far enough apart to simulate the free flow condition. The blockage factor of the computational domain was 0.04, which ensured that the distortion of the flow was reduced to the minimum.

The flow domain in Figure 24 was meshed with quadrilateral elements of number oscillating between 1.5 million elements and 2.5 million elements. In the scenario where there wasn’t any fence, a shorter number of elements were needed, increasing when the fence was installed without eave and increasing again up to 2.5 million in the scenarios where the fence was equipped with an eave on top.

The element size grows as the cells gets closer to the limits of the computational domain, that is to say, the mesh is non-uniform to save elements in those regions where the flow wasn’t perturbed by the geometry of the car and the bridge.

In the vicinity of the carbody, a fine layer of cells was included to keep the dimensionless wall distance \((y^+\) within limits. Although this model is not 3D, the limits of the standard EN 14067-6:2010 were adopted. For RANS simulations, the limit for
low-Reynolds simulations is the order of 1 and between 30 and 150 for high-Reynolds number. No layer was included around the bridge although cells were kept small to represent the flow accurately.

Two checks were done to guarantee the quality of the mesh. Firstly, we used the equiangle skew factor to assess the angular distortion of each element of the mesh and sets a value to each one between zero, that means no distorted, and one, that means highly distorted. Measures showed an average value of $1 \times 10^{-5}$. The area that surrounds the vehicle and the bridge was analyzed in more detail, noting that only 1 per cent of the elements had a value above 0.3. Secondly, by meshing again the scenario in which there is no fence, we could assert that results were independent of the mesh. The refinement ratio was 1.25 that changed the aerodynamic coefficients a value that didn’t reach 2 per cent at both Re.

b) Turbulence model, flow assumptions and numerical considerations

Calculations were done with the turbulence model $k-\varepsilon$ Standard although the Realizable and RNG versions were also tested. Results showed that the Standard version fit better than the others when they were contrasted with the tests carried out in the wind tunnel; therefore, this was the model that was used in the simulations.

In spite of having constant boundary conditions at the inlet, the nature of the problem is transient since when there is a bluff body inside an air stream, a vortex shedding phenomenon emerges. The convergence criterion was to reach the periodic regime, so the method consisted of getting the time signals of the coefficients by providing enough cycles so that this state was guaranteed. Thus, when the value of the aerodynamic coefficient is presented for a certain scenario, it is given as the average in time.

Each wind velocity value required a specific time step because the frequency of the vortex shedding phenomenon depends on Re; therefore, two time steps were needed. The step size was 1 s in the cases of $v_\infty = 0.22$ m/s, whereas for $v_\infty = 30$ m/s it was 0.02 s; this made it possible to drive the scaled residuals to $10^{-6}$ in each time step.

Second order upwind numerical schemes were employed for the convective terms of momentum and turbulence equations (Mathur and Murthy, 1997). The pressure-velocity coupling was made in terms of the PISO algorithm (Issa, 1986). Scaled residuals were driven to $10^{-6}$ in each time step.
Mock-up to carry on experimental tests

Experimental tests in a wind tunnel were carried out by IDR/UPM, E.T.S.I. Aeronáuticos, Universidad Politécnica de Madrid with the aim of checking the CFD model. The experimental set-up looks to characterize the aerodynamic loads on the coach's surface in order to calculate the aerodynamic coefficients that make a comparison with the results of the CFD model possible. This means that no other conclusion was extracted from the test campaign apart from testing the CFD model. In order to fulfil this goal, an open-circuit wind tunnel with a closed test section was used to perform a set of two-dimensional tests. The wind tunnel's working section is 1.8 m high, 0.2 m wide and 1.2 m long.

For the case study presented here and in order to reproduce the most important aerodynamic characteristics, a scale of 1/50 was chosen for the whole mock-up. The geometry of the coach was simplified due to limitations of the mock-up manufacturing process (see FIGURE 25). It will be shown that this is one of the reasons for the differences between the experimental and the CFD results.

A Scanivalve Corp. pressure scanner, model ZOC33, with 128 pressure inputs, was used to measure the pressure on the model surface. The train model was equipped with 48 evenly distributed pressure taps near the mid section (FIGURE 25), each them connected to the pressure scanner through the pneumatic inputs. A pitot tube located upstream from the mock-up and near the top of the chamber was used to measure both the total and the static pressure to determine the dynamic pressure of the airflow. The time average value corresponding to each pressure tap was computed in order to determine the pressure coefficient. The global aerodynamic coefficients were obtained by numerical integration of the pressure coefficients on the model surface. Different values for the sampling rate and the sampling period were checked to measure the pressure signal corresponding to each pressure tap. After several observations, the selected sampling period was 12 seconds as the average values of the coefficients did not change appreciably for larger sampling periods. The selected sampling rate was 100 Hz to fully capture the time history.
The Reynolds number at which the tests were performed was approximately $1 \times 10^5$ and the free flow speed was uniform and equal to 11 m/s within ± 1 per cent. The free flow turbulence intensity was measured using a DANTEC (CTA module 90C10 and probe type 55P16) hot wire anemometer system. EN 1991-1-4, Eurocode 1 gives recommendations for setting up the value of the free flow turbulence intensity, which is around 7 per cent for the case of a location with distance to the ground level that is similar to the case of a bridge. However, this standard is written for the study of the infrastructure and it does not cover the case of vehicles situated on it. Therefore, different values of turbulence intensity were checked in the range from 3 per cent to 10 per cent by placing different grids at the beginning of the working section. Measures show that larger values of turbulence intensity produced slightly smaller
loads on the vehicle. The reason is that wind tunnel tests with low turbulence levels create a more severe condition since the average of the experimental loads is higher (Suzuki et al., 2003). However, these differences are not significant since the bridge itself perturbs the flow enough to reduce the impact of this parameter (Avila-Sanchez et al., 2010). In this context, the selected value for the turbulence intensity was 3 per cent since the safety margin is increased as actual loads are expected to be lower. No atmospheric boundary layer was reproduced since the height at which the vertical velocity gradient changes significantly is much larger than the parapet height.

Due to the limited dimensions of the test section, the blockage factor was close to 0.14, which can be considered to be high. As a result of the blockage effect, the velocity of the airstream around the model increased slightly, which amplified the loads that the model withstood. Typical increases in the aerodynamic coefficients in this kind of experiment are expected to be 15 per cent, so the coefficients were corrected. Two-dimensional tests may require of corrections due to the growth of the boundary layer of the tunnel, solid blockage as a consequence of the size of the mock-up and wake blockage (Maskell, 1965) caused by the model. The growth of boundary layers on the tunnel walls modifies the static pressure along the wind tunnel. For bodies whose longitudinal dimension is much larger than the vertical dimension, the effect is an increase in the aerodynamic resistance. Since our model dimensions are similar in both directions, this correction was not considered. Solid blockage correction as well as wake correction were applied according to (Barlow et al., 1999). However, corrections are based on empirical considerations, and thus the uncertainty in including the blockage effects can be considered an explanation for the disagreement between the experimental and computational values of the aerodynamic coefficients.

**d) Validation of the CFD model with experimental tests**

Wind tunnel tests were done to check the CFD model and given the purpose of this paper only the mean values of the aerodynamic coefficients are provided.

According to the results in FIGURE 26, CFD simulations satisfactorily predicted the trends of the aerodynamic coefficients that were measured in wind tunnel tests. In general terms, the CFD model provided lower values for all the aerodynamic coefficients, the only exception being the case where the vehicle was not protected by any wind breaking device; in such a case CFD predicted higher values. Three were the main causes expected to be source of uncertainty: the turbulence model used in the CFD, the simplified geometry of the mock-up and the wind tunnel blockage. The well-known limitations of the employed RANS turbulence model introduce uncertainty in the CFD model for all the computed cases. The carbody of the mock-up does not
Chapter 3: vehicle models

represent exactly the true-geometry of the vehicle but it does in the CFD model. The downside of the mock-up is flat whereas the true-geometry has a low inclination angle. Moreover, the true-lateral sides of the geometry are a succession of curves of different radius whereas the mock-up adopts the curves’ envelop. This lead to differences between the CFD model and the experiments in the case of unsheltered vehicle. The blockage can be the cause of getting larger differences between the computational model and the experimental tests in the cases of \( h_t = 2\,250 \text{ mm} \) and \( h_t = 2\,750 \text{ mm} \).

Additionally, the free flow velocity is constant in CFD whereas in the wind tunnel it has variations of 1 per cent. The same applies to the turbulence intensity, where its value does not change in CFD but in tests it does. However, the influence of the inlet conditions is expected to be low since the wind profile at the inlet is almost identical.

In the case of the drag coefficient \((C_d)\), the difference between wind tunnel tests and simulations grew as the fence height increased. The computed coefficients of lift force \((C_l)\) and roll moment \((C_{m0})\) showed the biggest disparity within the experimental tests in the configuration where the train was completely exposed to crosswinds. In the scenario where there is no fence to shelter the vehicle the coach reorients the flow, which makes it highly dependent on the carbody geometry. Thus, as long as the geometry between the mock-up and the CFD model agree, the pressure differences between top and bottom agree and so do the aerodynamic coefficients of lift and roll moment. Note that in the case of \( C_{m0} \), the CFD model predicted reasonable well the effect of fences.

In view of the comparison, CFD simulations were able to predict reasonably well the value of \( C_{mv} \), extracted from the tests in the scenarios where the vehicle was unsheltered and when the fence height was 1\,250 mm and 1\,750 mm. The difference increased in the cases of the highest fences, which might be due to the larger effect of blockage. \( C_{mv} \) presents a similar trend to \( C_d \), simulations accurately predicted the situation of the unprotected vehicle, but the inaccuracy increased with fence height. Although \( C_{m0} \) also determines \( C_{mv} \), its near-zero value when the bridge is equipped with fences made it not significant.

The fact of having fences with eaves on the top of them did not modify the conclusions presented above. Simulations properly reproduced the difference between the geometry where the fence had no eave and the geometries where it did. The computational model was in agreement with the experimental tests where adding an eave reduced the value of the coefficients but an increase in eave length had a very low impact on the coefficients. In most cases, the CFD calculations predicted smaller variations between the two eave lengths than the results of the tests gave.
In general terms, the CFD simulations were able to represent the variations in the aerodynamic coefficients as the result of building fences with eaves. The CFD model was able to reproduce the trends of the coefficients, showing the effect of increasing the fence height and also adding an eave. However, the difference between the measured and the computed coefficients is larger in the cases of fences with height 250 mm and 2750 mm. The CFD calculations also matched the effect of increasing the value of the parameter $l_e$ from 500 mm to 750 mm, which appears to be low in both wind tunnel experiments and computed scenarios. Therefore, CFD is a suitable tool for performing a parametric study of the fence design and analyzing how the airflow is affected by it.

### 3.2.2 3D models

The study of the vehicle aerodynamics in the 3D space was done in two stages: a CFD code validation and a brief analysis of wind breaking devices. Section 1.6 describes that one of the goals of this thesis was to perform simulations with a CFD
code based on the Lattice-Boltzmann Method (LBM). However, the current standards of the crosswind field firstly require validating CFD codes before doing actual simulations. To validate the CFD code we performed static tests in a simple scenario, which was the flat ground with 235 mm gap (see Section 2.2.2), and in a more realistic configuration using the double track ballast and rail scenario (see Section 2.2.3). The validation was done following recommendations by the European Standard EN 14067-6:2010, being the results presented in Section 5.3. Only when the employed CFD code was checked using simple configurations, we did a short study about the sheltering effect of wind breaking devices in the 3D space (presented in Section 6.2).

The following sections present the vehicle model used in both the CFD validation and the three-dimensional study of fences. The domain configurations employed for the simulations of each study are also described.

a) Aerodynamic model of the ICE 3 train

The model consists of the leading car of an ICE 3 and half of the adjacent car. The CAD model was provided by the European Standard EN 14067-6:2010 so the model with which we worked and the one whose results are given in the Annex C, part 2 and Annex E of the Standard were the same. In such way, differences that came out from inaccuracies of the car shape are avoided.

FIGURE 27 shows the geometry of both cars and the main dimensions. It is important to remind that the reference normalization area $A$ and length $d_0$ used to calculate the aerodynamic coefficients are both set by the CEN standard, being $10\, \text{m}^2$ and $3\, \text{m}$ respectively.

The leading vehicle is the one that needs to be aerodynamically characterized since it is the critical one from the point of view of overturning risk. The adjacent car needs to be modeled to emulate the situation at which the real vehicle would be if it were subjected to crosswinds. To do so, the standard allows reproducing only half of the adjacent car and this is what it is given in the CAD data.

The model reproduces the real external geometry of the leading car, assuming simplifications only on the bogies. Since the separation between cars is normalized, the distance between adjacent cars as well as the shape of the connection are carefully modeled. We considered important to have the same geometry provided in the standard because the model was firstly used to validate the CDF code XFlow that is based on the Lattice-Boltzmann method (details are given in Section 5.2).
b) General aspects of simulation

Calculation of aerodynamic coefficients

The aerodynamic loads \( (F_i) \) and moments \( (M_i) \) of the leading car are calculated according to the CEN code, and assuming the coordinate system from the standard which is presented in FIGURE 30:

\[
C_n = \frac{F_n}{0.5 \cdot \rho \cdot A \cdot \nu_\infty^2}
\]  
(3.6)

\[
C_{M_n} = \frac{M_n}{0.5 \cdot \rho \cdot A \cdot d_0 \cdot \nu_\infty^2}
\]  
(3.7)

being \( \nu_\infty \) the free flow air velocity, \( \rho \) the air density (1.225 kg/m\(^3\)), \( A \) the reference area (10 m\(^2\)) and \( d_0 \) the reference length (3 m).

The moment around the leeward rail \( (M_{x,lee}) \) assesses the turnover of the train under crosswinds. This moment can be considered as the one that the restoring moment should balance to keep the vehicle in equilibrium; its expression for the yaw angle represented in FIGURE 30 and FIGURE 31 is:

\[
M_{x,lee} = M_x + \frac{b}{6} \cdot F_y
\]  
(3.8)

being \( b \), the distance between wheel/rail contact points, 1.5 [m]. \( M_{x,lee} \) is the same as \( M_x \), but expressed in a different reference system of coordinates.
The calculation of $M_{x,\text{lee}}$ is also necessary to validate the numerical calculations. As Section 2.1.1 explained, the CEN code states that the non-dimensional coefficient of the moment around the leeward rail ($C_{Mx,\text{lee}}$) should be calculated in each simulation and compared with the benchmark results given by the standard. The standard specifies tolerances for the maximum instant value and for the mean value of $C_{Mx,\text{lee}}$ that were already described in Section 2.1.1:

\[
\max \left\{ \frac{C_{Mx,\text{lee}} - C_{Mx,\text{lee},\text{bench}}}{C_{Mx,\text{lee},\text{bench}}} \right\} < E_{\max} \quad (3.9)
\]

\[
\text{mean} \left\{ \frac{C_{Mx,\text{lee}} - C_{Mx,\text{lee},\text{bench}}}{C_{Mx,\text{lee},\text{bench}}} \right\} < E_{\text{mean}} \quad (3.10)
\]

**Domain discretization**

The Lattice-Boltzmann Method (LBM) is a CFD meshless approach based on mesoscopic theories. It can be considered a meshless approach in the sense that the setup of the problem does not include the meshing process in which the domain is divided manually into a finite number of volumes of control. The main difference between a CFD code that is grounded on the Finite Volume Method (FVM) and a code based on LBM is that in the latter, including complex geometries is straightforward. However, there is still a discretization of the domain (by a lattice structure) that replaces the mesh, and as in CFD codes based on FVM, the discretization have influence on the results.

The idealistic representation of the flow at the mesoscopic scale is only accurate with small lattice sizes that reduce the distance between adjacent lattices to the minimum. This is a key point since the reconstruction of the flow at the macroscopic level definitely depends on how the mesoscopic level is modeled. In addition, the directions at each lattice are discrete, and the evolution of the system is assessed at the lattices’ centers. A short description of the method is available in Section 5.2.

The software carries out the discretization of the domain automatically and builds the initial octree lattice structure based on the parameters defined by the user. The octree can have different levels, and each lattice belonging to one of these levels is two times smaller than a lattice in the previous level. Usually, the dimensions are given for three positions: lattices in the far field, lattices on the geometries’ surfaces and lattices at the wake. In this way, lattices located at the wake (triggered by the dimensionless vorticity of the flow) or near the geometries will have smaller dimensions than those in the far field. This procedure would be equivalent to create small cells near to the areas of interest and big cells in the far field in conventional CFD approaches. Logically, this is due to the fact that performing simulations with small lattices in the entire domain would lead to non-acceptable computational times.
Chapter 3: vehicle models

The pre-processor creates the initial octree based on the input given by the user and it evolves dynamically. Once the simulation starts, the lattices dimension evolves either by the presence of moving parts or to adapt to new flow conditions. Therefore, the size of the lattice for a certain position in the domain can change between time steps because the lattice evolves dynamically with the flow. Figure 28 shows an example of the octree at a certain instant of time of a simulation with wind blowing from right to left. The plane cuts the geometry perpendicularly, showing the octree for the case where the wake is at the leeward (left hand) side of the vehicle. The octree has 8 levels, where the smaller dimension is on the geometry surface (in orange color) and the largest dimension is at the far field represented by a cross symbol in blue color.

The modelization of the boundary layer on the geometry surface is done similarly as in conventional CFD codes. In order to accurately calculate the near-wall flow, the transition from the small lattices at the wall to the lattices at the wake should be smooth. Different numbers of element layers have been used for the simulations according to the limitations of the available computational resources.

Lattices in LBM are all cubes so the ‘mesh quality’ concept of conventional CFD cannot be applied here. To check the convergence of the problem, XFlow gives the user feedback in terms of the ‘stability parameter’ (\( \eta \)):

\[
\eta = \frac{v_{\text{max}}}{c_s}
\]  

(3.11)

where \( v_{\text{max}} \) is the maximum speed of the fluid in the domain and \( c_s \) is the numerical speed of sound (see Section 5.2.2).

The stability parameter should be kept below 1 to satisfy the Courant-Friedichs-Levy (CFL) condition, hence it gives an estimation of the numerical error of the problem. All
the results herein presented fulfill the rule of keeping this value under 0.3 since the stability parameter in our simulations was always below 0.1.

**Turbulence model and wall functions**

The air was considered isothermal and incompressible and the properties used for the simulations are those for the standard conditions, i.e. a density $\rho = 1.225 \text{ kg/m}^3$ and a viscosity $\mu = 1.7894 \times 10^{-5} \text{ kg/(m·s)}$. With these properties and the value set to the free flow air velocity ($v_\infty = 2 \text{ m/s}$), the Reynolds number based on the height of the train is $Re = 4 \times 10^5$. Therefore, the crosswind flow around the train is known to be turbulent.

The simulations were done with the turbulence model Wall-Adapting Local Eddy-Viscosity (WALE) model for being especially well suited for CFD calculations in terms of the LBM. The software provides two wall models: the Enhanced wall function and the Non-equilibrium Enhanced wall function, and only the latter takes into account the favorable and adverse pressure gradients effects (see Section 5.2.3). This feature was determined to be of the greatest importance, since the characteristic vortex at the leeward side of the vehicle was only reproduced when the Non-equilibrium model was employed. FIGURE 29 shows a comparison of the flow (30-degree case) calculated in terms of the two wall models, checking that the Enhanced wall model failed to correctly represent the near-wall flow whereas the Non-Equilibrium model provided the conditions for the vortex generation. The inclusion of the pressure-gradient effect in wall functions has also been reported to provide more accurate predictions of separated flows with the LBM-based code PowerFLOW (Shock et al., 2002; Fares, 2006).

According to the explanation given above, the Non-equilibrium enhanced wall function was selected for the surfaces of the train so as to guarantee that the characteristic vortex is reproduced. On the other hand, the simpler Enhanced wall
function was selected for the track and the fences since the pressure gradient is not significant in these geometries.

c) Setup for the tests to validate the CFD code

Domain configuration

There are several ground configurations available in the literature; in fact, the appropriateness of which scenario could be the most representative has been object of discussion in the scientific community. By choosing the flat ground (FG) scenario with 235 mm gap that is available in both the CEN and the TSI standards (FIGURE 14), we avoided disturbances from the track model. This allowed us to focus the discussion on the flow simulation and to avoid additional sources of inaccuracies in the first set of simulations. We also performed simulations with the ground with ballast and rail FIGURE 15 in order to check the accuracy of the results provided by the software with a more realistic ground representation.

The computational domain was three-dimensional and represented a parallelepiped (see FIGURE 30). Its dimensions were checked so that the numerical results were independent from an increase in any direction. The blockage ratio on the vertical direction was 5 per cent and at yaw angle 90° takes place the maximum blockage on the spanwise direction that was 25 per cent.

The boundary conditions are indicated in FIGURE 30 too. A uniform wind profile was set at the inlet and at the top of the domain. The free flow velocity was set to 2 m/s ($Re = 4 \cdot 10^5$) to match the Reynolds number (calculated with the height of the train) at which the coefficients are independent from the wind velocity, as stated by the CEN standard and authors like Schober et al., 2010. The turbulence intensity was 0.5 per cent since low turbulence values are recommended by the standard. At the outlet, we adopted a constant pressure boundary. A periodic boundary was selected for the laterals, keeping them far enough from the train so as to simulate the free flow condition. The boundary condition for the ground wall was a slip condition to guarantee that a boundary layer near the ground did not develop before the flow reached the vehicle. We also tested a non-slip boundary condition at the surroundings of the vehicle to simulate the true ground, but results show that it had low impact on the aerodynamic coefficients of the vehicle.
Figure 3.1: Non-scaled representation of the CFD domain, front view (up) and top view (down). Dimensions are given in terms of the length \( L_r = 40 \) m and the height \( H = 4 \) m of the train model. The boundary conditions are in grey for the top and the laterals of the domain. The boundary conditions of the ground is in black.

**Domain discretization.**

The dimensions of the lattices in our finest simulations were: 1.6 m at the far field positions, 25 mm at the wake and 25 mm at the shapes' surfaces. Six element layers between two refinement levels were included to properly solve the flow around the leading car and reproduce the boundary layer detachment on the vehicle surface as accurate as possible. This produced simulations with up to 150 million elements, needing more than 10 days to get a solution for the problem. However, we observed that the lattice size of the wake could be increased up to 50 mm without losing accuracy, allowing us to decrease substantially the number of elements to 100 million.
elements. Thus, the following lattices sizes were used to discretize the domain of the two scenarios simulated:

<table>
<thead>
<tr>
<th>Flat ground with 235 mm gap</th>
<th>Double track ballast and rail</th>
</tr>
</thead>
<tbody>
<tr>
<td>Leading car</td>
<td>Leading car</td>
</tr>
<tr>
<td>Downstream car</td>
<td>Downstream car</td>
</tr>
<tr>
<td>Wake</td>
<td>Wake</td>
</tr>
<tr>
<td>Track under the end car</td>
<td>Track under the car</td>
</tr>
<tr>
<td>Track under the car</td>
<td></td>
</tr>
<tr>
<td>Far field</td>
<td>Far field</td>
</tr>
<tr>
<td>0.025 m</td>
<td>0.025 m</td>
</tr>
<tr>
<td>0.025 m</td>
<td>0.05 m</td>
</tr>
<tr>
<td>0.05 m</td>
<td>0.05 m</td>
</tr>
<tr>
<td>0.025 m</td>
<td>0.05 m</td>
</tr>
<tr>
<td>1.6 m</td>
<td>1.6 m</td>
</tr>
</tbody>
</table>

TABLE 12: Lattice sizes for the configuration of the ground scenarios used to validate the code based on the Lattice-Boltzmann Method

In view of the results presented in Section 5.3, a refinement of the lattices on the vehicle with half the size used in our simulations would be needed to predict the aerodynamic coefficients better. However, this was not possible to be done due to both time and computational resources restrictions. The computational time required for this purpose increased substantially with simulations needing approximately 30 days. The reason is that the factor between the sizes of the lattices on the geometry surface and the lattices on the wake must have a maximum value of two. Otherwise, we observed that the flow around the train did not represent the one observed in real operations.

d) Setup for the simulations with wind breaking devices

Domain configuration

The goal of this 3D model is to assess the sheltering effect of some of the fence designs that were tested with the aerodynamic 2D model. The vehicle stood on the double track ballast and rail ground (see FIGURE 15) since this ground scenario allowed us to correctly simulate the flow underneath the train. Since the ground model has two tracks, the vehicle model was located at the windward side for being known as the position at which the vehicle withstands higher loads. The track is shorter than the domain’s width yet long enough to recreate realistic flow conditions around the vehicle.

The wind fences included in the model were selected among the fence designs tested with the 2D model (see Section 3.1.2). Two fence configurations were assessed at a yaw angle of 90 degrees, which are according to the already known parameter of fence height: $h_f = 1.750$ mm and $h_f = 2.750$ mm. The distance between fences and the distance between fences and the vehicle were taken from the 2D model, so that the
flow simulated in the three-dimensional space could be as similar as possible to the simplified model.

FIGURE 31: Non-scaled representation of the CFD domain, front view (up) and top view (down). Dimensions are given in terms of the length ($L_r = 40$ m) and the height ($H = 4$ m) of the train model. The boundary conditions are in grey for the top and the laterals of the domain. The boundary conditions of the ground are in black.

FIGURE 31 shows the computational domain and the boundary conditions used to carry out the simulations. In this case, the fences must have the same length of the domain's width in order to avoid unreal border effects. Thus, the limited computer resources indirectly set the Z-dimension of the computational domain since the longer the fences are the highest is the number of lattices on the wake. The blockage ratio was 5 per cent on the vertical direction and 25 per cent on the spanwise direction. The boundary conditions at the domain laterals were both pressure outlets to allow the flow exit the domain and to reduce the blockage effect on the spanwise direction. A uniform airflow profile was prescribed at both the inlet and at the top of the domain, being the velocity 2 m/s and the turbulence intensity 0.5 per cent, so that
the conditions matched with the ones selected in the simulations to validate the code. A constant pressure boundary condition was set at the outlet. The ground of the domain was a free slip condition to avoid the development of the shear layer.

**Domain discretization**

We already presented the discretization procedure in part B of the current section, and further information can be found in Section 5.2. The whole set of lattice sizes used for these simulations are in shown TABLE 13, and the lattice sizes of both the vehicle and the track model are schematically represented in FIGURE 32.

![FIGURE 32: Definition of the lattice sizes of both the vehicle and the track model](image)

Each one of the geometries included in the model had only three layers of lattices near walls so as to meet computational requirements. These were the finest lattices dimensions we could afford due to the limitations of the computer used to carry out these simulations, which has 96 GB of RAM memory. The current set up made the simulation to reach 250 million elements that is the maximum number of elements that the computer can retain in memory. The time needed however to reach the steady state was similar to the simulations that were done to validate the code (more than 10 days). We compared this configuration with the set up used to validate the code (see TABLE 12) and the difference of the aerodynamic coefficients at the yaw angle of 90 degrees was around 7 per cent.

<table>
<thead>
<tr>
<th>Vehicle models</th>
<th>Track and fence models</th>
</tr>
</thead>
<tbody>
<tr>
<td>Leading car</td>
<td>Track under the end car</td>
</tr>
<tr>
<td>Downstream car</td>
<td>Track under the car</td>
</tr>
<tr>
<td>Wake</td>
<td>Track ends</td>
</tr>
<tr>
<td>Far field</td>
<td>Fences</td>
</tr>
<tr>
<td>0.025 m</td>
<td>0.025 m</td>
</tr>
<tr>
<td>0.2 m</td>
<td>0.2 m</td>
</tr>
<tr>
<td>0.05 m</td>
<td>0.2 m</td>
</tr>
<tr>
<td>1.6 m</td>
<td>0.1 m</td>
</tr>
</tbody>
</table>

TABLE 13: Lattice sizes for the configuration of the simulations to assess the sheltering effect of fences in the 3D space
Chapter 4

vehicle dynamics in a crosswind context

This section focuses firstly on studying and comparing calculations from the 2D model and the full multibody model. The goal is to identify and to quantify differences between both models, understanding where these differences come from and knowing whether the simple model gives a bigger or smaller margin of safety. Although the current standard that cares about crosswinds prescribes the use of a full multi-body model for most situations, there are some simulations in which the standard allows a 2D model. In these cases, we can assume a 3D-2D difference. If the error is small, it can be similar to the one implicit in the characterization of the suspension components or the calculation of the 3D aerodynamic coefficients. Therefore, the 2D model would be advantageous for studying some aspects related to the response of the vehicle, like the construction of CWCs, the measurement of body displacement or the wheel unloading evaluation.

The comparison will be made in the time domain in order to clearly note the difference between both models. Finally, those aspects related to the vehicle model that differentiate between the full and the 2D model and determine whether the 2D model is affordable will be noted. One important aspect of this difference is how the wheel/rail contact is modeled. The different possibilities SIMPACK gives the user will be presented and then compared by means of the wheel unloading factor. Another aspect is the aerodynamic loads considered in the simulations. Obviously, the six components cannot be introduced in the scenario when working with the 2D model; this limitation will be also considered and compared in the calculation of the CWCs.

Once the scope of the 2D model is cleared, this section studies the vehicle dynamics when the train runs on a curved track. This study can only be done in the 3D space, thus it only requires simulations from the full multibody model. If the effect of crosswinds is added to the centripetal acceleration, high-speed vehicles may tend to yaw, becoming instable. This situation is of interest because the vehicle's comfort is significantly reduced even though the vehicle fulfills the safety regulation. In this
context, we studied the cause that makes the vehicle yaw and the effect of a few secondary suspension elements on the hunting movement.

4.1 Standard EN 14067-6:2010

Standard EN 14067-6:2010 presents various methods for assessing the crosswind stability of railway vehicles (see Chapter 2). In the standard, stability is characterized in terms of the wheel unloading factor. From the dynamic point of view, the standard prescribes different procedures for determining that value: building a full multi-body model, building a simplified five-mass model, or building just a two-dimensional vehicle model.

For both quasi-static models, the 2D model and the five-mass model (a carbody, two bogies and two mass-equivalent wheelsets), the results from these configurations are considered to be less precise but conservative because of built-in margins: the characteristic wind speeds are lower than those of a multi-body model. Furthermore, just the multi-body model is expected to integrate the wind scenario in time, this means that only the full model provides time-dependent simulations.

The standard prescribes that ‘the five-mass model is suitable for many vehicles’, giving the reader the choice of deciding between the five-mass and the multi-body model, always taking into account the limitations of the quasi-static study of the five-mass model. In the end, the simplest way to study the effect of crosswinds is through a 2D model. The standard warns that “the method is less precise but more conservative because of built-in margins” and is only suitable for conventional vehicles with two sets of running gears or two single axes.

In the context of the conditions referred to in the standard, the purpose of our analysis is to check whether a simplified 2D model is appropriate for studying the response of the vehicle to crosswinds through a time-dependent simulation. In fact, when wheel unloading is calculated, the destabilization process and thus the movements that the bodies describe are mainly two-dimensional. The proposal of this study consists of a simplified dynamic model of the full one. Nevertheless, in the process of simplifying the full multi-body model into a 2D model some elements will necessarily be neglected because they have a strictly 3D configuration. Some others will be simplified in transforming their three-dimensional behavior to two-dimensional space. The final 2D model, which is the result of the simplification process of the full one, will lead to an error that must be quantified before being accepted.
4.2 Comparison between vehicle models: response to wind loads

Introducing the gust model presented in the EN standard, which was the Chinese Hat, we did a comparison of the response to wind loads between the 2D model (see Section 3.1.2) and the full multibody model (see Section 3.1.1). This wind model fulfilled perfectly our needs since it consisted of intervals with steady loads and a gust. The same loads were applied to both models to compare how the models respond to them; hence, we checked whether by integrating a three-mass model in the time domain we were able to get reliable results to study the vehicle dynamics under crosswinds.

Both models included bumpstop with a non-linear stiffness characteristic. The full model had two at each bogie whereas the equivalent element was introduced in the simplified model. One of the most critical elements of the suspension set of the vehicle is the bumpstop since how it works is a key factor in the response of the vehicle in terms of dynamic behavior under crosswinds. The standard also identifies it as a critical element if the study is centered on dynamics under crosswinds. FIGURE 33 compares the lateral deformation of the bumpstop for the two models. It is remarkable how both curves follow the same pattern through the entire wind scenario; the stop is located at 60 mm of displacement, and both models reach it at the same time. Amplitude of the oscillations differs but the error is acceptable.

![Figure 33: Lateral deformation of the bumpstop. Comparison between the 2D model and the multibody model](image-url)
We already verified the rotation angle with the sway test (see Section 3.1.2b) though that was a quasi-static test. Such test anticipated the agreement of the rotation of the carbody between models and FIGURE 34 confirms the statement. The oscillations of the 2D have higher amplitude but its frequency matches, similarly as what we observed with the lateral deformation of the bumpstop.

The primary suspension was of type trailing radial arm; hence, a model in the 3D space was required in order to properly model this element. The simplified model could not faithfully represent this suspension stage since some components were removed. As FIGURE 35 shows, this simplification had some influence on the measurement of the vertical displacement. At this stage of the suspension, oscillations between models do not agree as good as at the measures at the second stage of suspension. Moreover, the two-dimensional model would have required more time to reach a steady state after the gust of wind.
In FIGURE 36 the displacement of the carbody CoG is shown. Note that the three-mass model is less damped, which means the model needs more time to get to the steady state after being hit by the wind gust. While the multi-body model reaches equilibrium at around 40 s, the 2D model needs a few more seconds. Small differences can be observed when displacements reach the maximum value at the moment the wind gust hits the vehicle. Furthermore, the three-mass model gives the highest value of the measure as a consequence of the damping characteristic of the model. If the same aerodynamic forces load the multi-body and the three-mass model, almost the same lateral displacement is observed when loads are constant. The difference becomes greater when the full set of loads is applied to the multi-body model, but in that case different load cases are compared.
4.3 Guaranteeing safety with a 2D model

The 2D "is considered to be less precise but conservative because of built-in margins": the characteristic wind speeds are lower than those of a multi-body model. The simplest way to study the effect of crosswinds is through a 2D model. The standard warns that "the method is less precise but more conservative because of built-in margins" and is only suitable for conventional vehicles with two sets of running gears or two single axes.

In the context of the conditions referred to in the standard, the purpose of our analysis is to check whether a simplified 2D model is appropriate for calculating the so-called CWCs of the vehicle. Simplified models built with fewer bodies, simpler suspensions and without a wheel/rail contact model could safe. The problem is that some elements could not be included in the simplified model as a consequence of the inherent 3D stiffness or damping characteristics.

This section compares results of wheel unloading calculations that were got with both models. Since the wheel unloading factor is calculated in order to build the vehicle's CWCs, a comparison of the CWC is also described. Finally, we analyzed the influence of the contact model and its impact on the CWCs.
4.3.1 Wheel unloading calculation. Restrictions of the 2D model

In FIGURE 37 the comparison of wheel unloading from the simplified model and the multi-body model is shown. Throughout this section the crosswind scenario considered is one where the vehicle is running on a straight track and the angle between the flow and the track is 90°. The wind scenario matches with the standard double-track ballast and rails. The load case corresponds to a planar state, which means only \( F_y, F_z, \) and \( M_x \) were considered. In general terms, both curves have the same appearance. The differences between the models appear because some elements cannot be introduced in the simplified model. This is what happened with the primary suspension, where the sway link was suppressed in the simplified model, as well as the yaw damper in the secondary suspension. The suppression of the lateral and longitudinal rigidities of the silent block produce a deviation in sustained wind, as shown from second 15 to second 25. What the suppression of the yaw damper does is modify the transitory response of the model; this effect is observed between second 30 and second 40. The peak amplitude of the wheel unloading has a small error in concordance, giving a difference of 0.034 while the difference when the wind loads are constant is 0.024.

![Wheel unloading comparison](image)

FIGURE 37: Wheel unloading comparison between the full model and the simplified model

Summing up, it can be said that the three-mass model is less damped and that the maximum wheel unloading is lower in comparison with the multi-body.
Chapter 4: vehicle dynamics in a crosswind context

Table 14: Calculation of the wheel unloading factor through different methods and with both models. The factor is calculated at each wheelset, in terms of the CEN standard and averaging the four wheelsets of the vehicle. WS denotes wheelset 'i'

<table>
<thead>
<tr>
<th></th>
<th>Multibody model</th>
<th>2D model</th>
</tr>
</thead>
<tbody>
<tr>
<td>( \Delta Q/Q_0 )</td>
<td>WS₁</td>
<td>WS₂</td>
</tr>
<tr>
<td>( (Q_{i1}+Q_{i2})/2 \cdot Q_0 ) (standard)</td>
<td></td>
<td></td>
</tr>
<tr>
<td>( \Sigma Q_i/4 \cdot Q_0 )</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

The CEN standard recommends averaging the Q-forces of the two wheelsets on the first bogie whenever a full multibody model is used. The three-mass model does not give such an option because the four wheels of each side are concentrated in one. Therefore, the measure given by this model will be closer to an average of the four wheels at the leeward side of the full vehicle. Table 14 compares the wheel unloading factor for the same wind scenario as the previous figure. The table contains the factor calculated for each wheelset; using the formula given by the standard for the wheelsets of the first bogie; and by averaging the ratio of the four wheelsets—in this case, using the full and the simplified models. If the purpose of building the simplified model is to calculate the wheel unloading ratio, this demonstrates that averaging is a good approximation. The difference between the 2D model and the 3D model is estimated to be 3 per cent. Obviously, the limitation of not being able to calculate Q-forces separately in every wheelset cannot be overcome. If they are needed, the full model is the only suitable way to solve the problem.

Another question arises and it is related to the wheel/rail contact model. The model of suspension components is important but wheel/rail contact model is key on most studies of railway dynamics. In our context, averaging Q-forces at the contact during the wind scenario assesses safety. We considered interesting an evaluation on how the wheel/rail contact model influence the wheel unloading ratio since whenever a vehicle model is built, one must choose among the many possibilities that multibody codes offers to simulate contact. Furthermore, the representation of the contact in the three-mass model is limited because the interaction between wheels and rails can only be simulated with model of a spring and a damper.

As in any rail vehicle multi-body simulation a contact option has to be selected, this section explores if the different options might have any influence in the context of CWC calculations. When using SIMPACK, different ways of modeling the contact are possible, and results may differ, as explained below. In SIMPACK the user can choose between:
• Option 1: constraint or elastic.
• Option 2: onepoint or multipoint (see FIGURE 38).

The first option refers to how the forces in the contact patch are calculated. If constraint is selected, an internal SIMPACK algorithm is used. Selecting elastic introduces a spring/damper with certain rigidity and damping perpendicular to the contact direction by introducing force only in compression. In the crosswind context, both methods produce similar results. The second option allows the user to choose how many contact points to take into account.

In order to check what is the difference between onepoint and multipoint contact is, the full vehicle model is loaded by a wind force type step whose components are $F_y = 250 \text{ kN}$ and $M_x = 250 \text{ kN}$; the others are zero. The load is big enough to move the wheelset up to get flange contact. If the contact path is represented with one point, the equilibrium will be given by the displacement of the single point and the load that is transmitted through it. The contact point can move from the static position towards the flange, remaining at the flange if equilibrium requires it. On the contrary, when multipoint contact option is selected, it will be possible to represent the appearance of two contact points simultaneously: one at the tread and another at the flange. In the last case, the total Q-force will be obtained by adding the Q-force of the tread and the Q-force of the flange.

![FIGURE 38: Schema of the contact points location and Q-forces calculation, for onepoint contact and multipoint contact](image)
Chapter 4: vehicle dynamics in a crosswind context

TABLE 15 shows a compilation of the Q-forces at the wheel where flange contact occurs, calculated with both methods. With onepoint contact the total load is almost the same as with multipoint contact.

<table>
<thead>
<tr>
<th></th>
<th>Wheelset 1</th>
<th>Wheelset 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Onepoint contact</td>
<td>$Q_{\text{contact}}$ 134,867</td>
<td>$Q_{\text{contact}}$ 144,674</td>
</tr>
<tr>
<td>Multipoint contact</td>
<td>$Q_{\text{tread}}$ 122,722, $Q_{\text{flange}}$ 12,153, $Q_{\text{total}}$ 134,875</td>
<td>$Q_{\text{tread}}$ 97,207, $Q_{\text{flange}}$ 47,332, $Q_{\text{total}}$ 144,539</td>
</tr>
</tbody>
</table>

TABLE 15: Comparison of the Q-forces at the contact calculated with onepoint and multipoint contact

TABLE 16 compares the maximum lateral displacement of the contact point with two contact configurations for the same load case from TABLE 15. It is easy to check that the onepoint contact locates the contact point somewhere between the locations of the two points of the multipoint contact. This is reasonable because equilibrium cannot be achieved in another way since the load is transmitted only through that point.

If lateral forces are checked, the conclusion is that Y-forces behave similarly to Q-forces. The total Y-force does not change between the two configurations.

<table>
<thead>
<tr>
<th></th>
<th>Wheelset 1</th>
<th>Wheelset 2</th>
</tr>
</thead>
<tbody>
<tr>
<td>Onepoint contact</td>
<td>Contact point 33.60</td>
<td>Contact point 35.37</td>
</tr>
<tr>
<td>Multipoint contact</td>
<td>Tread 31.15, Flange 37.40</td>
<td>Tread 31.43, Flange 37.68</td>
</tr>
</tbody>
</table>

TABLE 16: Comparison of the displacement of the contact point calculated with onepoint and multipoint contact

As the multipoint option includes more than one area of contact, vertical and lateral forces are calculated separately for each area. The contact force appears in each area as the contact point moves from one to another or when two contact points appear at any one time. At this point the question to answer is if the number of areas—one or two—has any effect in the wheel unloading calculation; that is to say, whether considering just the tread is equivalent to consider the tread and the flange.

TABLE 17 gives an example of the wheel unloading calculation using Chinese Hat (see Section 2.1.2). It compares the different possibilities that must be selected to set up the vehicle model. As stated previously, there is no difference between the constraint and elastic options. The difference appears in the multipoint contact option where there is the possibility of choosing the number of areas from which the wheel
unloading is calculated. The assessments indicate that neglecting the Q-force at the flange leads to an error, which is no higher than 5 per cent. Obviously, it is mandatory to remember that the calculation of the wheel unloading only from the Q-forces at the tread assumes that the Q-force at the flange is small. This is only true when aerodynamic loads are modeled according to Chinese Hat. In this wind scenario, flange contact takes place for a very short time because it is only reached when the maximum wind load hits the vehicle. Under this hypothesis, the load transmitted through the flange is rather small and thus it might be disregarded.

<table>
<thead>
<tr>
<th>Constraint</th>
<th>Onepoint</th>
<th>Multipoint (tread and flange)</th>
<th>Multipoint (tread)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic</td>
<td>0.8404</td>
<td>0.8445</td>
<td>0.8855</td>
</tr>
<tr>
<td></td>
<td>0.8410</td>
<td>0.8459</td>
<td>0.8834</td>
</tr>
</tbody>
</table>

TABLE 17: Comparison of the wheel unloading including contact types (constraint or elastic) and number of contact points (onepoint or multipoint). For multipoint, it shows the difference between the number of areas from which the ratio is calculated.

### 4.3.2 Calculating CWCs with both models

The problem arises when CWC curves must be calculated. It is well known that wheel unloading is a conservative criterion: reaching 90 per cent of wheel unloading does not mean the vehicle is just going to overturn. Carrarini, 2008 studies the probability of overturning in detail.

An example calculation of CWC is given in FIGURE 39. The scenario consists of the vehicle circulating on a straight track on the windward rail over a typical 6m embankment; CWC calculations in three different configurations are given:

- Three-mass model subjected to planar configuration forces \((F_y, F_z, M_x)\).
- Multibody model subjected to planar configuration forces \((F_y, F_z, M_x)\).
- Multibody model subjected to all components of wind gust loads \((F_y, F_z, M_x, M_y, M_z)\).

Concordance between the multi-body model and simplified model is obtained when the same loads are applied, and the deviation between them oscillates between 0-0.5 m/s, a reasonable result due to the fact that the sway test and vibration modes are quasi-equivalent. However, due to the elimination of momentum around the Y and Z axes, which contributes to vehicle unload, results differ when CWC is calculated with full sets of loads. Since the contribution of \(M_y\) and \(M_z\) to the overall wind load is higher when the vehicle slows, the disparity between the curves increases at lower vehicle velocities.
4.3.3 Calculating CWCs. A parametric study

One of the main goals of a wheel unloading calculation is to build the CWC curves of the vehicle, which characterize the vehicle’s behaviour under crosswind loads. This section presents the CWC curves when using the different hypothesis that are summarized in TABLE 18, as well as the differences from the so-called ‘reference case’, which is the full vehicle model which has all the aerodynamic force components and multipoint wheel/rail contact. In such a way, it will be possible to estimate the influence over $v_w$ when the load at the flange may be dismissed under the conditions which were presented in Section 4.3.1 under the ‘multipoint (tread)’ hypothesis. The distinction between load scenarios (planar state or full set of loads) contrasts the influence of the aerodynamic moments over the CWC calculation. In addition, the simplified model will also be contrasted with other cases. Calculations are made considering the same scenario of Section 4.2.

<table>
<thead>
<tr>
<th>Curve</th>
<th>Model</th>
<th>Contact type</th>
<th>Wind loads</th>
</tr>
</thead>
<tbody>
<tr>
<td>Reference</td>
<td>Multibody</td>
<td>Multipoint (tread and flange)</td>
<td>$F_y, F_z, M_x, M_y, M_z$</td>
</tr>
<tr>
<td>Case 1</td>
<td>Multibody</td>
<td>Multipoint (tread)</td>
<td>$F_y, F_z, M_x, M_y, M_z$</td>
</tr>
<tr>
<td>Case 2</td>
<td>Multibody</td>
<td>Onepoint</td>
<td>$F_y, F_z, M_x, M_y, M_z$</td>
</tr>
<tr>
<td>Case 3</td>
<td>Multibody</td>
<td>Multipoint (tread and flange)</td>
<td>$F_y, F_z, M_x$</td>
</tr>
<tr>
<td>Case 4</td>
<td>Three-mass</td>
<td></td>
<td>$F_y, F_z, M_x$</td>
</tr>
</tbody>
</table>

TABLE 18: Options considered for comparison to the ‘Reference case’ in CWC calculations. Results appear in FIGURE 40.
When the three-mass model was presented in Section 2.2.1, one of the most critical simplifications—which is also the cause of inaccuracy—was the elimination of the aerodynamic momentum around the Y and Z axes. That simplification leads to a level of inaccuracy higher than the error that results when considering no more than three masses in the vehicle representation.

Curve (A) shows the influence of the number of Q-forces accounted for the wheel unloading calculation, that is to say, in Case 1 the Q-force at the flange is dismissed. This simplification modifies $\nu_w$ approximately 1.0 m/s.

Curve (B) reflects the deduction from Section 3.2, in which we concluded that the choice between onepoint and multipoint did not affect the calculations of the Q-forces. Nevertheless, it must be remembered that when using onepoint contact, the contact path is not correctly modeled and therefore the actual dynamic of the vehicle will differ from the computer simulation results.

Curve (C) compares how the load state of the multi-body modifies the calculated $\nu_w$ and thus it evaluates the influence of the aerodynamic momentums $M_y$ and $M_z$. From this curve we can state that the results from the multi-body model are highly dependent on the choice of the forces introduced in the model. The difference between the planar load case ($F_y$, $F_z$, $M_x$) and the full load case (six components of the aerodynamic loads) results in 2.0-3.5 m/s in the estimations of $\nu_w$. 

FIGURE 40: Difference in results of CWCs when cases 1 to 4 are compared to the Reference case (see TABLE 18)
Chapter 4: vehicle dynamics in a crosswind context

Curve (D) shows that calculations with the 2D model leads to a difference of 2.0-3.0 m/s with respect to the multi-body model when simulating with multipoint contact.

By comparing Curve (A) and Curve (C) we deduct that the aerodynamic momentums have more influence on wheel unloading than the number of Q-forces from which it is measured. The difference between Curves (C) and Curve (D) is around 0.0-0.5 m/s; hence, if $My$ and $Mz$ are ignored, the 2D model is as accurate as the multi-body model because both give the same results.

4.4 Vehicles circulating on a curved track

The analysis of high-speed trains on crosswind is carried out on tangent track. The effect of the centripetal acceleration due to the curve can be added to calculate the wheel unloading and eventually reduce the vehicle speed if the wheel unloading ratio gets close to the limit value (90 per cent). There is a difference however between tangent track and curve circulation as in the later case due to the added effect of crosswind and curve centripetal acceleration the secondary lateral suspension might reach its limits and contact the lateral bumpstops. The goal of this section is to show that the circulation on curve presents a new scenario in which safety might be guaranteed (if the wheel unloading ratio is below 90 per cent) but the passenger’s comfort is highly jeopardized. Wind loads cannot overturn the vehicle but they are able to make the vehicle instable, which converts the wind into a comfort problem in addition to the already known safety (overturning) issue.

The analysis of the vehicle running on curve requires studying the problem in the 3D space, thus only the full multibody model can be used. The wind scenario was the so-called ‘double track ballast and rail’ described in Section 2.2.3. The track introduced in the model is fully described in TABLE 19. The curve radius and the curve cant were chosen according to the regulation given by the Spanish rail manager ADIF. The selected values are the minimum established by ADIF for the case of a new line with a nominal speed of 300 km/h, which is the vehicle velocity that is being used to study vehicle dynamics.

<table>
<thead>
<tr>
<th>Length of initial tangent track [m]</th>
<th>300</th>
<th>Total length [m]</th>
<th>6250</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length of the transition tracks [m]</td>
<td>470</td>
<td>Curve radius [m]</td>
<td>5350</td>
</tr>
<tr>
<td>Length of circular arc [m]</td>
<td>3600</td>
<td>Track cant [mm]</td>
<td>140</td>
</tr>
<tr>
<td>Length of tangent track at the end [m]</td>
<td>1410</td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

TABLE 19: Track layout of the curved track
The scenario begins and ends with a tangent track. In between, there is the curved section connected to the tangent tracks by a transition track where both the cant and the radius increase linearly. Concerning the wind direction, and in order to create the most critical loading case, the angle of attack of wind was set to 90 degrees along the complete scenario. Wind forces act in the opposite direction of the centripetal force and therefore the effects of both the wind and the curve are added. Finally, the Chinese Hat wind scenario begins and ends when the vehicle runs on the constant radius curve (see FIGURE 41).

Ride comfort was assessed according to the European Standard EN 12299:2009 (CEN, 2009). Wind scenario was considered a ‘discrete event’, and the corresponding section of the standard was applied. The time signals of the non-compensated acceleration were extracted from the simulations following the methodology explained in the standard. Two sensors provided the signals; one was located at the front-end of the passenger compartment above the bogie, whereas the other sensor was located in the geometric center of the same car. Time signals were filtered using a low-pass 2nd order Butterworth filter with an upper corner frequency of 2.52 Hz.

The results herein presented compare three load cases created by three different wind velocities: 20 m/s, 25 m/s and 30 m/s. These wind scenarios are compared to a fourth scenario without wind. The goal is to check the stability of the vehicle running on curve without wind and to show that the hunting movement does not appear until crosswinds reach certain speed.

FIGURE 42 shows the time signal of the wheel unloading ratio. The value of this ratio was close to 0.1 when the vehicle runs without wind. This measurement seems
reasonable since the value of the curve radius was set to keep the vehicle stable, with low values of wheel unloading ratio and derailment factor. Obviously, the wheel unloading increases with the wind speed, being 0.75 the maximum value; therefore, there is still certain margin until the limit value (0.9) is reached. The vehicle is stable with wind speed 20 m/s, however it ceases to be in this state somewhere near 22 m/s when the instability becomes noticeable (this is not represented in the figure). The time signal already shows the instability with a wind speed of 25 m/s that increases when the speed is 30 m/s. A Fast Fourier Transform (FFT) of the signals reveals that the frequency matches the natural frequency of the hunting movement of the vehicle.

![FIGURE 42: Wheel unloading ratio of the vehicle running on a curved track under different crosswind conditions](image)

FIGURE 43 represents the lateral non-compensated accelerations ($a_{xy}$) setting as a reference the no-wind scenario. Both the track layout and the vehicle satisfy the normative of maximum lateral accelerations since the restriction from the corresponding CEN standard is to keep accelerations below 0.6 m/s² for high-speed vehicles. The 20 m/s wind speed case increases the acceleration value but still within the limits, even though the maximum value is 0.9 m/s² when the gust of wind hits the vehicle. However this acceleration is not considered since instant values are not in the scope of comfort standards. The cases of wind speeds 25 m/s and 30 m/s make the average value of the accelerations to be above limits, and also the peak-to-peak values of accelerations to be high. Leaving aside the influence of frequencies on the comfort, peak-to-peak values are significant since passengers feel both the average value and the amplitude of accelerations. This means that the worst comfort indexes
are reached not only because of higher average values but also as a result of having higher amplitudes of the accelerations. Since for the cases of wind speeds 25 m/s and 30 m/s the peak-to-peak values increase 0.3 m/s² and 0.4 m/s², the comfort index will be substantially worse in comparison with the case of wind speed 20 m/s. According to the results of FIGURE 43, we can conclude that crosswinds can become also a source of instability that reduces significantly the comfort of the vehicle.

If the vehicle runs on tangent track, the sensor located in the front-end records the same acceleration level as the middle sensor does. However, in curved tracks, positions near the leading end of the passenger’s compartment have always worse ride comfort quality (see FIGURE 44). The reason for this behavior is that the leading bogie is always more instable, hence near positions to the front-end lead to higher peak-to-peak values. Taking as an example the 30 m/s wind speed scenario, the peak-to-peak value at the front-end of the carbody increases up to 1 m/s², which is a 0.6 m/s² higher in comparison with the value measured at the center of the carbody. It means that the increase of the peak-to-peak acceleration value equals the limit value of the non-compensated acceleration given by the CEN standard. Nevertheless and as could be expected, the average value of accelerations does not change between the front-end and the center of the carbody.
Chapter 4: vehicle dynamics in a crosswind context

The increase of the instability is easily observed in the next two figures, which represent the movement of the wheel-rail contact point. FIGURE 45 shows the displacement of the contact point at the wheel's tread and FIGURE 46 indicates whether there is flange contact or not. The coordinate system is defined with the Y-axis pointing to the flange, i.e. positive values denotes that the contact point is moving towards the flange. Further information of how the multibody software represents the contact can be found in FIGURE 38.

The contact point moves towards the interior of the curve when there is no wind loading the vehicle. Once crosswinds begin to load the vehicle, the contact point changes the direction, moving towards the flange. As previous figures showed, the 20 m/s wind speed case showed no hunting movement, and in fact, the contact point describes a stable movement along the whole scenario. FIGURE 45 shows that the larger the loads, the more instable becomes the contact point movement along the tread, with higher amplitudes as the wind loads increase. In the 30 m/s wind speed case, the contact point moves up to the limit of the thread, reaching flange contact (see FIGURE 46) at the instant of time when the gust of wind appears (around second 28). Since the wheel unloading ratio limit was not reached at any of the presented load cases, what we expect is to have severe flange contact at the time when wind loads are constant (between 12 s to 28 s and from 32s to 47 s) if larger loads are introduced in the model. When the wind speed is 30 m/s, the contact point already reaches the limit of the wheel’s tread, and as there is still margin to increase the wind
Chapter 4: vehicle dynamics in a crosswind context

loads until the wheel unloading reaches 0.9, the instability of the vehicle will also increase. The leading wheelset will describe an instable movement in which the wheels’ flanges at both sides will make contact with the rail.

FIGURE 45: Displacement of the contact point on the tread under different wind load conditions. Measures were taken on the left wheel of wheelset one

FIGURE 46: Measures of the flange contact on the left wheel of wheelset one under different crosswind conditions
The instability of the vehicle is due to the effect of the added stiffness of the lateral bumpstops. These elements have a non-linear stiffness characteristic (see FIGURE 19) so that the value of the stiffness depends on the lateral deformation of the suspension element. TABLE 6 showed that bumpstops have a +/- 25 mm clearance (stiffness 0 N/m). The element starts introducing stiffness to the model for large displacements than 25 mm and the bumpstop reaction force progressively increases until the suspension is blocked (limit of bumpstop deformation). The suspension cannot get deformed more than 60 mm (lateral deformation from centered position) as the lateral stiffness introduced by the element is very high (stiffness 230 000 N/m). FIGURE 47 shows that in the case of vehicle running on curve without wind, the lateral displacement of the bumpstop is 25 mm which means that the element is not working yet. For low wind speeds, such as 20 m/s, the vehicle does not hunt even though the lateral deformation of the bumpstop is substantially high (52 mm) and the bumpstop is already increasing the overall lateral stiffness of the vehicle. By increasing the wind speed to 25 m/s, the vehicle already hunts when the wind loads are constant and without having the lateral suspension blocked. For this wind speed, the bumpstop only blocks at the wind gust. However, the suspension is blocked during the whole wind scenario when the wind speed is 30 m/s. We will see in the next section that by increasing the maximum deformation of the lateral bumpstop, the vehicle becomes stable but this might be unfeasible due to gage clearance limits.

The crosswinds problem is mostly quasi-static and for those wind scenarios in which the vehicle does not yaw, the non-compensated accelerations could be calculated by
introducing the equivalent centripetal force into a simpler 2D model. Evidently, this approach would not represent the hunting movement that we presented in this section. Nevertheless, we checked that the following conclusions apply in the scenarios in which the vehicle does not hunt:

- The difference of the non-compensated acceleration value between the no-wind scenario and the wind scenario is the same in both tangent and curved track.

\[
\begin{align*}
\begin{bmatrix}
    a_{\text{tangent track, wind}} & -a_{\text{tangent track, no wind}}
\end{bmatrix} & =
\begin{bmatrix}
    a_{\text{curved track, wind}} & -a_{\text{curved track, no wind}}
\end{bmatrix}
\end{align*}
\]

(4.1)

- The non-compensated acceleration in curved track under crosswinds can be calculated by the following approximation:

\[
\begin{align*}
    a_{\text{curved track, wind}} + a_{\text{tangent track, wind}} & =
    a_{\text{curved track, no wind}} + a_{\text{tangent track, no wind}}
\end{align*}
\]

(4.2)

4.4.1 Influence of some suspension components on safety and comfort

The previous section concluded that crosswinds can also become a comfort issue. A few secondary suspension elements were analysed in order to assess their contribution to the vehicle’s comfort: the lateral bumpstop, the anti-yaw damper and the anti-roll bar. The study was done taking into account the data from the highest wind speed (30 m/s) of the previous section. The goal was to study whether the non-compensated accelerations felt by passengers were improved by modifying either the stiffness or the damping of any of these elements. Their influence on the wheel unloading ratio was calculated though it was expected to be low. Moreover, since the anti-roll bar and the bumpstop have an effect on the lateral displacement of the carbody, the movement of the carbody’s CoG was measured to ensure compliance with gauge requirements.

a) Deformation of the bumpstop

The standard configuration of this element sets the maximum deformation of the vehicle’s lateral bumpstop at 60 mm. If wind loads are high enough, the limit can be reached leading to an unexpected discomfort felt by the passenger. A suitable solution to this problem could be to increase the maximum deformation of the suspension element. If this change is applied, comfort improves but the counterpart is that the lateral displacement of the carbody increases. Thus, gauge problems can appear, and consequently the solution should meet both gauge and bumpstop deformation requirements.
The multibody model had four lateral bumpstops: two in each bogie. The results presented here refer to the bumpstop located in the front bogie because simulations showed that it was the most critical. The sensitivity analysis herein discussed consisted on modifying the stiffness characteristic of the bumpstop checking six different hypotheses. FIGURE 48 compares the stiffness characteristic of the standard configuration (suspension blocked at 60 mm) and the hypotheses of the limit cases (i.e. bumpstop blocked at 45 mm and 75 mm). The stiffness values of the bumpstop were not modified but the values of the lateral deformation corresponding to the lateral stiffness. In such a way, we were able to proportionally reduce—or increase—the three operating stages of the bumpstop. Hence, the lateral suspension clearance that allows the element to freely move, the stage at which linear increases of the deformation produced linear increases of the stiffness and the deformation that blocks the element. For example, by increasing the maximum lateral deformation of the element to 75 mm, we also increased the displacement where the lateral stiffness is zero (clearance) by 15 mm. We named the hypotheses by referring to the value of maximum deformation on the following discussion, assuming that the other working stages of the element were also modified.

FIGURE 49 represents the variation of the wheel unloading ratio after modifying the maximum deformation of the bumpstop from its nominal value (60 mm). The variations are small for both the tangent track and the curved track. Although the wheel unloading ratio is slightly more sensible to variations when the vehicle runs on tangent track, the differences does not seem very important.
By increasing the maximum displacement of the bumpstop, the behavior of the vehicle against crosswinds improve and the hunting movement is most likely to be avoided. However, standards limit the maximum displacement of the carbody to avoid gauge problems. FIGURE 50 shows the increase in percentage of the lateral displacement of the carbody’s CoG. The graph shows the increase with respect to the displacement of the carbody observed when the bumpstop can be deformed 60 mm. There are no differences between track layouts since the maximum deformation of the bumpstop is already reached when the vehicle runs on tangent track. Even by increasing the maximum allowed deformation, the limit is reached when the gust of wind loads the vehicle, in both tangent and curved track. However, the bumpstop does not reach the maximum deformation at the interval of time when wind loads are constant, when the limit of the bumpstop lateral deformation is set at 70 mm and 75 mm.
FIGURE 50: Increase of the lateral displacement of the carbody’s CoG as function of the maximum lateral deformation of the bumpstop

FIGURE 51 shows a comparison of the non-compensated acceleration signals obtained when the vehicle runs on curve. The chart compares the scenario when there are no crosswinds with the results of modifying the stiffness characteristic of the bumpstop for the 30 m/s wind speed case. In the latter case, the accelerations obtained with the nominal value (suspension blocked at 60 mm of lateral deformation) are compared with the effect of reducing the maximum deformation to 45 mm and also when the bumpstop blocks at 75 mm. By decreasing the allowed deformation of the bumpstop, we make the element to operate at deformations corresponding to higher stiffness values, which indirectly increases the overall lateral stiffness of the vehicle. Results show that comfort is drastically reduced when the blocking of the bumpstop is set to 45 mm as the peak-to-peak values of accelerations increase, being the maximum amplitude almost 1.6 m/s². In this case, the element operates during the whole scenario with the suspension blocked, increasing the amplitude of the hunting movement in comparison to the 'standard configuration' hypothesis. On the contrary, by allowing the bumpstop to get deformed up to 75 mm, the bumpstop operates at deformation values where the lateral stiffness is lower and the vehicle stops describing the hunting movement.
b) Anti-yaw damper

The two characteristics of the anti-yaw damper were checked, stiffness and damping. Although the stiffness and the damping were modified within commercial limits, the vehicle’s behaviour showed a small sensibility to this element. FIGURE 52 shows the result of increasing the damping of the element a 33 per cent, i.e., the damping was increased from its nominal value \(600 \text{ 000 (N·s)/m}\) to \(800 \text{ 000 (N·s)/m}\). The instability of the vehicle was reduced but, in the author’s opinion, it was not enough to consider it relevant. The wheel unloading and the lateral displacement of the carbody’s CoG were not influenced by these variations, as the anti-yaw damper cannot have effect on these parameters.
c) Anti-roll bar

This is another key element in the crosswinds topic since it has direct influence on the other characteristic movement, which is the roll of the carbody. The restoring movement is mainly a consequence of the carbody’s displacement and rotation. In the same way that bumpstops work controlling lateral displacements, anti-roll bars limit rotations. We modified the anti-roll bar by increasing the nominal value of its rotational stiffness \(1.5 \times 10^6\) N/rad·m. We checked the influence of two increments, 33 per cent and a 66 per cent, so that the new values of the stiffness were \(k = 2.0 \times 10^6\) N/rad·m and \(k = 2.5 \times 10^6\) N/rad·m respectively.

FIGURE 53 contains the calculated wheel unloading increase factor for each value of the stiffness. The wheel unloading decreases by making the anti-roll bar stiffer, showing a similar effect as when the maximum deformation of the lateral bumpstop was reduced. Being the maximum decrease of the wheel unloading ratio a 4 per cent, the influence of the anti-roll bar on this ratio seems not significant.
The lateral displacement of the carbody’s CoG as function of anti-roll stiffness is showed in FIGURE 54. The anti-roll bar had nearly the same impact over the CoG displacement as the lateral bumpstop. The displacement of the carbody's CoG showed a linear influence to changes of the anti-roll bar stiffness, similarly as what we observed with the maximum deformation of the bumpstop. We could establish a relation between the influence of increasing the anti-roll bar stiffness and the
deformation of the lateral bumpstop. Thus, in order to decrease over 6 per cent the lateral movement of the carbody’s CoG, we could increase by 33 per cent the anti-roll bar stiffness or decrease about 7 mm the deformation of the bumpstop.

The mean value of the non-compensated accelerations decreased when the stiffness increased (see FIGURE 55). By increasing the stiffness of this element, the rotation of the carbody decreases, reducing the lateral accelerations felt by the passenger. The decrease of the average value of the non-compensated accelerations is linear, as it was the influence of the element on the displacement of the carbody’s CoG. The reduction of the accelerations’ average value was quantified to be 0.09 m/s² for the case of curved track and 0.06 m/s² in the case of tangent track.

As a counterpart of decreasing the average acceleration felt by the passenger, the peak-to-peak value increased. Lower values of the rotational stiffness produce higher average values of the accelerations but lower peak-to-peak values (see FIGURE 56). Thus, the highest value of the peak-to-peak value was observed when the value of the rotation stiffness was set to $k = 2.5 \times 10^6$ N/ rad-m, although was also the case that leads to the lowest average accelerations value. Finally, the anti-roll bar stiffness has a low impact on a possible improvement of the stability of the vehicle since the hunting movement is a consequence of the lateral movement of the wheelsets. Consequently, the hunting movement can be observed for the three values of the anti-roll bar stiffness.
4.5 Conclusions

We quantified the differences that result from using a simplified 2D model rather than a full 3D multibody model to calculate the CWC. The differences arise from the fact that some suspension elements, as a consequence of a planar configuration, are not included in the simplified model: the sway link in the primary suspension and the yaw damper in the secondary suspension. Both models achieved similar results, and consequently wheel unloading simulations with the simplest model could provide sufficient precision.

What is important in terms of safety is that the wind unloads the wheel located at the leeside, which can lead to the overturning of the vehicle if the wind forces are strong enough. When the problem is studied in 2D, wheel unloading is calculated in terms of an averaged factor. In the simplified model, the factor that is calculated is one that represents the average of the four wheels of a typical two wheelsets bogie. However, the standard prescribes that the most representative bogie, and the most critical one, is the leading bogie. This study has checked that in the wind conditions considered the average calculated in the 2D model can accurately represent the safety clause established in the standard.
It has been shown that wind load is a key factor. Moreover, if $M_y$ and $M_z$ are ignored, the multi-body model contact gives the same results as the 2D model. Taking into account that these two components are not easy to measure in tests or to estimate by calculations, one could conclude that when only three components of the aerodynamic forces are introduced, it is not worth building a whole multibody model. Lastly, the difference between using onepoint or multipoint contact in the wheel/rail contact options does not change the final result of the wheel unloading ratio.

Crosswinds can also be cause of discomfort. Crosswinds increase the value of the non-compensated accelerations that can be above the limits set by the standards. Moreover, simulations showed that the vehicle tended to describe a hunting movement due to the effect of bumpstops stiffness/deformation characteristics on the lateral suspension. This instable movement produce high lateral accelerations and worse comfort indexes. The influence of several secondary suspension elements was checked. Vehicle dynamics were very sensible to the maximum deformation of the lateral bumpstop. The overall lateral stiffness of the vehicle can be modified by changing the characteristics of the bumpstops, showing that the amplitude of the hunting movement could be reduced or even avoided if the maximum allowed deformation was high enough. The stiffness and the damping of the yaw damper did not modify the vehicle behavior. Finally, we were able to reduce the main value of the lateral accelerations by modifying the anti-roll bar stiffness, but a cost of assuming a slight increase of the peak-to-peak values of accelerations.
Chapter 5

CFD approaches to study aerodynamics

Computational Fluid Dynamics (CFD) has confirmed to be a very valuable tool to study high-speed train aerodynamics, as the state of the art presented in Section 1.5 clearly shows. Two different CFD approaches have been considered in this thesis to study the crosswind flow over high-speed trains in different scenarios. On the one hand, the traditional eulerian approach has been considered with the use of the Finite Volume Method (FVM). This discretization method is doubtless the most widely used in vehicle aerodynamics simulations. On the other hand, the lagrangian approach has been considered with the use of the Lattice Boltzmann Method (LBM). This particle-based method is being established as an alternative to traditional CFD methods. One reason that explains the increased popularity of this method is that complex phenomena, like turbulence or multiphase flows, are treated naturally in its formulation. Another significant reason is that the method can be parallelized very easily and efficiently.

The crosswind flow over high-speed trains is always turbulent, thus it is very complex (transient and three-dimensional) and it is characterized by vortices of different sizes that continuously interact. These features make the computational simulation of the flow more difficult equally for the FVM and LBM methods. The simulation of such turbulent flows can be handled by different approaches, depending on the level of turbulence modeling that is adopted (see FIGURE 57). Direct Numerical Simulations (DNS), in which no turbulence model is introduced, accurately resolve vortices of all sizes and the flux of kinetic energy produced from the larger to the smaller vortices, in the so called energy cascade. However, this approach would require currently unattainable computational resources to resolve the smallest vortices found in the crosswind flow over high-speed trains. Large Eddy Simulations (LES) resolve vortices that are larger than a prescribed size ($\Delta_{LES}$) and introduce a model for the smaller vortices, which have a more universal behavior. Finally, Reynolds-Averaged Navier-Stokes Equations (RANS) is a statistical approach in which all scales of turbulence are modeled, and thus it only provides averaged flow quantities. This
Chapter 5: CFD approaches to study aerodynamics

The three approaches for turbulent flow simulations, namely DNS, LES and URANS, can be equally employed with both the FVM and LBM methods. In this thesis, however, URANS simulations were performed with the FVM and LES simulations with the LBM.

In the following, the two CFD approaches employed to study the crosswind flow over high-speed trains are presented. Description is by no means exhaustive, but helps the reader compare the two approximations. Finally, at the end of this section we show the validation of the CFD software based on the Lattice Boltzmann method.

5.1 A eulerian approach: URANS with FVM

In the eulerian description of a continuum field, physical quantities are defined as functions of position in space ($x$) and time ($t$). Therefore, the fundamental physical laws are expressed as partial differential equations in terms of the spatial coordinates and time. For the flow of uniform fluids, Navier-Stokes equations are known to govern the dynamics of the flow in both laminar and turbulent regimes. These fundamental equations can be expressed in tensor notation as
Chapter 5: CFD approaches to study aerodynamics

\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho \cdot u_i \right) = 0 \]  
\[ \frac{\partial}{\partial t} \left( \rho \cdot u_i \right) + \frac{\partial}{\partial x_i} \left( \rho \cdot u_i \cdot u_j \right) - \frac{\partial}{\partial x_j} \left( \rho \cdot u_i \right) - \frac{\partial p}{\partial x_i} - f_{V,i} = 0 \]

where Equation (5.1) is the continuity equation and Equation (5.2) is the momentum conservation. In the latter, the term \( f_{V,i} \) stands for body forces per unit volume acting on the fluid (e.g. gravitational forces).

The Navier-Stokes equations in Equation (5.1) and (5.2) are the most general equations that govern the flow of uniform fluids. However, they are usually space-filtered (in LES) or ensemble-averaged (in URANS) when the simulation of turbulent flows is concerned. With the latter procedures new unknowns arise that require new governing equations to be added.

As is well known, the equations governing the fluid flow, whether the most general equations or model-specific equations, cannot be solved analytically even in very simple cases. Therefore, it is necessary to discretize the differential equations and transform them into algebraic equations, so that they can be numerically solved by a computer. The Finite Volume Method (Patankar, 1980) is the most commonly used discretization method, and it is briefly explained below.

5.1.1 Turbulence modelling: URANS

The Unsteady Reynolds-Averaged Navier-Stokes Equations (URANS) approach is based on the so-called Reynolds decomposition, by means of which flow quantities are decomposed into an averaged component \( \Phi \) and a fluctuation component \( \phi' \) as:

\[ \phi(x,t) = \Phi(x,t) + \phi'(x,t) \]  

Different types of averages may be employed in the context of this statistical approach, depending on flow characteristics (steadiness, periodicity, etc). The so-called Ensemble average is the most general choice and it consist in averaging the quantity of interest over a large number of successive realizations (simulations or experiments):

\[ \Phi(x,t) = \lim_{M \to \infty} \frac{1}{M} \sum_{n=1}^{M} \Phi_n(x,t) \]  

where \( M \) is the number of realizations. The Ensemble average is a rather theoretical concept when computer simulations are concerned, but it allows arbitrary unsteady
flows to be computed from a statistical point of view. Moreover, the Ensemble average is equivalent to the Phase average.

$$\Phi(x,t) = \lim_{N \to \infty} \frac{1}{N} \sum_{n=0}^{N-1} \phi(x,t + nt_p)$$  \hspace{1cm} (5.5)

when periodic flows are analyzed (FIGURE 58). In the latter equation, $t_p$ is the time period and $N$ is the number of periods that are used for the averaging.

![Figure 58: Example of a decomposition of a generic flow variable, $\phi$](image)

The Unsteady Reynolds-Averaged Navier-Stokes (URANS) equations are obtained by introducing the Reynolds decomposition of velocity and pressure:

$$u_i(x,t) = U_i(x,t) + u'_i(x,t)$$  \hspace{1cm} (5.6)

$$p(x,t) = P(x,t) + p'(x,t)$$  \hspace{1cm} (5.7)

into the general Navier-Stokes equations, Equation (5.1) and (5.2), and subsequently applying the Ensemble average to these equations as:

$$\frac{\partial p}{\partial t} + \frac{\partial}{\partial x_j} \left[ p(U + u'_i) \right] = 0$$  \hspace{1cm} (5.8)

$$\frac{\partial}{\partial t} \left[ \rho (U + u'_i) (U + u'_j) \right] - \frac{\partial}{\partial x_j} \left[ \mu \frac{\partial (U + u'_i)}{\partial x_j} \right] + \frac{\partial}{\partial x_j} \left[ \rho (P + p') \right] - f_i = 0$$  \hspace{1cm} (5.9)

Since the Ensemble average satisfies the properties of constant conservation, linearity, associability of the averaging operator, and commutation with time and space derivatives, the equations resulting from the process explained above are:
\[ \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i} \left( \rho U_i \right) = 0 \] (5.10)

\[ \frac{\partial}{\partial t} \left( \rho U_i \right) + \frac{\partial}{\partial x_j} \left( \rho U_i U_j \right) = \frac{\partial}{\partial x_j} \left( \mu \frac{\partial U_i}{\partial x_j} - \rho \overline{u_i' u_j'} \right) - \frac{\partial P}{\partial x_i} + f_i \] (5.11)

\[ \tau_{ij} = -\rho \overline{u_i' u_j'} \] (5.12)

The term in Equation (5.12) is the Reynolds stress tensor, which represents the effect of turbulent fluctuations on the averaged flow. Each element of the tensor is defined by statistical correlations between velocity fluctuations which, considering the symmetry of the tensor, entails the addition of six new unknowns to the problem. Then, to close the set of equations, additional equations are needed. However, when formulating new transport equations for the elements of the Reynolds stress tensor, new unknowns appear; in such a way that it is impossible to close the set of equations by this procedure. This fact is known as the Turbulence Closure Problem, and makes necessary the introduction of physical models to achieve the closure of the set of equations. These models provide approximations to the Reynolds stresses, defining them in terms of the averaged flow variables; in such a way that the number of variables is reduced until it reaches the number of equations (Wilcox, 1998). Some turbulence models pose a transport equation for each Reynolds stress and then model the statistical correlations that appear in them. These models are known as Reynolds Stress Models. Other turbulence models, called Turbulent Viscosity Models, write the Reynolds stresses in terms of the averaged strain rates. All the simulations carried out in this thesis have been run using turbulent viscosity models (see Section 3.2.1b) and, consequently, their basis is detailed below.

a) Boussinesq hypothesis

Turbulent Viscosity Models (or Eddy Viscosity Models) model the Reynolds Stresses through \( \tau_{ij} \) the Boussinesq hypothesis (Equation(5.13)). This hypothesis assumes that there is an analogy between the molecular and the turbulence-induced momentum exchanges. Hence, the Reynolds stress tensor is proportional to the averaged strain rate tensor, the constant of proportionality being the same for all the elements of the tensor.

\[ \tau_{ij} = \mu_t \left[ \frac{\partial U_i}{\partial x} + \frac{\partial U_j}{\partial x} \right] \frac{2}{3} \rho \overline{k \delta_i} \] (5.13)

That constant of proportionality \( \mu_t \) is usually called turbulent viscosity and, unlike molecular viscosity, is not a thermodynamic property of the fluid but a function of the
flow characteristics. The second term on the right hand of the equation includes the turbulent kinetic energy \( k \), defined as twice the trace of the Reynolds stress tensor; and its addition guarantees that the new modeled tensor has the same trace as the actual tensor (Wilcox, 1998).

The goodness of Boussinesq hypothesis has been questioned in several works (Schmitt, 2007) by comparing its results with either experimental measurements or DNS or LES simulations. Even though the proportionality between the Reynolds stress and the averaged strain rate tensor is not fulfilled in most of the domain, the prediction of the averaged flow variables is quite satisfactory. In some complex flows featuring a strong rotation or developing on a highly curved surface, Boussinesq hypothesis is clearly erroneous and not even the averaged flow variables are correctly predicted. In these cases, Reynolds Stress Models usually provide better results than the Turbulent Viscosity models.

b) \( k-\varepsilon \) turbulence models

The main objective of Turbulent Viscosity Models is the adequate calculation of the turbulent viscosity, which is written in terms of different turbulent statistical variables. The two equations \( k-\varepsilon \) models employed in this thesis solve transport equations for both the turbulent kinetic energy \( k \) and the turbulent dissipation rate \( \varepsilon \). These statistical variables are defined as:

\[
\begin{align*}
k &= \frac{1}{2} \overline{u'_i u'_j} \\
\varepsilon &= \nu \frac{\partial u'_i}{\partial x_j} \frac{\partial u'_j}{\partial x_i}
\end{align*}
\]  

(5.14)  

(5.15)

where \( \nu \) stands for the kinematic viscosity. The turbulent viscosity can be calculated from the latter variables as:

\[
\mu_t = \rho \varepsilon C_s \frac{k^2}{\varepsilon}
\]

(5.16)

where the coefficient \( C_s \) can either be constant or a function of averaged flow variables, depending on the type of \( k-\varepsilon \) turbulence model adopted.

Both the turbulent kinetic energy and its dissipation rate are calculated solving a transport equation for each of them, in which some terms need to be modeled. The equation of transport for the turbulent kinetic energy results
Chapter 5: CFD approaches to study aerodynamics

\[
\frac{\partial}{\partial t} (\rho \cdot k) + \frac{\partial}{\partial x_i} (\rho \cdot k \cdot U_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_i} \right] + P_k - \rho \cdot \varepsilon \tag{5.17}
\]

where \( \sigma_k \) is a constant and \( P_k \) represents the production of turbulent kinetic energy. This latter term can be defined from the exact transport equation as:

\[
P_k = \tau_i \frac{\partial U_i}{\partial x_j} \tag{5.18}
\]

However, being consistent with the Boussinesq hypothesis, \( P_k \) is calculated as:

\[
P_k = \mu_t \cdot S_i^2 \tag{5.19}
\]

where,

\[
S = \sqrt{2 \cdot S_i \cdot S_j} = \frac{1}{2} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \tag{5.20}
\]

On the other hand, the exact transport equation for the turbulent dissipation rate (\( \varepsilon \)) is modified to include terms similar to those of the \( k \) transport equation:

\[
\frac{\partial}{\partial t} (\rho \cdot \varepsilon) + \frac{\partial}{\partial x_i} (\rho \cdot \varepsilon \cdot U_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{\varepsilon_1} \frac{\varepsilon}{k} - \rho \cdot \varepsilon - \rho \cdot C_{\varepsilon_2} \frac{\varepsilon^2}{k} \tag{5.21}
\]

where coefficients \( C_{\varepsilon_1}, C_{\varepsilon_2} \) and \( \sigma_\varepsilon \) are usually taken as constants. These two equations Equation (5.17) and Equation (5.21) make up the so-called Standard \( k-\varepsilon \) model, developed by Jones and Lauder, 1972 and later improved by Lauder and Sharma, 1974 among others.

In several works it has been stated that the \( \varepsilon \) transport modeling is a critical issue that entails significant modeling errors. Due to this interest, it has been object of intensive research and several turbulence models with different approaches on the \( \varepsilon \) transport equation have been proposed. One of the most significant examples is the Realizable \( k-\varepsilon \) model proposed by Shih et al., 1995, whose \( \varepsilon \) transport equation is:

\[
\frac{\partial}{\partial t} (\rho \cdot \varepsilon) + \frac{\partial}{\partial x_i} (\rho \cdot \varepsilon \cdot U_i) = \frac{\partial}{\partial x_i} \left[ \left( \mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_i} \right] + C_{\varepsilon_1} \frac{\varepsilon}{k} - \rho \cdot \varepsilon - \rho \cdot C_{\varepsilon_2} \frac{\varepsilon^2}{k + \sqrt{\varepsilon}} \tag{5.22}
\]

In this equation, coefficients \( \sigma_\varepsilon \) and \( C_2 \) have a constant value, whereas coefficient \( C_1 \) is calculated as:

\[
C_1 = \max \left[ 0.43 \frac{\eta}{\eta + 5} \right] \quad \eta = S \cdot \frac{k}{\varepsilon} \tag{5.23}
\]
Unlike Equation (5.20) and (5.22) is derived modeling the different terms of the exact equation what, according to (Shih et al., 1995), improves the predictive capacity of the model. Apart from the differences in the $\varepsilon$ transport equation, there is an additional special characteristic of the Realizable $k$-$\varepsilon$ model: the coefficient $C_\mu$ is not taken as a constant, but it is calculated from the strain rate and vorticity averaged tensors. In theory, by following this modeling for the calculation of turbulent viscosity negative values of the normal Reynolds stresses are avoided and the fulfillment of Schwarz inequality between Reynolds tangential stresses is fulfilled.

Another interesting approach is that of the Renormalization Group (RNG) $k$-$\varepsilon$ model proposed by Yakhot and Orszag, 1986. This model is derived from the instantaneous Navier-Stokes equations, using a mathematical technique called ‘renormalization group’ method. Through this method, the contribution of all turbulent scales to eddy viscosity is considered, resulting in a model with constants different from those in the standard $k$-$\varepsilon$ model, and an additional term in the transport equation for $\varepsilon$.

\[
\frac{\partial}{\partial t}(\rho \cdot k) + \frac{\partial}{\partial x_i} (\rho \cdot k \cdot U) = \frac{\partial}{\partial x_i} \left[ \alpha_e (\mu + \mu_t) \frac{\partial k}{\partial x_i} \right] + P_k - \rho \cdot \varepsilon \tag{5.24}
\]

\[
\frac{\partial}{\partial t}(\rho \cdot \varepsilon) + \frac{\partial}{\partial x_i} (\rho \cdot \varepsilon \cdot U) = \frac{\partial}{\partial x_i} \left[ \alpha_e (\mu + \mu_t) \frac{\partial \varepsilon}{\partial x_i} \right] + C_1 \frac{\varepsilon}{k} P_k - \rho \cdot \varepsilon \cdot C_2 \varepsilon^2 \tag{5.25}
\]

where,

\[
C_{z*'} = C_{z*} + \frac{C_{T^*} \left[ 1 - \eta / \eta_0 \right]}{1 + \beta \eta \eta^2} \tag{5.26}
\]

and coefficients $C_{z*}$, $C_{z*'}$, $C_{z*}$, $\beta$ and $\eta_0$ are taken as constants. The quantities $\alpha_k$ and $\alpha_e$ are the inverse Prandtl numbers derived analytically from the RNG theory. The effective viscosity $(\mu + \mu_t)$ is derived from a differential equation instead of using Equation (5.16), in such a way that the capabilities of the model to handle low-Reynolds-number flows and near-wall flows are improved.

c) Near wall turbulence treatment

The presence of solid walls in a flow introduces a boundary condition on the velocity field and all the magnitudes related to it, since the fluid particles in contact are enforced to have the same velocity as the wall. Therefore, when significant relative velocities between the flow and the wall exist, great velocity gradients appear in the direction perpendicular to the wall. When the flow is on the turbulent regime, the presence of walls affects all the statistical magnitudes of the flow, i.e. averaged
values of pressure and velocity, correlations between velocity fluctuations and correlations between pressure and velocity fluctuations.

The precise calculation of the flow near the walls is beyond the scope of the k-ε models presented so far. Therefore, the resolution of the near-wall turbulent flow was tackled in the simulations by including them in a Zonal model. This kind of models combine a conventional turbulence model with a Low-Reynolds model, being necessary the definition on some criteria to pass from one model to the other. One of the most used is that of Chen and Patel, 1969, in which a two-equation k-ε model is combined with the one-equation model of Wolfshtein, 1969. In this zonal model, the change criterion is given by the value of a characteristic Reynolds number calculated with the distance to the wall as characteristic length:

\[
Re_y = \frac{\rho \cdot y \cdot \nu_k}{\mu}
\]  

(5.27)

So, when \(Re_y > 200\) (far away from the wall) the two equation k-ε model is used and when \(Re_y < 200\) (near the wall) the Wolfshtein model is adopted. The latter solves only the transport equation for turbulent kinetic energy (Equation (5.17), where the dissipation of turbulent energy and the turbulent viscosity are calculated with simple algebraic expressions.

5.1.2 The Finite Volume Method

The Finite Volume Method (FVM) discretizes the governing equations by first dividing the spatial domain into a mesh of control volumes or cells. The types of elements commonly used in the discretization of the spatial domain are three or four-sided elements for two-dimensional geometries and hexahedrons, tetrahedrons, pyramids and prisms for the three-dimensional ones. The type of elements used depends mainly on the geometry of the problem and the desired kind of mesh, that is, whether it will be structured or unstructured. A structured mesh is characterized by regular connectivity; that is to say, each vertex of a cell is connected to a specific number of vertices from the neighbouring cells. An unstructured mesh, on the other hand, is characterized by irregular connectivity. The latter configuration, although leads to higher computational requirements in order to explicitly store the connections between elements, can be used in complex geometries.

The flow variables of the problem in each cell can be stored either in its vertices or in the centroids of the cell and its faces. The former approach is known as Node Based and the latter as Cell Based. The CFD code ANSYS Fluent employed in this thesis uses a Cell Based approach in unstructured meshes (see FIGURE 59).
Chapter 5: CFD approaches to study aerodynamics

The steady governing equations can be written in a generic form as:

$$\frac{\partial}{\partial x_j} \left( \rho \mathbf{u}_j \phi - \Gamma_\phi \frac{\partial \phi}{\partial x_j} \right) - \mathbf{S}_\phi = 0$$ (5.28)

its terms (in order) representing convection, diffusion and the difference between generation and destruction of the variable of interest $\phi$. In the FVM this equation is integrated for each cell giving:

$$\int \sigma \frac{\partial}{\partial x_j} \left( \rho \mathbf{u}_j \phi \right) dV - \int \sigma \frac{\partial}{\partial x_j} \left( \Gamma_\phi \frac{\partial \phi}{\partial x_j} \right) dV - \int \mathbf{S}_\phi dV = 0$$ (5.29)

where $V$ stands for the cell volume. Next, Gauss divergence theorem is applied to the first two terms, so that they are transformed to surface integrals as:

$$\int \left( \rho \mathbf{u} \cdot \nabla \right) \phi dA - \left( \int \Gamma_\phi \nabla \cdot \phi dA \right) - \int \mathbf{S}_\phi dV = 0$$ (5.30)

Finally, the integrals are approximated by assuming that all the integrands are constant and equal to the value of the corresponding face's or cell's centroid:

$$\sum_f \left( \rho f \phi f \mathbf{v} \cdot \mathbf{n} \right) dA - \sum_f \left( \Gamma_{\phi f} \nabla \cdot \phi \right) dA - \int \mathbf{S}_\phi dV = 0$$ (5.31)

In the latter equation the subindex $f$ makes reference to a face of the cell; and $\mathbf{v}$, $\mathbf{n}$, and $\mathbf{A}_f$ are the velocity and the face area vectors, respectively. This spatial discretization of the equations requires computing the variables at the centroid of the faces and the gradient of the variables at the face and cell centroids. As the values for all fields are stored at the centroid of each cell, the values at the cell faces must be interpolated from the surrounding cell. Once all the interpolation schemes are introduced in the transport Equation (5.31), it can be rewritten for a generic cell $C_0$ as:
\[
\left( \delta_{e,i} \right) \phi_e = \sum_{nb} \left( \delta_{e,i} \right) \phi_{nb} + b_e
\]  
(5.32)

where subindex \( nb \) makes reference to neighboring cells and \( b_e \) stands for the source terms and other independent terms derived from the interpolation. In order to do so, among all the interpolation schemes available in the literature, the following ones have been adopted for the simulations carried out in this thesis (Mathur and Murthy, 1997):

- Two different approaches based on the Gauss theorem are used for the evaluation of gradients at the cell centers, depending to how the values at the face centers are calculated.

  **Reconstructed gradient:** this kind of gradient is used in second order upwind schemes to calculate the values of the variables at the cell centers. It is calculated through the following expression, where \( \phi_i \) are computed through arithmetic average of the values at the centroids of the cells sharing face \( f \), \( V_0 \) is the cell volume and \( \lambda_0 \) is a limiter factor used to prevent spurious oscillations.

\[
\left( \nabla \phi \right)_{R,0} = \frac{\lambda_0}{V_0} \sum \phi_i \cdot x_i
\]  
(5.33)

**Regular cell gradient:** in this case the gradient is evaluated as:

\[
\left( \nabla \phi \right)_o = \frac{1}{V_o} \sum \phi_i \cdot x_i
\]  
(5.34)

where values of the unknown at the face centers are evaluated through arithmetic averaging of the reconstructed values:

\[
\phi_i = \frac{\phi_{i,o} + \phi_{i,1}}{2}
\]  
(5.35)

\[
\phi_{i,o} = \phi_e + \left( \nabla \phi \right)_{R,o} \cdot \delta_{f,o}
\]  
(5.36)

\[
\phi_{i,1} = \phi_1 + \left( \nabla \phi \right)_{R,1} \cdot \delta_{f,1}
\]  
(5.37)

- A second order centered scheme has been chosen for the discretization of diffusive terms. In this fashion, the gradient in the diffusive flux through a face \( f \):

\[
D_i = \Gamma_i \cdot \nabla_d \cdot \phi
\]  
(5.38)
is evaluated with the Regular cell gradients as:

\[ \nabla \phi_i = \frac{1}{2} \left( \nabla \phi_i^0 + \nabla \phi_i^1 \right) \]  

\[ \text{(5.39)} \]

- A second order upwind scheme has been employed for the discretization of convective terms. Thus, only the data from the upwind cell, i.e. the one from where the mass flow comes, is taken into account to calculate the value of the variable at the face centroid:

\[ \phi_i = \phi_{i_{up}} + \left( \nabla \phi_i \right)_{\text{up}} \cdot \delta s_{\text{up}} \]

\[ \text{(5.40)} \]

In the latter equation the subindex up makes reference to the upwind cell.

The continuity equation in a cell enforces the balance of the flow rates through each of its faces. Although this equation can be related to the generic transport equation Equation (5.28) by doing \( \phi = 1 \), its discretization entails some numerical issues that make necessary a special treatment. With this aim, an interpolation scheme similar to the one proposed by (Rhie and Chow, 1983) has been adopted.

Regarding the momentum equation, a staggered-grid based scheme (Patankar, 1980) has been adopted for pressure interpolation within the source terms of the momentum equation while the velocity derivatives and the turbulent kinetic energy are computed through arithmetic average of their values at the centroids of the neighboring cells. Eventually, the discrete momentum equation in the \( i \)-direction is given by:

\[ \left( a_w \right)_i \left( u \right)_{i} = \sum_{r=1}^{n_{\text{cell}}} \left( a_{wr} \right)_{i} \left( u \right)_{r} + \sum_{t=1}^{n_{\text{cell}}} \bar{A}_t \cdot \bar{e}_t + b_w \]

\[ \text{(5.41)} \]

where \( \bar{e}_t \) is the unit vector of the face local coordinates system in the \( i \)-direction. The discretized equations for the turbulent kinetic energy \( (k) \) and its dissipation rate \( (\varepsilon) \) have the form of Equation (5.32) once they are introduced the interpolation schemes explained above.

### 5.2 A lagrangian approach: LES with LBM

The eulerian approach based on the Finite Volume Method (FVM) can be classified as a ‘top-down’ approach since macroscopic phenomena are described in the continuum level by means of partial differential equations. On the other hand, the so-called ‘bottom-up’ approach is based on a microscopic, particle level governed by the equations of molecular dynamics, from which the continuum is reconstructed. Particle-based methods are lagrangian approaches to solve fluid flows, and they can be classified according to the level that they model. For example, Direct Simulation
Montecarlo (DSM) methods model the behavior of the fluid at the molecular level, whereas Smoothed Particle Hydrodynamics (SPH) and Vortex Panel Method (VPN) solve the equations at the macroscopic level. Intermediate between the two levels are the Lattice Gas (LG) and Lattice Boltzmann Method (LBM), which might be considered mesoscopic approaches. The main advantage of solving fluid dynamics at the mesoscopic scale is the affinity to computational calculations, i.e. the algorithms are efficient and can be easily programmed. However, it is complex to build the macroscopic level from mesoscopic laws.

In recent years, the LBM has developed into an alternative and promising numerical scheme for simulating fluid flows. The fundamental idea of the LBM is to construct simplified kinetic models that incorporate the essential physics of microscopic or mesoscopic processes so that the macroscopic averaged properties obey the Navier-Stokes equations (Chen and Doolen, 1998).

A short introduction of the LBM fundamentals is presented in this section. However, further information can be found in (Bhathagar et al., 1954; McNamara and Zanetti, 1988; Higuera and Jimenez, 1989; Chen et al., 1992; Quian et al., 1992).

5.2.1 A previous step prior to LBM: Lattice Gas Automata schemes

Lattice Gas Automata (LGA) is a kinetic model based on simple particle dynamics in discretized space and time that was developed to reproduce gas behavior. The basis of the method is that discrete particles move on a d-dimensional lattice in one of the predetermined b-directions, with a predetermined velocity \( c_i \) \( (i = 0, ..., b) \) at discrete times \( t = 0, 1, ... \) (FIGURE 60). The exclusion principle, which states that no more than one particle with a given direction is allowed at the same time in a lattice, is normally imposed for simplicity and memory efficiency.

In a LGA simulation, the state \( n \) of a lattice located at position \( r \) is given for time \( t \) by a set of Boolean variables called occupation number \( n_i(r,t) \) which have a value
n_i = 1 in the case of presence and n_i = 0 in the case of absence of a particle travelling in the i direction:

\[ n_i(τ, t) = \{ n_1(τ, t), n_2(τ, t), \ldots, n_m(τ, t) \} \] (5.42)

Moreover, the state \( N \) of the system consisting of \( m \) lattices may be defined for time \( t \) as the function:

\[ N(τ, t) = \{ n_1(τ, t), n_2(τ, t), \ldots, n_m(τ, t) \} \] (5.43)

The evolution equation for the LGA can be written as:

\[ n_i(τ + \tau_i dt, t + dt) = n_i(τ, t) + \Omega_i \{ n_1, \ldots, n_m \} \] (5.44)

where \( dt \) is the (integer) time step and \( \Omega_i \) stands for the collision operator, which for each state previous to collision \( (n_1, \ldots, n_m) \) returns the state after it \( (n'_1, \ldots, n'_m) \), conserving the global mass and momentum. Interestingly, the evolution process can be divided into two sequential steps:

- **Streaming**: each particle moves to the nearest lattice in the direction of its velocity. This step needs almost no use of the processor, since it is just a reorganization of the components of the \( N \) state.

- **Collision**: particles arrive at a lattice and interact by changing their velocity directions according to defined scattering rules. As this step takes place in each lattice separately, it can be parallelized very efficiently.

In this way, a LGA simulation consists in calculating the \( N \) state of the system in successive time steps by solving the evolution equation (Equation (5.44)) in the two steps described. Frisch et al., 1986 showed that Equation (5.44) with a suitable model of the collision operator leads to the Navier-Stokes equations in the macroscopic limit, provided that the underlying lattice possesses a sufficient symmetry. The essential requirements for the collision operator are that mass, momentum and energy have to be conserved:

\[ \sum \Omega_i = 0 \] (5.45)

\[ \sum \xi_i \Omega_i = \xi \] (5.46)

\[ \sum \tau_i \Omega_i = \tau \] (5.47)

Macroscopic flow properties can be directly obtained from the values of the occupation numbers \( (n_i) \) of each lattice. For example, the macroscopic density and
momentum may be calculated as the zero-order and first-order moments, respectively:

\[
\rho = \frac{1}{b} \sum_b n_i \quad (5.48)
\]

\[
\rho \cdot v = \frac{1}{b} \sum_b n_i \cdot c_i \quad (5.49)
\]

However, the Boolean nature of the occupation numbers \( n_i \) makes the distribution of the macroscopic properties very noisy. This undesirable effect can be alleviated by performing a coarse graining (space averaging), the process requires however an enormous amount of memory.

### 5.2.2 The Lattice Boltzmann method

Lattice Boltzmann methods (LBM) lay on LGA schemes explained above. The main motivation for the transition from LGA to LBM was the need to remove the statistical noise by replacing the occupation numbers \( n_i \) (Boolean variables) by single-particle distribution functions (real variables) \( f_i = \langle n_i \rangle \), where \( \langle \cdot \rangle \) denotes an ensemble average. Noise is erased with this replacement, because \( f_i \) is by definition an average, i.e., a smooth quantity. In LBM the distribution functions \( f_i \) develop in time according to a kinetic equation similar to the one governing LGA (Equation (5.44)):

\[
f \left( t + \nabla \cdot v \right) = f \left( t \right) + \Omega^B \left( f_1, \ldots, f_b \right)
\]

Because the kinetic form is still the same as the LGA, the advantages of locality are retained, which is essential to parallelism. It has been shown (Chen and Doolen, 1998) that Equation (5.50) can be obtained from the continuum Boltzmann equation for discrete velocities by using a small Mach number expansion.

\[
\left[ \frac{\partial}{\partial t} + \nabla \cdot v \right] f \left( t \right) = \Omega^B \left( f_1, \ldots, f_b \right)
\]

For the simulation of single-phase incompressible flows, the collision operator is usually linearized around its local equilibrium solution by using the Bhatnagar-Gross-Krook (BGK) model:

\[
\Omega^{BGK} = \frac{1}{\tau} \left( f_0^e - f \right)
\]

where \( \tau \) is the relaxation time and \( f_0^e \) is the local equilibrium distribution function. The relaxation time \( \tau \) is constant for all the defined directions \( (i=0, \ldots, b) \), thus the BGK model is also known as the Single-Relaxation Time (SRT) model. The equilibrium
distribution function is usually defined as a small Mach number expansion of the Maxwell-Boltzmann distribution with the form:

$$f_i^m = t_i \cdot \rho_{i} + c_i \cdot \nu_{\alpha} \cdot \Delta_s \cdot \left[ \frac{1}{2} \left( \frac{c_i \cdot c_i - \delta_{\alpha \beta}}{c_i^2} \right) \right]$$  \hspace{1cm} (5.53)$$

where $c_i$ is the speed of sound, $\nu$ the macroscopic velocity, $\delta$ the Kronecker delta, and $t_i$ are coefficients selected to preserve the isotropy in space. This election of the equilibrium distribution function allows the macroscopic Navier-Stokes equations in the nearly incompressible limit to be recovered from Equation (5.50) and Equation (5.51). The procedure is accomplished by means of the Chapman-Enskog expansion and gives rise to two fundamental relations. The first is the equation of state of an ideal gas:

$$P = \rho \cdot c_i^2$$ \hspace{1cm} (5.54)$$

which can be employed to calculate pressure $(P)$. The speed of sound $c_i$ is constant in LBM computations and its value is related to the type of lattice employed. Density, however, changes from point to point, although variations are very small in incompressible flow simulations. The value of the density is computed in the LBM, similarly to LGA (Equation (5.48), as the zero-order moment:

$$\sum f = \sum f_i^m = \rho$$ \hspace{1cm} (5.55)$$

The other important relation that arises from the Chapman-Enskog expansion is the one that relates the viscosity of the fluid and the relaxation time as

$$\nu = c_i \left[ t - \frac{1}{2} \right]$$ \hspace{1cm} (5.56)$$

In order for the viscosity to be positive, the relaxation time has to be greater than 0.5. This latter condition becomes problematic in the simulation of high Reynolds number flows, in which the SRT model suffers from numerical instabilities. These instabilities readily manifest themselves as local blowups and spurious oscillations (Brownlee et al., 2007).

The inherent limitations of the SRT model with regard to numerical stability and compressibility of the fluid can be avoided if more than one relaxation time is employed in the definition of the collision operator. In the multi-relaxation time (MRT) model proposed by D’Humieres et al., 2002, the collision process is carried out in the moment space instead of the velocity space. The collision operator is thus defined as:

$$\Omega^{\text{MRT}} = M \cdot \hat{S} \left( m^m - m \right)$$ \hspace{1cm} (5.57)$$
where $S_{ij}$ is a diagonal collision matrix, $m_{ij}^e$ is the equilibrium value of the moment $m_j$ and $M_{ij}$ is the transformation matrix. One particular advantage of this representation is that the time scales of the various processes represented in terms of moments can be controlled independently. Therefore, by choosing different and carefully separated time scales, the stability of the LBM can be significantly improved. Another advantage of the MRT model is that it allows specific schemes for simulating compressible flows to be developed. In this regard, Chen et al., 2010 proposed a scheme with which they performed simulations of compressible flow with strong shocks up to Mach 5.

The stability of the MRT models can be further enhanced if the collision operator is implemented in central moment space (Geier et al., 2006). For a raw moment defined as:

$$\mu_{x'y'm'} = \sum_i c_i^x c_i^y c_i^m$$

the corresponding central moment is:

$$\mu_{k'lm'} = \sum_i \left( c_i^x - u_i \right) \left( c_i^y - u_i \right) \left( c_i^z - u_i \right)^m$$

where $x, y$ and $z$ are the spatial directions, $k, l$ and $m$ are indices, and $c_i^\alpha$ and $u_\alpha$ with $\alpha \in \{x,y,z\}$ are the Cartesian components of discrete and macroscopic velocities, respectively. The order of the moment is referred by the sum $k+l+m$. By following this method, the relaxation process is performed in a moving reference frame by shifting the discrete particle velocities with the local macroscopic velocity. In this way the Galilean invariance is naturally improved and hence the numerical stability is increased (Premnath and Banerjee, 2011). The LBM-based code XFlow claims that it employs a MRT model in central moment space for the collision operator, so that it would benefit from this enhanced robustness.

### 5.2.3 Turbulence modelling in LBM

Since the kinetic equations of the LBM (Equation (5.50)) are shown to reproduce the hydrodynamics described by the Navier-Stokes equations, the latter method can be precisely employed to perform Direct Numerical Simulations (DNS) of turbulent flows. However, a large amount of very small lattices would be needed if the Kolmogorov scales of the turbulence were to be resolved in very high Reynolds number flows, such as those found in external aerodynamics. Therefore, the simulation with the LBM of the crosswind flow around a train requires some level of turbulence modelling. Due to the essential transient character of the LBM, the Large Eddy Simulation (LES) approach is a natural way of addressing turbulent flow simulations with LBM.
As was explained before, in the LES approach the larger scales of turbulence are resolved and the smaller scales are modeled. In the context of LBM-based LES, the Subgrid Scale (SGS) turbulence models account for the effect of the small unresolved scales by adding an extra viscosity, which in turn provides an extended relaxation time ($\tau$).

The Wall-Adapting Local Eddy-viscosity (WALE) SGS model proposed by (Nicou and Ducros, 1999) has been successfully employed for the simulation of turbulent flows with LBM-based LES (Weickert et al., 2010; Holman et al., 2012). This model computes the (extra) eddy viscosity in each lattice as

The software we used uses a LES model called Wall-Adapting Local Eddy-viscosity (WALE) who adjusts automatically the behavior near and far the walls. In addition, WALE distinguishes from laminar or turbulent regions. Wall functions will be needed to solve the flow near the walls and to represent boundary layers. In the WALE model the eddy viscosity is expressed as:

$$\nu_t = \Delta^2 \frac{\left( G_{\alpha\beta} G_{\alpha\beta}^d \right)^3}{\left( S_{\alpha\beta} S_{\alpha\beta} \right)^{5/2} + \left( G_{\alpha\beta} G_{\alpha\beta}^d \right)^5}$$

(5.60)

where Einstein’s summation convention has to be applied for the subscripts $\alpha$ and $\beta$. In the latter equation $\Delta$ is the filter scale, which is defined in terms of the size (volume) of the lattice as

$$\Delta = C_w \cdot \text{Vol}^{2/3}$$

(5.61)

with $C_w = 0.325$. Moreover, $S_{\alpha\beta}$ is the strain-rate tensor of the resolved velocity scale:

$$S_{\alpha\beta} = \frac{1}{2} \left( \frac{\partial u_\alpha}{\partial x_\beta} + \frac{\partial u_\beta}{\partial x_\alpha} \right)$$

(5.62)

and $G_{\alpha\beta}^d$ is the traceless symmetric part of the square of the velocity-gradient tensor ($g_{\alpha\beta}$):

$$G_{\alpha\beta}^d = \frac{1}{2} \left( g_{\alpha\beta} + g_{\beta\alpha} \right) - \frac{1}{3} \delta_{\alpha\beta} g_{\gamma\gamma}$$

(5.63)

$$g_{\alpha\beta} = \frac{\partial u_\alpha}{\partial x_\beta}$$

(5.64)

The most important feature of the WALE model is that the eddy viscosity goes naturally to zero in the vicinity of a wall, so that neither dynamical adjustment of the $C_w$ constant nor use of a damping function is needed. Moreover, the model produces
zero eddy viscosity in case of pure shear and thus it is able to reproduce the laminar
to turbulent transition process.

The eddy viscosity calculated by the WALE model is incorporated to LBM calculations
by defining an effective viscosity \( \nu_{\text{eff}} \), which is the sum of molecular and eddy
viscosities:

\[
\nu_{\text{eff}} = \nu_{\text{molecular}} + \nu_{\text{turbulent}} \tag{5.65}
\]

and using this effective viscosity to calculate the relaxation time as:

\[
\tau = \frac{\nu_{\text{eff}}}{c_s^2} + \frac{1}{2} \tag{5.66}
\]

Therefore, the locally (and temporally) varying eddy viscosity makes the relaxation
time to change from point to point of the domain. Furthermore, since it is always
positive, the eddy viscosity effectively stabilizes the numerical simulation. It has to be
pointed out that the eddy viscosity is incorporated equally for the SRT and the MRT
models of the collision operator, since in the MRT model the viscosity is related to
one of the relaxation times just the same as is related to the single relaxation time in
the SRT model (Equation (5.56)).

Although the WALE model is well-behaved and can be integrated down to solid walls,
its too expensive to fully resolve the thin boundary layers that are found near walls
at the typical high Reynolds numbers of aerodynamics applications. As such, Wall
Functions are needed to model the behavior of the flow at the inner layer close to
walls. The Generalized Wall Functions proposed by (Shih et al., 1999) make use of a
unified law of the wall that is valid for the whole inner layer, i.e. viscous sublayer,
buffer layer and inertial sublayer, and accounts for the pressure-gradient effects:

\[
\frac{U}{u_t} = \frac{\tau_w}{\rho \cdot u_t^2} f_1 \left( y^+ \right) + \frac{d \rho \cdot u}{d x} \frac{u}{u_t} f_2 \left( y^+ \right) \tag{5.67}
\]

In the latter equation, \( U \) is the mean velocity at a given distance \( y \) from the wall and
\( u_t \) is a velocity scale calculated from the skin friction velocity \( (u_t) \) and the
characteristic velocity of the pressure-gradient \( (u_p) \) as:

\[
u_t = u_t + u_p = \left( \frac{\tau_t}{\rho} + \left( \frac{d \rho \cdot u}{d x} \right) \right)^{1/2} \tag{5.68}
\]

where, \( \tau_w \) is the wall shear stress as in Equation (5.67) and the quotient \( d \rho \cdot u/dx \) is the
wall pressure gradient. The unified law of the wall has two interpolating functions, \( f_1 \)
and \( f_2 \), which are defined in terms of the dimensionless distance from the wall:
and the ratios of the characteristic velocities \((u_\tau, u_p)\) to the velocity scale \((u_c)\). Both interpolating functions are piecemeal (Shih et al. (1990)) and have been plotted in FIGURE 61.

The main advantage of using the unified law of the wall is that it provides a suitable boundary condition for velocity at wall-adjacent lattices, regardless of their central node’s location within the inner layer. The LBM-based code XFlow employs the unified law of the wall in two variants:

- **Enhanced wall function**: The effects of the pressure gradient are not taken into account and thus the second term in the right-hand side of Equation (5.67) is not calculated.

- **Non-equilibrium enhanced wall function**: Pressure gradient effects are taken into account and both terms in the right-hand side of Equation (5.67) are calculated.

Interestingly, the XFlow user’s manual points out that the Non-equilibrium enhanced wall function tends to estimate higher aerodynamic forces and more separation than the Enhanced wall function.

![Interpolating functions for the Non-Equilibrium wall functions (Shih et al., 1999)](image)

**FIGURE 61**: Interpolating functions for the Non-Equilibrium wall functions (Shih et al., 1999)

### 5.3 Validation of a CFD code based on LBM to simulate crosswinds over trains

CFD codes based on the LBM are not spread within the industry or in the scientific community, yet the current EN 14067:6-2010 considers using them as a feasible alternative to perform simulations. As every CFD code, even those based on the eulerian approach (such as ANSYS Fluent), they must firstly pass the validation process determined by the standard (see Section 2.1.1a). Firstly, the coefficient of
moment around the leeward rail \((C_{Mx,lee})\) must be calculated, being the expression herein applied in Equation (5.70). The standard validates the simulations that fulfills the recommendations in Equation (5.71) and (5.72).

\[
C_{Mx,lee} = C_{Rll} - \frac{C_{Llf}}{6} \quad (5.70)
\]

\[
\max \left( \frac{C_{Mx,lee} - C_{Mx,lee,brk}}{C_{Mx,lee,brk}} \right) < 0.15 \quad (5.71)
\]

\[
\text{mean} \left( \frac{C_{Mx,lee} - C_{Mx,lee,brk}}{C_{Mx,lee,brk}} \right) < 0.1 \quad (5.72)
\]

The standard provides the aerodynamic coefficients of the reference vehicles at the ground scenarios to be used as benchmark values \((C_{Mx,lee,brk})\). The average value of the \(C_{Mx,lee}\) calculated with the CFD software has to be below 10 per cent in order to pass the validation, allowing instant maximum differences of 15 per cent.

The aerodynamic coefficients in the whole range of yaw angles (between 0 and 90 degrees) are available in the standard. Nevertheless, high-speed trains operate at yaw angles below 45 degrees due to the combination of the wind speed and the vehicle speed, which is usually above 250 km/h. At low angles up to 15 degrees, the calculations would approximately correspond to drag tests where the main component of wind is parallel to the vehicle. Thus, high-speed trains subjected to crosswinds are usually observed to be working between 15 degrees and 45 degrees. The behavior of the train with regard to crosswinds changes at 45 degrees. From this yaw angle on, the train starts to behave as a bluff body, which can be clearly observed above 60 degrees (see FIGURE 6).

The validation herein presented was only done for the angles of attack that were relevant for the purposes of this thesis, considering two ground scenarios. The flat ground with 235 mm gap (see Section 2.2.2) was chosen due to the easiness of the scenario configuration, as it only requires the geometry of the train. The angles of attack under study were 15, 30 and 45 degrees since they were observed to be the ones that indicated a change in the flow pattern (see FIGURE 6). In addition, the double track ballast and rails (see Section 2.2.3) was also tested aiming to simulate more realistic ground scenarios and to later use it in the simulations with wind fences. In order to compare the results using the double track with the flat ground scenario, we simulated the flow at 30 and 45 degrees. Moreover, the flow at 35 degrees was also solved in order to check the coefficients’ trends between 30 and 45 degrees. Lastly, the 90-degree case was solved for the subsequent assessment of the sheltering effect of wind fences.
The validation of the LBM code is focused on the level of accuracy that the software can provide from the point of view of reproducing wind conditions that produces wheel unloading ratios close to the ones provided by the standard. As Section 4.3 showed, the overturning moment created by wind is mainly a consequence of both side and lift forces, and the roll moment. This is important from the point of view of CFD simulations since they need to essentially provide accurate results for these three components of wind forces. Yaw and pitch moments are both less significant when the key is guaranteeing safety since their contribution to the wheel unloading is much less important. The coefficient that takes into account the contribution of the aerodynamic forces and moment to the roll of the vehicle is $C_{M_{x,lee}}$, i.e. the value of the coefficient of roll around the leeward rail (see Equation (5.70)).

The following sections describe the results obtained from the simulations in terms of the LBM-based code XFlow. The analysis emphasize on the numeric results and the description of the flow pattern. FIGURE 62 shows the cutting planes that were applied to show the flow at different sections of the leading vehicle. These planes allowed us to show the detachment of the vortex at positions close to the nose ($X = -8 \text{ m}$), the development of the vortex at the origin of coordinates ($X = 0 \text{ m}$) and at positions close to the end of the leading car ($X = 10 \text{ m}$).

FIGURE 62: Cutting planes to show the flow pattern

5.3.1 Flat ground with 235 mm gap

This scenario was simulated since it was straightforward to recreate the conditions from the wind tunnel, as the configuration only requires to elevate the cars geometry 235 mm. The comparison was made for the angles of attack 15, 30 and 45 degrees to catch the different flow conditions that these yaw angles create. The results in Schober et al., 2010 are also included in this section since the authors replicate the standard experiments in a different wind tunnel with a similar setup.

CFD simulations were able to represent the overall trends of the whole set of coefficients, both the force and the moment coefficients, as FIGURE 63 shows. The six coefficients behaved similarly which allowed us to conclude that the CFD simulation introduced an error evenly distributed to all the forces and moments. Differences grew with the yaw angle, showing the largest disparity between tests and simulations.
in the case of 45 degrees and the lowest for the case of 15 degrees, being the case of 30 degrees somewhere in the middle.

Results showed an overestimation of the lift force coefficient and an underestimation of the side force coefficient. Therefore, a transformation of lateral force into lift force took place which approximately generated an equivalent force state to the one measured in the wind tunnel tests. Schober et al., 2010 also measured higher lift forces and lower side forces, although in their case the difference with the coefficients from the standard was lower. Other authors that have previously done CFD simulations, such as Diedrichs, 2003, already reported this compensation effect with coefficients that do not match between experimental tests and simulations. Nevertheless, their RANS simulations produce the opposite effect, i.e. an overestimation of side forces and an underestimation of lift forces. In addition, the error introduced is rather small in comparison with our results.

The coefficients of aerodynamic moment behaved analogously to the aerodynamic force coefficients. The prediction of the roll moment was key since the calculation of the moment around the leeward rail highly depends on this value. The roll coefficient followed the trend of the side force; hence, the absolute value of the computational...
results was always lower than the benchmark data. We observed that this coefficient only matches with the value of the standard when the side force coefficient was also accurately computed. The software precisely computed the coefficient of the yaw moment for the three yaw angles under study. It was difficult to find a relation between the aerodynamic forces and the pitch moment. However, its value was set somewhat from both the drag and the lift forces. In this way, the difference between the experimental tests and simulations grew proportionally to the differences between these force coefficients.

![Graph with Cmx,lee values and epsilon vs. yaw angle](image)

**FIGURE 64:** Comparison of the \( C_{mx,lee} \) and the error of the coefficients from FIGURE 63 between the results from wind tunnel tests provided by the standard and Schober et al, 2010, and the computed coefficients with CFD. Ground scenario: flat ground with 235 mm gap

According to Equation (5.70), the prediction of the roll coefficient around the leeward rail was more accurate in the cases with correct predictions of the roll coefficient. FIGURE 64 shows that the error on \( C_{mx,lee} \) increased with the yaw angle, which was mainly a consequence of the failure on the calculation of the roll moment. Equation (5.72) was applied to validate our results in terms of the EN 14067-6:2010, showing that none of the simulations at the flat ground with gap would pass the validation since the mean value of epsilon was always above the limit value, 0.1. The reason of getting the highest value of epsilon at the yaw angle with the most accurate prediction of \( C_{mx,lee} \) can be found on the definition of the value. The standard defines the maximum error as function of the value of \( C_{mx,lee} \), which indirectly sets severer conditions for those yaw angles with lower values. Results from Schober et al., 2010 revealed that the coefficients from the standard are hard to be exactly replicated but measurements within the established tolerances are somewhat feasible to get.

FIGURE 65 shows the vortex at the leeward side of the vehicle which definitely determines the flow around the vehicle. The diameter and the vorticity of the vortex increase with the yaw angle, which makes it to have greater influence on the flow at the larger angles of attack. In fact, the vortex imprints the pressure field at the leeward side of the vehicle and it has a great influence on the pressure at the top of the vehicle (see FIGURE 66). The detachment point of the vortex changes with the value of the yaw angle. FIGURE 65 shows that the vortex detaches by the end of the leading car in the case of 15 degrees, and the detachment of the vortex took place at
positions closer to the nose of the vehicle as the yaw angle increases. Moreover, the angle formed between the vehicle and the vortex once the latter is completely detached approximately matches the value of the wind yaw angle.

FIGURE 65: Three-dimensional view of a iso-surface of the mean pressure field colored with values of the vorticity \([s^{-1}]\) for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap

FIGURE 66 shows the contours of the mean pressure. The low-pressure region at the leeward side corresponds to the vortex that was previously shown in FIGURE 65. It was also quite evident that the vortex diameter grew with the angle of attack. The vortex created a low pressure area that was the main cause of generating the side forces and the roll moment. Hence, a better behavior against crosswind is got by providing the conditions to have the vortex detaching as close as possible to the end of the vehicle. The vehicle starts acting as a bluff body at 45 degrees, and the pressure field indicates this behavior. The field shows high-pressure values at the windward side of the vehicle and very low-pressure values at the top of the vehicle. These phenomena working together create the highest side and lift forces of the three cases. When the vehicle is placed at more aerodynamic angles, i.e. the 15-degree case, the pressure differences between the two sides of the vehicle and between the top and the bottom tend to have similar values.
FIGURE 66: Contours of the mean pressure field [Pa] at the positions showed in FIGURE 62 for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap

FIGURE 67 shows the mean velocity field and FIGURE 68 the vectors of the averaged velocity field. They indicate that low-pressure areas obviously corresponded with higher velocity values, which were located either on the vortex or on the top of the vehicle. The flow underneath the vehicle accelerated at higher yaw angles since the airflow reached the cross-section between the bottom of the vehicle and the ground more perpendicularly. It may be also pointed out that the flow on the top reached the leeward side with higher velocity when it did not detach from the vehicle. Lastly, we observed larger differences between the yaw angles at the rear sections (X = 10 m), mainly produced due to the manner in which the vortex develops.
FIGURE 67: Contours of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap

FIGURE 68: Vectors of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 15, 30 and 45 degrees. Ground scenario: flat ground with 235 mm gap
The averaged value of pressure on the vehicle surface is shown in FIGURE 69. We observed that the 15-degree had similar pressure values at the leeward and the windward sides, which was the cause of having aerodynamic coefficients with low values. The pressure differences between the vehicle sides increased with the yaw angle, thus creating higher aerodynamic coefficients. The pressure at the surface reached minimum values when the vortex was still close to the vehicle's surface since the flow was turning and slipping on the coach surface with high velocity. This effect was more remarkable in the 45-degree yaw angle case, being clearly observed at the windward side of the vehicle. The phenomenon was also present with the other two yaw angles but the pressure values were higher since the flow velocity reached lower values. Furthermore, the vortex produced a low-pressure strip on the surface of the vehicle when the vortex detached, which was parallel to the direction followed by the vortex at the wake. This was a sign of the great influence that this phenomenon has on the aerodynamic coefficients of the vehicle. Hence, as long as the process of detachment and evolution of the vortex is correctly predicted, the aerodynamic coefficients are also accurately calculated.

Although trials are not included in the current text, the following explanation is required to understand the problems encountered during the validation process. We observed that the code represents faithfully the simulations close to pure drag when the wind hits longitudinally the train. However, as the yaw angle increases, the code failed to faithfully represent the detachment of the characteristic vortex that appears at the leeward side of the vehicle (see FIGURE 65). This phenomenon takes place in the inner layer, which means that the wall boundary condition for the surfaces of the vehicle is accountable for predicting accurately the vortex detachment. The wall boundary layer is also responsible for the calculation of the pressure value on the surface of the vehicle since LES simulations only model the small eddies of
turbulence. The code under study was significantly modified during the realization of this thesis, being the Non-equilibrium wall function (see Section 5.2.3) one of the parameters that was improved during this time. FIGURE 70 shows the instant values of pressure at the coach surface calculated with the previous version of XFlow (89.03). Large values of pressure appeared close to locations with low-pressure values as a consequence of implementation errors in the wall boundary condition. The current version of XFlow (90.00) partially solved this error, which allowed us to reduce the overall differences with the benchmark results around 10 per cent. In addition, the requirements to produce the wake detachment in terms of lattice number also decreased, allowing us to reduce the minimum size of the lattice on the coach surface. We expect that next updates of the software will definitely solve this issue, decreasing more the error of the simulations and showing a better behavior on the prediction of the vortex detachment.

5.3.2 Double track ballast and rail

This scenario provides more realism since the computational model also includes a real representation of the track. The vehicle was located on the windward track where it is known to withstand larger aerodynamic loads. The simulation of this configuration, which is explained in 2.2.3, requires more computational time since the track also needs to be discretized, which substantially increased the number of elements. We simulated more yaw angles than in the case of the flat ground with gap since the tests with wind barriers (see Chapter 6) included this ground scenario. The simulated yaw angles were 30, 35, 45 and 90 degrees.

The aerodynamic coefficients of the leading car are shown in FIGURE 71, which have similar values and trends to the flat ground with gap scenario. The level of accuracy provided by the CFD code decreases as the yaw angle increases, following the
expected trend at the yaw angles of 30, 35 and 45 degrees. In the cases of 30, 35 and 45 degrees—similarly to the flat ground with gap scenario—the computed coefficients of side force and roll moment were lower than those in the CEN standard. On the contrary, the computed lift force was slightly higher than the experimental value, which puts closer the force state of simulations and tests. Results from the 90-degree case did not share the conclusions of the smaller angles of attack since the vehicle herein behaved as a more-or-less bluff body. The latter term is applied to those bodies having bluff shapes with rounded corners where the point of flow detachment changes with the flow conditions. The 60-degree yaw angle sets a trend change as the coefficient values decrease for higher yaw angles (see FIGURE 6). Above 60 degrees, the vortex of the leeward side does not appear, which was the main cause of having larger aerodynamic forces for larger yaw angles. In the 90-degree case, the CFD code was not able to correctly predict the values of the coefficients of both the drag and the side forces, and the roll moment. The absolute value of both the side and roll coefficients were higher than the coefficients at 45 degree. However, the coefficients at 90 degrees were expected to be substantially lower, as the measurements from experimental tests show. The side force, hence the roll moment, was overestimated showing the largest level of inaccuracy of the yaw angles under analysis. In addition, some of the results corresponding to 90 degrees showed further differences with those of the lower yaw angles. The computed coefficients of side force and roll moment were lower in comparison with the experiments, and the coefficient of lift force got with CFD was lower than its equivalent on the standard.

The prediction of the $C_{mx,lee}$ coefficient, which FIGURE 72 shows, was slightly enhanced when we included the track in the scenario since the flow conditions at the simulations were closer to the wind tunnel experiments, mainly due to a proper simulation of the flow underneath the train. The calculated values of epsilon showed lower values than those observed in the flat ground with gap scenario for the cases of 30 and 45 degrees (see FIGURE 63). In fact, the simulations corresponding to both 30 and 35 degrees could pass the validation proposed by the CEN standard since the epsilon value was lower than 0.1. The 90-degree case is poorly predicted due to the reasons explained in the previous paragraph. Being close to 0.3, this angle of attack set the maximum value of epsilon from the cases that were studied. The explanation can be mainly found on the coefficient of roll around the leeward rail since its value is almost 1.5 times higher than the value of the coefficient provided by the standard.
FIGURE 71: Comparison of the aerodynamic coefficients of a ICE 3 between the results from wind tunnel tests provided by the standard and the computed coefficients with CFD. Ground scenario: double track ballast and rail

FIGURE 72: Comparison of the $C_{mx,lee}$ and the error of the coefficients from FIGURE 70 between the results from wind tunnel tests provided by the standard and the computed coefficients with CFD. Ground scenario: double track ballast and rail

FIGURE 73 shows the vortex at the leeward side of the vehicle; by comparing these results with FIGURE 65 we realized that the general aspect of the vortex was similar with both ground scenarios. The vortex detached approximately at the same distance from the nose so that the flow fields were similar. Therefore, the variations introduced by changing the ground did not substantially produce changes in the boundary conditions that were able to modify the flow topography. This reason...
Chapter 5: CFD approaches to study aerodynamics

partially explained why both ground scenarios showed similar levels of precision in the computation of the aerodynamic coefficients.

FIGURE 73: Three-dimensional view of a iso-surface of the mean pressure field colored with values of the vorticity \( [s^{-1}] \) for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail

FIGURE 74 shows the average pressure fields at the cutting planes that were already introduced in the section regarding the flat ground with gap. The topography of the flow in the double track was very similar to the case of flat ground. One of the main differences was that the flat ground created flow conditions to have the vortex slightly closer to the vehicle geometry. The track was mostly able to modify the flow underneath the train, whereas it did not have a great influence on the vortex detachment, since this phenomenon took place several meters above the ground. A small vortex induced by the track shape appearing at the left bottom corner of the vehicle would be the main difference between ground scenarios. It had however a low influence on the overall pressure field, hence on the aerodynamic coefficients of the vehicle. The flow was also comparable to the flat ground at the windward side, where the highest values of pressure could be found.

The point at which the flow accelerates to overpass the top of the vehicle kept the same position between the different sections of the vehicle, regardless of the yaw angle (see FIGURE 74). The flow at the top of the vehicle detached closer to the top right corner of the vehicle at sections located further from the nose \((X = 0 \text{ m and } X = 10 \text{ m})\). The vortex was able to induce the detachment of the flow over the top of the vehicle as the value of its diameter increased, which partially set the pressure
value on the top of the vehicle. Hence, the pressure on the vehicle surface had lower values at positions near the nose (X = -8 m) as long as the flow slipped from the windward to the leeward side of the vehicle without getting detached.

![Pressure Contours](image)

FIGURE 74: Contours of the mean pressure field [Pa] at the positions showed in FIGURE 62 for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail

The mean velocity fields are shown in FIGURE 75, and the velocity vectors are shown in FIGURE 76. Higher velocity values were observed at the perimeter of the vortex, which accelerated the flow at both yaw angles. As FIGURE 75 shows, the contour of the vortex can be checked in yellow color at the yaw angle of 30 degrees and 45 degrees. A stagnation area with low velocities values appeared at the middle section of the train (X = 0 m) that was created in the confluence between the vortex and the flow overpassing the vehicle over the top. The small vortex at the bottom of the leeward side appears at sections close to the nose (X = -8 m) but it has a short development since the predominant vortex was able to diffuse it. FIGURE 76 confirms this statement as the velocity vectors do not reveal any vortex at the middle section (X = 0 m) despite the vortex is fully shaped in the section X = -8 m.
FIGURE 75: Contours of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail

FIGURE 76: Vectors of the mean velocity field [m/s] at the positions showed in FIGURE 62 for the yaw angles 30 and 45 degrees. Ground scenario: double track ballast and rail
Chapter 5: CFD approaches to study aerodynamics

Figure 77 shows what we already pointed out in the discussion regarding the pressure fields around the vehicle in Figure 74. Calculations of the pressure field on the vehicle surface showed that the distribution was almost identical in both scenarios, which is the reason for obtaining similar aerodynamic coefficients values too. The flow at the surroundings of the vehicle substantially changed between sections but this is not reflected on the pressure field on the vehicle surface. The strip in blue color at the windward side of the vehicle confirms that the height at which the flow accelerated to overpass the vehicle remains constant along the whole longitudinal dimension of the vehicle.

5.3.3 A comparison of wheel unloading calculations

Several sections of this thesis stated that the study of crosswinds is a multidisciplinary subject that requires both the dynamic and the aerodynamic analysis to fully capture the problem. Describing the coefficient variations and the implications of the flow do not completely explain the problem since the influence on the dynamic calculations is not considered. This section partially answers the question in regard to the influence of computed coefficients in terms of the LBM on the vehicle dynamics. The approach herein used was to calculate the wheel unloading ratios using both the computed and the experimental coefficients from EN 14067:6-2010 to assess the influence of the differences observed in the flow. The full multibody model described in Section 3.1.1 was needed to introduce all the wind loads components, even though the 2D components mainly create the aerodynamic overturning moment. The calculations were done for a hypothetical wind case on which the vehicle runs on a tangent track at 200 km/h and the wind speed is 20 m/s.

Figure 78 shows the wheel unloading ratios for the case of the flat ground with 235 mm gap. The wheel unloading ratio similarly behaved as the $C_{mx,lee}$ coefficient, getting larger differences for the higher yaw angles. By calculating the epsilon of the wheel unloading ratio similarly as we did with $C_{mx,lee}$ in Equation (5.72), we observed that the epsilon of the wheel unloading ratio is almost identical to the epsilon of the
Chapter 5: CFD approaches to study aerodynamics

aerodynamic coefficients (see FIGURE 63), as the difference between both values is just 1 per cent.

![Graph showing wheel unloading ratio and epsilon comparison](image)

FIGURE 78: Comparison of the wheel unloading ratio (up) and its epsilon (down) using both the experimental and the computed coefficients in the flat ground with gap scenario

The wheel unloading ratios and the corresponding epsilons for the double track ballast and rail scenario are shown in FIGURE 79. The trend of the wheel unloading ratio using the computed coefficients reproduced that of the dynamic simulations which included the experimental coefficients, similarly to the flat ground scenario and for the cases of yaw angles up to 45 degrees. The exception was the 90-degree yaw angle case where the value of the wheel unloading ratio almost matched the value of the 45-degree angle case. The CFD computations failed to predict the value of the roll moment at the angle of attack of 90 degrees. The overestimation of the computed roll moment made the coefficient to reach the value of the 45-degree case, which strongly influenced the wheel unloading ratio. Unlike the flat ground scenario, the calculation of the epsilon ratio revealed that the differences with the epsilon that corresponds to $C_{mx,lee}$ were slightly higher (see FIGURE 71). In the case of the double track ballast and rail, a 5 per cent difference was observed at the yaw angles of 35 and 90 degrees, being for the cases of 30 and 45 degrees close to 2 per cent. These slightly higher differences were due to the yaw and pitch moments which were not included in the calculation of $C_{mx,lee}$ but they were introduced in the full multibody model. In any case, these simulations showed that the epsilon of just $C_{mx,lee}$ and the epsilon of the wheel unloading ratio are very similar. This is significant since a calculation of $C_{mx,lee}$
can allow us to estimate the error of the wheel unloading ratio without performing any simulation with the vehicle dynamic model.

![Comparison of the wheel unloading ratio (up) and its epsilon (down) using both the experimental and the computed coefficients in the double track ballast and rail scenario.](image)

**FIGURE 79:** Comparison of the wheel unloading ratio (up) and its epsilon (down) using both the experimental and the computed coefficients in the double track ballast and rail scenario

## 5.4 Conclusions

The theory of the eulerian and lagrangian approaches employed to perform computational aerodynamic simulations was presented. A CFD code based on the LBM was validated in the range of yaw angles of interest to the study of crosswinds over high-speed trains. Calculations show that LBM codes can provide results with a reasonable error that might be reduced using smaller lattices in the set up of the simulations. The difference between the coefficients calculated with CFD simulations and the experimental tests available in the CEN standard grew with the yaw angle, keeping the maximum error of $C_{Mx,lee}$ below 15 per cent. The influence of the ground scenario did not seem important since the results using the flat ground with gap were similar to those obtained by introducing the double track ballast and rail ground.

By comparing the value of $C_{Mx,lee}$ measured in experimental tests with the value calculated in simulations, we were able to accurately estimate the difference of the wheel unloading ratio. The differences observed in the wheel unloading ratio once the aerodynamic coefficients were introduced in the multibody model were similar to the variations of the $C_{Mx,lee}$ coefficient.
Chapter 6

effect of fences on vehicle aerodynamics

The following chapter is fully dedicated to the study of wind fences for being one of the main goals of this thesis. The assessment includes the influence of fences both on the value of the vehicle’s aerodynamic coefficients and on the flow characteristics. Firstly, a two-dimensional model is used to carry out a parametric study of the fence design, in which both the height of the fence and the inclusion of an eave on the top are analyzed. A few fence designs were selected to assess the sheltering effect in terms of three-dimensional models. The results from both approaches are compared to verify whether simplified models are appropriate to study this topic.

6.1 A 2D study of the effects of fences on a vehicle standing on a bridge

When designing a new wind fence, the influence of parameters such as the height and length of the eave, if there are any, needs to be tested in order to decide which designs are the most appropriate to safeguard the train from wind in a particular scenario. Tests can be done in a wind tunnel as the literature proves it to be a valid method; however, the costs of running the experiments are high and using a wind tunnel also may not be the most effective approach since each study requires a particular mock-up be built. In addition, the Reynolds number that can be achieved in experimental tests is limited by the size and the power of wind tunnels, which makes it almost impossible to represent real wind conditions. In recent years, CFD has appeared as a suitable alternative to reducing wind tunnel tests because it is able to simulate flow accurately and allows real wind conditions to be reproduced.

This section analyses the relative effectiveness of solid wind fences that are built on a bridge to decrease the aerodynamic loads that might cause a vehicle to overturn in strong wind conditions ($Re = 12 \times 10^5$). In the scope of a two-dimensional study, we
Chapter 6: effect of fences on vehicle aerodynamics

144

discuss about the design parameters of fences, i.e. fence height and eave length, along with how such fences modify the flow structure because fences determines the difference in pressure on the surface of the vehicle and thus the aerodynamic coefficients. The problem must be studied using URANS equations because there is a vortex shedding phenomenon that forces the study to go beyond steady state simulations. The discussion will be from the point of view of whether the vehicle is more protected and whether loads over it are lower. The influence of the fence design on the frequency and peak values of the signals of the aerodynamic coefficients is also studied.

Three-dimensional calculations provide the best approach to real phenomena but they are not the most efficient way if one wants to complete a parametric study. These kinds of calculations require high computational resources and they are costly in most cases because of the time it takes to get a set of results. Nevertheless, a 3D approach is necessary to extract conclusions in relation to the influence of fence design on vehicle dynamics since it is the behavior of the three-dimensional flow which differentiates one vehicle from another (Baker et al., 2004) and Copley, (1987).

The 2D approach we used is valid for fulfilling our goal, which is to compare different wind fence designs and to determine which features (fence height and eave length) are more important from a safety perspective. Two-dimensional simulations are a useful tool for comparing the relative effectiveness of the wind fences and for measuring the efficiency of train sheltering. When there is a set of wind fences and one wants to select a few from among the set in order to carry out a detailed study in 3D, two-dimensional simulations are sufficient. However, this study does not discuss how fences modify the wake around a high-speed train since a 3D study is required for that purpose.

The structure of the section is as follows. First, the transient regime of the problem was analyzed by looking at the time signals of the coefficients calculated with simulations at Re $\approx 12 \times 10^6$. Then, the average values of the aerodynamic coefficients of the train coach were studied in order to find the most effective fence. At this point, conclusions regarding the fence construction parameters (fence height and eave length) were extracted. The flow around the vehicle and the bridge was observed in detail, establishing differences between the two wind velocities under consideration in this study. Finally, the vehicle coefficients from simulations with the two Reynolds numbers were compared, in order to determine whether $Re$ influences the values of the coefficients.
6.1.1 Effects on vehicle coefficients and flow

a) Study of the time signals. Characteristics and influence of fences

Inlet conditions are stationary, that is to say, free flow velocity and turbulence intensity are constant. A bridge inside a flow (with or without a train on it) makes a vortex shedding phenomenon to appear; thus, the problem does not have a stationary solution (FIGURE 80).

Flow needs some time until it fully develops and the periodic state is reached. Obviously, by adding fences of different designs, we modify indirectly the vortex characteristics since the bridge section changes. We could state that vortexes determine the aerodynamic characteristics of the bridge and the vehicle.

We found of interest the study of time signals, mainly two characteristics: time period and peak-to-peak amplitude. The latter refers to the difference between the highest value and the lowest value of the signals in the measured time. Average values of the time signals, which are in fact the proper aerodynamic coefficients, are discussed in the next section.

Peak-to-peak values depend on the aerodynamic coefficient and on the fence design. There was not any clear trend in how peak-to-peak values changed when we applied modifications to the fence design, but we observed a common law which applied in all the scenarios. The highest peak-to-peak value was always found in the drag coefficient, with an order of $10^{-2}$. The lift coefficient peak-to-peak value was between the drag force coefficient and the roll moment coefficient and its order was also $10^{-2}$. Roll moment coefficient had an order of $10^{-3}$, which made it being the coefficient with the lowest peak-to-peak value. In order to complete this information, we calculated what is the percentage that represents the peak-to-peak value over the average. In such way, the peak-to-peak value of $C_d$ represented around 25 per cent, $C_l$ only 5 per cent and 15 per cent was the variation of $C_{m0}$. Nevertheless, $C_{m0}$ usually takes near-zero values so small variations provide big variations in percentage terms.
Frequency of the vortex shedding phenomenon is observed through the signals' time period. The three aerodynamic coefficients had similar time periods despite applying changes to the fence design. In this sense, the study of how did the fence design increase—or decrease—the signals’ time period was only done in terms of the lift force coefficient. Our reference was the scenario in which the vehicle is unsheltered, where the time period \( T \) of \( C_l \) was 1.6 s (Strouhal = 1.1). In those scenarios where time periods were longer in comparison with the reference case, the variation gave positive values. In the same way, negative values means that time periods were shorter. FIGURE 81 show that \( h_f \) and \( l_e \) modified the time period of the signals but having a small impact and without any clear trend. The highest variation was observed in the case of a 2 250 mm-fence without eave that modified the time period around 15 per cent, an increase of 0.24 s over the reference case. The maximum variation was observed between this case and the 1 750 mm-fence with 750 mm-eave case, where the variation was 22.5 per cent (0.4 s).
b) Influence of fence design over the coefficients and the flow

We validated the model with experimental tests at $\text{Re} = 1 \times 10^5$ in Section 3.2.1, verifying that the CFD model represented the effect of fences on the vehicle aerodynamics. Nevertheless, carrying on tests or simulations at low Reynolds number is not be enough since the coefficients of the vehicle may depend on the $\text{Re}$ at which simulations are carried out. Results in this section were obtained from computations representing strong wind conditions, where free flow velocity equals to 30 m/s or $\text{Re} = 12 \times 10^6$.

FIGURE 82 shows the coefficients of the vehicle calculated under strong wind conditions. The value of every coefficient decreased as fence height increased, obtaining lower values when fences included an eave on top. The effect of fences on the coefficients was not uniform providing that each one had a different trend. A general overview of results indicates that fences had a bigger impact on $C_l$ (coefficient of lift force) than on $C_d$ (coefficient of side force herein called drag force for being a 2D model). In addition, $C_{m0}$ (coefficient of roll moment) decreased to value zero when the fence of 1 250 mm was included, and the value did not change despite changing the fence design. It means that fences provide of sheltering to the vehicle essentially by decreasing the lift force and the roll moment.

The drag force coefficient went down form 0.4 in the scenario without fence to -0.4 in the scenario where the fence was the highest ($h_f = 2 750$ mm). Following the standard convention, positive values shows that the vehicle is loaded with forces in the wind direction, and negative values reveals the appearance of suction forces. Simulations indicated that fences are able to reduce drag forces but at the same time, fence may...
create suction forces as larger as drag forces. $C_d$ dropped below zero with fences whose height was greater than 1,750 mm with eave or 2,250 mm without an eave on top. The drag force coefficient went down mainly as a result of decreasing the pressure at windward since pressure changed slightly at leeward (FIGURE 84). The bigger the fence was, the lower was the pressure at windward. It happens in this way because a vortex between the fence and the windward side of the vehicle appears in the configurations where $h_f = 2,250$ mm and $2,750$ mm (FIGURE 83). The vortex accelerated the flow contributing to decrease pressure at the windward. Nevertheless, the opposite phenomenon happens at leeward and that a low speed zone comes up at this side of the vehicle. All these phenomena working together are the origin of having the same pressure difference between windward and leeward as the case when any fence did not protect the vehicle.

![FIGURE 82: Two-dimensional aerodynamic coefficients of the vehicle calculated at Re = 12 x 10^6](image)

Fences had a greater influence on the difference of pressure between the top and the bottom of the coach than on the sides; therefore, $C_l$ went from 1.6 to 0.4 whereas the value of $C_d$ dropped a total of 0.77. By increasing the fence height, suction forces at the top decreased and so did pressure at the bottom, though at a lower rate (see FIGURE 84). We verified that these suction forces were the main cause of the pressure difference between the top and the bottom that produced high values of $C_l$. What makes the suction forces to be so high is the path that the flow follows to pass the vehicle. The main stream of flow reaches the windward side of the vehicle when the bridge is unequipped with fences. Wind breaking devices, such as fences, deviates the flow as fences become higher, reducing gradually the interaction between flow and the windward side of vehicle. Without fences, the main stream of flow reached the vehicle near the bottom, thus the flow needed to accelerate to pass over the carbody. In fact, the value of speed at the top of the vehicle reached 78 m/s when the vehicle
was unsheltered. By making higher fences, suction forces at the top decreased since the flow gets to the vehicle at a higher position, so the required acceleration was smaller. It means that fences helped the flow to pass over the vehicle and as long as fences were able to change the flow direction, $C_l$ dropped.

![Streamlines coloured with velocity in meters per second when Re $\approx$ 12 x 10$^6$. The scenario has a fence without eave and a height of 2 250 mm](image)

The stream that goes under the vehicle fixed the value of pressure at the bottom and it also had some impact on the value of $C_p$, although it was not as relevant as the value of the suction forces of the top. The cross-sectional area under the vehicle was small enough to speed-up the flow, modifying the flow at leeward (see FIGURE 84). Fences also affected the flow under the vehicle and it slowed down as fence height increased, hence, pressure at the bottom increased slightly. Furthermore, the vortex that appeared at the windward side was able to change the direction of the flow, going from leeward to windward in the cases of $h_t = 2 250$ mm and $h_t = 2 750$ mm as result of the suction forces created by the vortex. However, the direction change did not vary the pressure value at the bottom.
Chapter 6: effect of fences on vehicle aerodynamics

FIGURE 84: Contours of mean pressure [Pa] (up) and mean velocity [m/s] (down) when $Re \approx 12 \times 10^6$. Three scenarios are compared: no fence (left), $h_f = 1250$ mm (center), $h_f = 2250$ mm (right).

It was previously stated that adding an eave contributed to decrease the forces acting on the vehicle and thus the coefficients (see FIGURE 82). Nevertheless, the eave length parameter, $l_e$, did not seem important since the variation of the coefficients value between the eaves whose length is 500 mm and length 750 mm is almost zero. Eaves had a greater impact on $C_l$ than on $C_d$ and $C_m$, in fact, the lift force coefficient reached zero where the fence was $h_f = 2750$ mm and with eave.

We selected the fence with 2250 mm height to compare the outcome of adding an eave and modifying its length from 500 mm to 750 mm (FIGURE 85). The main contribution of eaves to the sheltering effect is the decrease of the suction forces at the top of the vehicle. As FIGURE 82 shows, the value of $C_l$ decreased by around 0.3 where the fence $h_f = 1750$ mm and above. What the eave does is to produce and additional deviation of the flow so the main stream of flow tends to gain distance with the coach as the value of $l_e$ is higher, causing two opposite effects. On the one hand, the flow needs to accelerate less to pass over the carbody since the eaves helps to produce and additional deviation of the flow; it makes $C_l$ to reduce as a result of getting lower suction forces. On the other hand, the flow has bigger values of speed at higher positions, which is an effect predicted by other authors. By increasing turbulence at positions near the catenary, it may result in the galloping of the cable (Avila-Sanchez et al., 2010). A 750-mm-eave did not increase the sheltering effect since the direction of the flow did not vary enough to change the pressure field, hence, the aerodynamic coefficients changed slightly.
Chapter 6: effect of fences on vehicle aerodynamics

Looking at FIGURE 85 and the difference in pressure between the windward and the leeward side of the vehicle, it is clear that pressure decreases slightly at the windward side and the pressure value at the leeward side remains constant. This resulted in a small variation in $C_d$ for both cases of fence with eaves and fence without eave, and the same thing happened between $l_e = 500$ mm and $l_e = 750$ mm.

FIGURE 86 shows the calculation of the moment coefficient around the leeward rail, $C_{mv}$. Results show that the moment decrease a little when the height of the fence is 1 250 mm regardless of whether the fence has eaves; therefore, fences can only be considered efficient with heights of 1 750 mm and above. Fences without eaves decrease the moment to -0.17, and $C_{mv}$ drops to -0.34 when they have eaves. The latter is explained by the large suction forces that turn $C_d$ into negative values. From the safety point of view, it means that the aerodynamic moment around the leeward rail in the case of the vehicle unsheltered is as big as the moment with fences of height 2 750 mm with eaves. This result is in our opinion influenced by the two-dimensional spaced in which the study was carried out. Suction forces are expected to appear in the cases fence height 2 250 mm and above but not as high as the ones predicted by this simulations. In order to confirm such effect, a three-dimensional study of this configuration would be needed.
Eaves contribution in the shielding of the vehicle increases as fences become higher. In the case of the lowest fence \( (h_f = 1250 \text{ mm}) \), eaves reduce \( C_{mv} \) slightly and so they do not increase safety. Eaves efficiency grows up to reduce the value of \( C_{mv} \) and additional 0.2 when the height is 2750 mm. In all the cases, by increasing the eave length from 500 mm to 750 mm safety conditions do not get better since longer eaves do not produce an additional reduction of the moment around the leeward rail.

### 6.1.2 Reynolds dependency of the aerodynamic coefficients

The 2D section of the vehicle plus the bridge under consideration in this study could be also classified as a more-or-less bluff body inside an airstream. In such cases, the Reynolds number is of vital importance since it could have a strong influence on certain flow parameters. This section studies what these magnitudes are, and in the case they are affected by \( Re \), analyses whether the change is considerable. This is possible since we simulated the same scenarios at two \( Re \) number, ones \( (Re = 12 \times 10^5) \) allowed us to study the relative effectiveness of fences whereas the others \( (Re = 1 \times 10^5) \) validated the model.

#### a) Effects on the time signals

Two aspects of the coefficients’ time signals were discussed in Section 6.1.1, which were the time period and the peak-to-peak values. In this section, the influence that \( Re \) has on the frequency of the vortex shedding phenomenon was studied in terms of the time period \( (T) \) and the Strouhal number \( (St) \). The effect of \( Re \) on the peak-to-peak values was checked as well.
Chapter 6: effect of fences on vehicle aerodynamics

Re has a strong influence on the coefficients time period. We observed that for the calculations carried out at Re = 12 x 10^5, the time period of the coefficients was around 1.5 s. The same scenarios at lower Reynolds (Re = 1 x 10^5) had a time period in the range 225 s < T < 250 s where 238 s was the case of the bridge without fence. FIGURE 87 shows the variations with respect to the case of an unsheltered vehicle calculated in the simulations at Re = 1 x 10^5, by applying the methodology that was used for FIGURE 81. The maximum variation appeared in the case of fence whose parameters are h_f = 2 750 mm and l_e = 500 mm which was 7.5 per cent and corresponded to 20 s, whereas the maximum difference at Re = 12 x 10^5 was approximately 15 per cent. Furthermore, the difference between the maximum and the minimum value of the period was 25 per cent at Re = 12 x 10^5 (see FIGURE 81), decreasing to 15 per cent at Re = 1 x 10^5.

FIGURE 87. Variation of the time period of the coefficients time signals in terms of percentage by taking as reference the 'No Fence' scenario. Calculations carried out at Re = 1 x 10^5

FIGURE 88 shows the result of calculating St in all the cases considered in the present study, arranged by Re. Analyzing the results at each Re separately, it was concluded that no clear trend relates the fence design parameters with the Strouhal number. Furthermore, St behavior changed when Re was increased, which made it impossible to analyze the effect of Re on the St. The graph shows that St changed slightly between the two Re; in fact, while the Strouhal number for simulations at low Reynolds numbers is always between 0.95 and 1.10, for Re = 12 x 10^5 it varies between 0.95 and 1.2. The small changes of St whenever Re, h_f or l_e was changed made it impossible to determine what changes happened in the flow that caused the variation of St. Lastly, by modifying a parameter of the fence design (h_f or l_e), changes were smooth at Re = 12 x 10^5, whereas at Re = 12 x 10^5 they were not.
Regarding peak-to-peak values, it is possible to add that for the three aerodynamic coefficients, their order at $Re = 1 \times 10^5$ was the same as the ones presented in Section 6.1.1. At low $Re$, peak-to-peak only represents 5 per cent of the average value in $C_d$, $Cl$ and $C_{m0}$, while at $Re = 12 \times 10^6$ the value was 25 per cent, 5 per cent and 15 per cent, respectively.

b) **Effects on the aerodynamic coefficients**

As FIGURE 89 shows the variation in $C_{mv}$ is small, oscillating between 0.1 and 0.2. It reaches the maximum value when $h_f = 2.750$ mm and there is no eave, and the minimum appears in the case where $h_f = 1.250$ mm and there is no eave. In general terms, $\Delta C_{mv}$ tends to have the value of $\Delta C_a$ getting closer to it as the fence height increases since $\Delta C_l$ also approaches zero. In calculating $C_{mv}$, the drag force coefficient prevailed over $Cl$ and that is reflected here, too. $C_{m0}$ also had an influence on $C_{mv}$ though it was shown in Figure 13 that $Re$ had no effect on it. The fact that the value of
$\Delta C_{mv}$ is given mainly by the variation of $C_d$ is easy to check in the ‘No Fence’ case and in the configuration where $h_f = 2750$ mm. When the vehicle is unprotected, the variation in $C_l$ reaches the maximum; however, $\Delta C_{mv}$ is clearly close to $\Delta C_d$ which is a sign of the effect of the drag force coefficient on $C_{mv}$. In addition, $\Delta C_{mv}$ almost has the same value as $\Delta C_d$ when $h_f = 2750$ mm since in such case the variation in the lift and roll moment coefficients is almost zero, and thus only the drag force coefficient sets the value of $\Delta C_{mv}$.

Eaves have a small influence on the conclusions presented in this section. Adding an eave meant a small increase in the dependence of $C_d$ on $Re$ in all the fence designs. The influence of $Re$ relies on the fence height in the case of $C_l$. For the small fences, $h_f = 1250$ mm and $1750$ mm, adding an eave made the coefficient more sensitive to $Re$ but the opposite happens in the other two fence heights. Lastly, $\Delta C_{mv}$ increased from 0.1 to approximately 0.2 in the cases where $h_f = 2250$ mm and $2750$ mm. What does not seem to be relevant in this matter is eave length since the variation in $\Delta C$ in every fence design is near zero in spite of having increased le from 500 mm to 750 mm.

![Figure 89](image.png)

**Figure 89.** Reynolds dependency of the aerodynamic coefficients $C_d$, $C_l$, $C_{mv}$. Difference between tests at $Re = 1 \times 10^5$ and $Re = 12 \times 10^6$.
6.2 A brief assessment of fences in the 3D space

The brief assessment of the sheltering effect of fences in the 3D space was done with two fence designs chosen from the 2D analysis presented in Section 6.1. Simulations were done in terms of the CFD software based on the Lattice-Boltzmann Method that was validated in Section 5.3.

The fence dimensions were very similar to the ones considered in the 2D analysis though they were not identical, as the selected designs were $h_f = 1,700$ mm and $h_f = 2,700$ mm. The fence height varied below 3 per cent for the case of $h_f = 1,700$ mm as the fence introduced in the 2D model was $h_f = 1,750$ mm. Similarly, the fence with height 2,700 mm has a difference below 2 per cent as the fence in the 2D model was $h_f = 2,750$ mm. We needed to apply changes to the fence height forced by the way in which the lattice discretizes the domain. As the lattices are cubes, the fence height needed to be a multiple number of the lattice size so as to properly fix the detachment of the flow at the top-end of the fence. In such a way, the discretization of the fence with height 1,700 mm was done with 17 lattices of size 0.1 m, and the fence with height 2,700 mm was discretized with 27 lattices also of size 0.1 m (see FIGURE 90).

![FIGURE 90: Mean velocity field and discretization of the domain in the surroundings of the fence $hf = 1,750$ mm (right) and $hf = 2,750$ mm (left)](image)

FIGURE 91 shows that the value of the aerodynamic coefficients taken into account in the calculation of $C_{M_{x,free}}$ (side and lift forces, and roll moment) decreased by the sheltering effect created by any of the two fence designs. The sheltering effect increased progressively as the fence height also increased (see FIGURE 93), unlike the results provided by the 2D model since the latter predicted that suction forces appear at the windward side of the vehicle (see FIGURE 82). Hence, such effect came up only from the inherent limitations of the 2D model, as we expected before doing any simulation in the 3D space (see Section 6.1.1). Moreover, 3D simulations show that both fences had more effect on the side force and roll moment than on the lift force.
Therefore, the analysis of the sheltering effect of fences in the 3D space does not match with the results provided by the two-dimensional model in which fences had less influence on the side force than on the other coefficients (see FIGURE 82).

The side coefficient is produced by the pressure difference between the sides of the vehicle (FIGURE 92). On the scenario where the vehicle is unsheltered, high values of pressure were computed at the windward side, which in addition to the low values at leeward, made the simulations to overestimate the side coefficient value. The fence with 1 750 mm height had a great influence on the side coefficient since the value dropped from 7.48 to 2.09 which corresponds to a 72 per cent decrease. The side force was however increased by installing the \( h_f = 2\,750 \) mm as the coefficient only decreased to 3.54. FIGURE 92 shows that the pressure at both sides of the vehicle decreases by the effect of fences, however the pressure difference between windward and leeward can increase despite having installed higher fences. The pressure difference between the vehicle’s sides in the case of \( h_f = 2\,750 \) mm is higher, thus the side coefficient is also higher in comparison with the case of \( h_f = 1\,750 \) mm. This effect created by the highest fence is mainly due to an additional decrease of the pressure at the leeward side.

The value of the lift coefficient was barely influenced by the 1 750-mm-height fence, as the variation was approximately a 2 per cent mainly due to the unchanging value...
of pressure on the vehicle's top (see FIGURE 92). On the contrary, the highest fence substantially increased the value of the pressure on the top, dropping the coefficient value from -6.19 to -1.72. Even though the pressure field on the bottom on the vehicle is not shown, we can point out that the pressure increased when the 1 750-mm-height fence was installed and remained constant in comparison with the highest fence scenario ($h_f = 2 750 \text{ mm}$).

![Diagram](image)

FIGURE 92: Contours of mean pressure on the vehicle surface [Pa]. Three scenarios are compared: No Fence (up), $h_f = 1 750 \text{ mm}$ (center) and $h_f = 2 750 \text{ mm}$ (down)

The fence $h_f = 1 750 \text{ mm}$ decreases the roll coefficient value from 3.73 to 1.54 (a 58 per cent) mainly due to the reduction on the side force value. Nevertheless, the
value of the roll coefficient increased up to 2.01 with the highest fence, which corresponds to a 8 per cent increase with respect to the lower fence. Moreover, the pitch coefficient showed that both fences contributed to modify the lift forces distribution on the surface of the vehicle since the coefficient changed from negative to positive. The coefficient changed from $-1.61$ to $1.15$ ($h_f = 1750$ mm) and to $1.43$ ($h_f = 2750$ mm). FIGURE 92 shows that the change on the sign of the coefficient was due to a progressive increase of the pressure on sections near the nose. Similarly, the yaw coefficient which varies by changes in the pressure distribution between the sides and the front/rear sections of the vehicle, changed from $-1.4$ to $0.5$ when $h_f = 2750$ mm. It is expected that the new pressure distribution observed on the leeward side on sections near the nose caused this changed to positive values.

To a lesser extent, the drag force, though not important in the crosswinds topic, was reduced by two thirds of the value computed in the unsheltered vehicle scenario when $h_f = 1750$ mm. The highest fence produced an additional 17-per-cent-decrease of the drag force in comparison with the other fence, caused by the increase of the pressure at the leeward side, similarly as the phenomena behind the increase of the yaw moment coefficient.

Measuring the sheltering effect of fences in terms of the overturning moment around the leeward rail ($C_{M_{x,lee}}$) shows that the overturning moment is mainly a consequence of reducing the roll moment since the contribution of the lift coefficient is small. The higher the fence was the larger decreases of $C_{M_{x,lee}}$ were computed (see FIGURE 93). The effect of the fence with height 1750 mm was to drop the value of the moment coefficient from 5.28 to 3.10 (a 41 per cent decrease), which is similar to the reduction of the roll (58 per cent). This effect can also be observed with the highest fence (2750 mm) since the value of $C_{M_{x,lee}}$ decreased by 53 per cent, close to the observed reduction of the roll moment coefficient (46 per cent).

![FIGURE 93: Effect of fences on the $C_{M_{x,lee}}$ coefficient](image-url)

The computations with the 2D model are not completely comparable to results got with the 3D model due to various reasons. Firstly, the scenario on which the vehicle
stands is different as the 2D model represented the cross-section of a car standing on a bridge (see FIGURE 23). However, the flow surrounding a middle section of the vehicle and in between the fences is similar, as FIGURE 95 and FIGURE 84 show. The vehicle under study was also different though a representative middle section of both vehicles is similar. A comparison assessing differences coefficient by coefficient is not feasible, as the forces measured on the 2D model are not directly equivalent to the forces measured in the 3D space. However, the force state of both models can be carefully compared in terms of the overturning around the leeward rail since this coefficient considers the overall force contribution to the overturning of the vehicle. FIGURE 94 shows a comparison of the sheltering effect of the fence design by calculating the decrease of $C_{\text{Mx,lee}}$ in each fence scenario, taking as reference the case where the vehicle stands unsheltered. Results verify that the 2D model is able to give an estimation of the sheltering effect of fences, which was the main statement on the discussion of the results in Section 6.1. Assuming the uncertainties inherent to the difference on the computational models, we observed that the 2D-3D difference is estimated in a 9 per cent for the case $h_f = 1750$ mm and a 13 per cent for the fence 2750-mm case. FIGURE 94 shows that each computed case behaved differently. By comparing both models, the 3D model predicted a greater sheltering effect for the case $h_f = 1750$ mm but lower when $h_f = 2750$ mm. However, the 2D model overestimated the suction forces at the windward side for the highest fence case and the value of the roll moment dropped from 0.45 to -0.15 (see FIGURE 86) so that the vehicle would overturn in the opposite direction of rotation.

FIGURE 94: Comparison of the moment around the leeward rail computed with the simplified model (gray color) and the 3D model (red color)

FIGURE 95 shows a comparison of the mean pressure field on the three computed scenarios. Fences modify the mainstream from the inlet by leading the flow over the top of the vehicle. As long as fences are high enough, the mainstream of flow is able to completely overpass the vehicle without hitting the windward side of the vehicle. This effect can be observed in the case $h_f = 1 750$ mm where a high-pressure zone appears at windward near the top of the vehicle since the flow is not able to overpass the vehicle. A vortex also appears at windward between the fence and the vehicle,
Chapter 6: effect of fences on vehicle aerodynamics

partially setting the pressure field at this side. Moreover, another vortex determines the pressure field at leeward, whose lateral distance from the vehicle sets the pressure value on the vehicle’s surface. Both vortices were also computed by the 2D model (see FIGURE 84) but the inherent limitations of a two-dimensional model could not predict the changes on the pressure field between sections. However, the general aspect of the pressure field is similar at the middle section of the vehicle \((X = 0 \text{ m})\) even though pressure values are different. This is the reason of why we observed suctions forces at leeward on the two-dimensional model which the 3D model did not reproduce. The value of pressure on the top of the vehicle is lower at sections near and further from the nose than on the middle section with the two fences at windward (see also FIGURE 92), created by the higher values of the vortex velocity. This vortex is fully structured at the middle where both the top and the down surfaces of the carbody are constant, enabling the vortex to grow in size and help the airflow to reach higher positions. The presence of the bogies alters the vortex at the other two positions which breaks apart, making the flow to overpass the vehicle closer to the top surface.

\[
\begin{align*}
X &= -8 \text{ m} \\
X &= 0 \text{ m} \\
X &= 10 \text{ m}
\end{align*}
\]

FIGURE 95: Contours of mean pressure [Pa] at the positions showed in FIGURE 62. Three scenarios are compared: No Fence (up), \(h_f = 1.750 \text{ mm}\) (center) and \(h_f = 2.750 \text{ mm}\) (down)

The mean velocity field is shown in FIGURE 96 and the vectors in FIGURE 97. The flow always reaches the maximum speed on the coach surface, decreasing on the wake as it evolves over the scenario. The flow goes under the carbody from windward to leeward with lower velocity as the fence height increases, showing the largest values in the scenario where the vehicle is unsheltered and the lowest when \(h_f = 2.750 \text{ mm}\). Similarly, the flow speed on the top of the carbody is lower as the fence height
increases, an effect which is clearly visible for the case $h_r = 2.750$ mm. Computations show that the rotation velocity of the leeward vortex changes between sections. The vortex decelerates at sections further from the nose in the two scenarios with fences. Furthermore, we realized that the vortex velocity is lower for the case $h_r = 2.750$ mm which is the cause of having higher pressure values on the area of influence of the vortex.

![Diagram showing contours of mean velocity](image)

**FIGURE 96**: Contours of mean velocity [m/s] at the positions showed in FIGURE 62. Three scenarios are compared: No Fence (up), $h_r = 1.750$ mm (center) and $h_r = 2.750$ mm (down).
6.2.1 Conclusions

We studied firstly the relative effectiveness of wind-breaking devices in the 2D space with simplified models. The problem did not have a stationary solution, thus it was studied in the time domain. A vortex shedding phenomenon appeared despite having stationary inlet conditions. It was shown that the CFD model is able to reproduce the overall trends of the aerodynamic coefficients with fence height and the presence of eaves in agreement with the experiments carried out in a wind tunnel. Moreover, it showed that a CFD model that represents a vehicle on a bridge is a useful tool for carrying out a parametric study as an alternative to do experimental tests.

Real wind conditions were simulated, which showed that fence design modifies the flow and thus the efficiency of the protection. When fence height increases, wind loads are reduced but fences must be high enough to provide the protection needed to safeguard the train’s running. Results showed that fences are able to modify the flow in such a way that suction forces that are as large as drag forces can appear when the fence height is large. This effect is enhanced with the inclusion of an eave. The CFD model showed that fences had more influence on lift force than on drag force, and roll moment approached zero when any fence was installed. The presence of an eave slightly increased the shielding effect of fences, and it had a higher impact on the lift force coefficient than on the others. Furthermore, simulations and
experiments showed that there is no difference between the scenarios where eave length was 500 mm or 750 mm.

With the cross-sectional area of the vehicle plus the bridge being a more-or-less bluff body, the aerodynamic properties changed with Re. The vehicle coefficients followed the same trend with the two Reynolds number that were simulated; nevertheless, the numerical values changed sufficiently to consider that the simulations carried on at low Reynolds number are not enough to decide which fence design is the best with the information from the wind tunnels tests.

On the other hand, the sheltering effect of two fences taken from the 2D study was assessed in the 3D space for a train located on the ground. The results obtained show that both fences produce a substantial decrease of the forces and moments that are significant for the overturning. In this regard, fences had more influence on the side force and the roll moment than on the lift force. This conclusion contrasts with the results provided by the 2D model, however the resultant force state achieved with both models is almost equivalent. The coefficients of the three-dimensional moments of pitch and yaw change the sign as a result of including fences, even though the absolute value of both moments remains similar. Finally, the drag coefficient largely decreases after installing any of the studied fences.

The efficiency of the protection increases as the fence is made higher, even though the side force and the roll moment rise slightly. For the case of the highest fence, the pressure difference between the windward and the leeward sides of the vehicle increases, hence the sheltering effect in terms of the side forces and the roll moment decreases. However, the reduction of the lift force provided by this fence definitively contributes to decrease the overall overturning moment of the vehicle.
Chapter 7

conclusions and future work

7.1 Conclusions

The present work has studied the effect of crosswinds on the circulation of railway vehicles. It is a coupled problem that involves the study of both the vehicle dynamics and aerodynamics.

In regard to the vehicle dynamics, this thesis concluded:

- The calculation of wheel unloading ratio to assess the safety of vehicles under crosswinds using 2D models provides satisfactory results and low errors.
- The difference between the CWCs calculated in terms of full multibody models and using 2D simplified models is small and acceptable in most cases.
- In the crosswind context and for the purpose of building the CWCs of the vehicle, the wheel/rail contact can be simplified to consider only one point at the contact. In addition, there is no difference between using the elastic or the constraint options.
- Crosswinds can also become a comfort issue when vehicles run on curved tracks. If the wind velocity is high enough, the value of the non-compensated accelerations can be above the limits established by the standards.
- If vehicles run on curved tracks in a crosswind scenario, a hunting movement can appear even in cases where the wheel unloading ratio is well below the 0.9 limit value proposed by the standards.
The study of the vehicle aerodynamics was able to provide the following conclusions:

- Calculations in terms of CFD simulations are a useful tool to perform parametric analysis of the fence design, reproducing the overall trends of the aerodynamic coefficients measured in wind tunnel tests.
- By installing wind fences next to the track, the wind force that vehicles withstand is reduced in a significant level. However, a bad fence design can be also the cause of obtaining unexpected high values of the moment around the leeward rail.
- Fences should be high enough to make the flow overpass the vehicle providing that too-high fences create suction forces that decrease the sheltering effect.
- The computed forces and moments on the vehicle highly depend on the Reynolds number at which the 2D tests are carried. Although the independence from the Reynolds number is reached above 250 000 in the 3D space, our results show that the variation between tests at $2 \cdot 10^6$ and at $1 \cdot 10^5$ in 2D studies can lead to important differences.
- A CFD software based on the Lattice-Boltzmann method has been validated in terms of the CEN standard showing an acceptable agreement between experimental tests and numerical results. Moreover, its meshless approach makes the study of the sheltering effect of wind fences easier.

### 7.2 Future work

The introduction explained that the topic herein covered is a coupled problem. However, the calculations presented in this thesis solved dynamics and aerodynamics in two stages. Wanting to move forward, the next step would consist on linking the software used to study dynamics with a CFD code in order to solve together both problems.

The parametric analysis of fences was useful to select a few among a full range of possible designs. We pointed out that 3D simulations are needed to confirm the results provided by the simplified models, and only a few barriers were tested. In addition, the study of porous fences would be also interesting since this kind of barriers are also currently being installed.

The initial goal of using a meshless code was to perform simulations with a moving model in order to measure the wind forces at the entrance and at the exit of a fenced track. However, this could not be accomplished since the validation of the software required more time than initially expected. This work finally managed to define the configuration to represent reasonable well the flow around vehicles subjected to
crosswinds, which would be necessary to apply in simulations with moving geometries. We believe that this assessment would answer some open questions in the field of crosswinds over high-speed trains.
Appendix A

In this appendix, the publications which have been developed within this thesis until this moment are presented, showing the first page of the articles.


A comparison of crosswind calculations using a full vehicle and a simplified 2D model

I Sesma 1,∗, J Vinolas 2, A San Emeterio 3, and J G Gimenez 2
1CEIT and Tecnun, University of Navarra, San Sebastian, Spain
2Construcciones y Auxiliar de Ferrocarriles SA, Beasain, and Tecnun, University of Navarra, San Sebastian, Spain

The manuscript was received on 31 March 2011 and was accepted after revision for publication on 24 August 2011.
DOI: 10.1177/0954409711424094

Abstract: High-speed vehicles are exposed to crosswinds that can result in the overturning of the vehicle. As part of the vehicle homologation process, the characteristic wind curve (CWC) must be calculated to determine the maximum vehicle velocity under different wind conditions. European Standard EN 14067-6 presents different approaches for studying this matter, from a two-dimensional (2D) model to a full multi-body model. The key to the problem rests in the uncertainties inherent in a multi-body model: characterizing its suspension elements and obtaining its aerodynamic coefficients. This paper compares a 2D model with a multi-body model of the same vehicle, and presents the advantages and disadvantages of each one. Results show that a 2D model is valid for graphing the CWC and sufficient for studying the basics of the overturning process. Furthermore, a wheel unloading time signal is also studied. It is concluded that the simplification process can modify the transitory response but that the maximum wheel unloading remains constant. Finally, a study of the wheel/rail contact is performed to check if the contact model has an influence on the calculation of the CWCs.

Keywords: crosswind stability, train overturning, overturning risk, simplified models, multi-body simulations

1 INTRODUCTION

This paper is driven by the requirement to maintain high levels of safety and comfort on the increasing number of high-speed lines that have started to operate or are currently under construction worldwide. In the last decade, and particularly in the last five years, considerable effort has been expended on the study of the effects of crosswinds on railway vehicles, however, despite this intense effort there still remain open questions in this area.

Weight reduction methods to reduce the dead-weight per passenger, for example, by developing lighter rolling stock [1], are of particular interest. Furthermore, the design of new lines with higher embankments and a large number of bridges or viaducts worsens the situation. Consequently, the risk of a train being overturned by crosswinds is increased due to the combination of increased vehicle speed and reduction in weight, this decreases the stabilizing momentum.

The aerodynamics community is also concerned about crosswinds; steady and unsteady aerodynamic forces and their interaction with vehicles have been studied in [2–4] and [5], respectively. The influence of crosswinds on the dynamic behaviour of a vehicle and the risk of it being overturned has been studied by several authors [6, 7]. In [8] a full multi-body model was used to study the dynamics of a vehicle while negotiating curves. In [9] a full multi-body model and experimental analysis were compared, and a two-dimensional (2D) model was used to


Bibliography


Launder, B.E., Sharma, B.I., 1974. Application of the energy-dissipation model of
turbulence to the calculation of flow near a spinning disc. Letters in Heat and
Mass Transfer 131–137.


in a Closed Wind Tunnel. ARC R&M.


McNamara, G., Zanetti, G., 1988. Use of the Boltzmann equation to simulate lattice-

Nicoud, F., Ducros, F., 1999. Subgrid-Scale Stress Modelling Based on the Square of

Train, in: Proceedings of the 4th WSEAS International Conference on Fluid


Premnath, K., Banerjee, S., 2011. On the three-dimensional central moment lattice

Quian, Y.H., D’Humieres, D., Lallemand, P., 1992. Lattice BGK models for Navier-
Stokes equation. Europhysics Letters 17, 479–484.

Raghunathan, R., Kim, H.-D., Setoguchi, T., 2002. Aerodynamics of high-speed railway
train. Progress in Aerospace Sciences 38, 469–514.

Rhie, C.M., Chow, W.L., 1983. Numerical study of the turbulent flow past an airfoil
with trailing edge separation. AIAA Journal 1525–1532.


Aerodyn. 92, 535–545.


SIMPACK AG, 2008. SIMPACK Reference Guide - SIMPACK.


