<table>
<thead>
<tr>
<th><strong>Title</strong></th>
<th>Capability analysis of computational fluid dynamics models in wind shield study on Queensferry Crossing, Scotland</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Authors(s)</strong></td>
<td>Zhu, Licheng, McCrum, Daniel, Keenahan, Jennifer</td>
</tr>
<tr>
<td><strong>Publication date</strong></td>
<td>2022-08-01</td>
</tr>
<tr>
<td><strong>Publication information</strong></td>
<td>Zhu, Licheng, Daniel McCrum, and Jennifer Keenahan. “Capability Analysis of Computational Fluid Dynamics Models in Wind Shield Study on Queensferry Crossing, Scotland” (August 1, 2022).</td>
</tr>
<tr>
<td><strong>Publisher</strong></td>
<td>ICE Publishing</td>
</tr>
<tr>
<td><strong>Item record/more information</strong></td>
<td><a href="http://hdl.handle.net/10197/13156">http://hdl.handle.net/10197/13156</a></td>
</tr>
<tr>
<td><strong>Publisher’s version (DOI)</strong></td>
<td>10.1680/jbren.21.00095</td>
</tr>
</tbody>
</table>
Accepted manuscript doi:
10.1680/jbren.21.00095

Accepted manuscript
As a service to our authors and readers, we are putting peer-reviewed accepted manuscripts (AM) online, in the Ahead of Print section of each journal web page, shortly after acceptance.

Disclaimer
The AM is yet to be copyedited and formatted in journal house style but can still be read and referenced by quoting its unique reference number, the digital object identifier (DOI). Once the AM has been typeset, an ‘uncorrected proof’ PDF will replace the ‘accepted manuscript’ PDF. These formatted articles may still be corrected by the authors. During the Production process, errors may be discovered which could affect the content, and all legal disclaimers that apply to the journal relate to these versions also.

Version of record
The final edited article will be published in PDF and HTML and will contain all author corrections and is considered the version of record. Authors wishing to reference an article published Ahead of Print should quote its DOI. When an issue becomes available, queuing Ahead of Print articles will move to that issue’s Table of Contents. When the article is published in a journal issue, the full reference should be cited in addition to the DOI.
Submitted: 10 December 2021

Published online in ‘accepted manuscript’ format: 02 June 2022

Manuscript title: Comparing computational modelling of bridge wind shields to wind tunnel tests

Authors: Licheng Zhu, Daniel McCrum and Jennifer Keenahan

Affiliation: School of Civil Engineering, University College Dublin, Belfield, Dublin 4, Ireland

Corresponding author: Licheng Zhu, School of Civil Engineering, University College Dublin, Belfield, Dublin 4, Ireland.

E-mail: licheng.zhu@ucdconnect.ie
Abstract

Bridge aerodynamic studies are essential in ensuring the safety and acceptable performance of long-span bridges vulnerable to the effects of crosswinds. Aerodynamic studies were traditionally carried out in wind tunnel facilities, but there are now greater opportunities for using computational fluid dynamics modelling. Few studies of three-dimensional aerodynamic simulations of lightweight vehicles on bridges exist but there has been limited validation and verification work done to date. In the study reported in this paper, three-dimensional computational fluid dynamics models were developed for the Queensferry Crossing cable-stayed bridge in Scotland, containing wind shields and sample vehicles. The models considered the wind effects from a range of yaw wind angles and subsequently determined the aerodynamic coefficients of vehicles. The models were verified by means of a mesh sensitivity study, a domain sensitivity study and comparisons with wind-tunnel test results. The models were then validated by using the same modelling process with a different type of wind shield, and again comparing results with wind-tunnel test data for the same configuration. Results demonstrated that the modelling can determine the aerodynamic coefficients to a similar level of accuracy to that of wind tunnel tests.

Keywords: aerodynamic; bridges; computational fluid dynamics; wind tunnel test; wind loading; wind shield
1. Introduction

By their nature, bridges tend to be constructed in areas that are exposed to strong wind conditions, such as canyons, lakes and rivers (Bastos et al., 2018; Lin et al., 2018). As a result, there can be concerns over vehicle safety, and often speed limits on vehicles are implemented (Yan et al., 2021). Installing wind shields on bridges can effectively protect vehicles as they create low-wind zones, reducing the risk of strong winds (Kozmar, H., Procino, L., Borsani, A. and Bartoli, 2012; Ma, C., Duan, Q., Li, Q., Liao, H. and Tao, 2019). Zhang et al. (2015), for example, investigated wind-vehicle-bridge interaction by examining the shielding effect of triangular wind shields in protecting bridge towers from wind on a railway bridge. Their dynamic analysis results show that the peak value of the train car body acceleration was significantly reduced when the wind barrier size was increased.

Typically, aerodynamic studies of wind effects on bridges are carried out in wind tunnel facilities. Wind tunnel tests have traditionally been a vital element of the design and operation of long-span bridges as they provide relatively reliable and referenceable information regarding the wind effects on the bridge behaviour model (Cochran and Derickson, 2011).

Computational fluid dynamics (CFD) modelling has been successfully applied in many areas of fluid mechanics, including the aerospace industry, and in Formula 1 (Kieffer et al., 2006), with its application to the built and natural environment and in the discipline of Civil Engineering is ever increasing due to its practical applications and access to increased computational power (Bernardo et al., 2019; McGuill, C. and Keenahan, 2020; Zhang et al., 2021). More recently CFD has been used in the assessment of wind effects on bridges (Morgenthal, 2000; Li et al., 2010; Kavrakov, 2018; Bi, 2014). Wang et al. (2014) completed a study using CFD to explore the shielding of vehicles from crosswinds as they pass by a bridge tower. A vehicle immersed in the wake of a bridge tower was simulated and compared with wind tunnel test results. The results show that the computed aerodynamic coefficients of the vehicle were generally larger than wind tunnel results. The results also showed that the sudden changing of the aerodynamic forces caused by the tower may be shorter than the average driver reaction time and thus the influence of crosswinds on lightweight vehicles such as buses and trucks can induce traffic accidents, even when exposed to relatively low wind velocities.

Alonso-Estébanez et al. (2017) studied the performance of an articulating wind shield by investigating the effect of the angle between the wind shield and the horizontal plane on vehicle aerodynamics. Results indicated that the articulating wind shield was better at protecting vehicles from crosswinds than crash barriers alone.

Salati et al. (2019) used 3D CFD simulations to optimize the aerodynamic response of heavy good vehicles in crosswinds by changing the shape of the trainer, vortex Generators installed on the leading edge of the trailer and passive devices installed along the sides/top of the trailer. The 2D CFD models were validated with wind tunnel tests and results indicated that all methods supported the reduction of vehicle overturning moments when exposed to crosswinds.
Zhang et al. (2020) numerically studied the aerodynamic coefficients under various sheltering effects of wind barriers with different heights and porosities for lorry, bus and van models. Results indicated that the non-perforated wind-shield walls can significantly reduce the wind velocity around the vehicle, but will also simulate eddies, which might cause vortex-related problems. As the wall gets higher, this effect becomes more significant. The perforated wall can also effectively reduce the wind velocity and result in relatively smaller eddy-related issues than non-perforated walls with the same height. (Yan et al., 2021) developed a 3D CFD model of a train-bridge-wind shield to measure the transient aerodynamic load of the train. Significant fluctuations in the aerodynamic coefficients were found when the train entered and exited the wind shield due to the dramatic change in flow pattern. The derailment coefficient decreased with the increasing height of wind barriers, up to a limit of about 4m. The findings from Zhang et al. (2020) and Yan et al., (2021) confirm the findings of Procino et al. (2008) who suggested that the wind velocity would decrease as wind shields porosity decrease and barrier wall height increased.

While the work discussed above represents significant advancement in the numerical analysis of dynamic wind-bridge interaction for wind shields, it is largely theoretical. The analysis has been done for a generic fictitious wind shield and bridge configuration and there has been limited validation and verification work done to date. In other fields of bluff body aerodynamics, the limitations of CFD have been widely reported. For example, the analysis of flow fields around bluff bodies is still in the relatively early stages of development. The physics of the flow field around a bluff body is complex and difficult to simulate, as it includes various fluid dynamics phenomenon that are considered difficult to resolve (Murakami, 1997; Morgenthal, G. and McRobie, 2002; Blocken, B., 2014). Other computational issues exist, such as sharp edges of bluff bodies needing fine meshes to ensure accurate flows (Murakami, 1997; Blocken, B., 2014). Haque et al. (2019) found that the aerodynamic response of single box streamlined bridge decks showed very high sensitivity to the shaping parameters such as the bottom plate slope, width ratio and side ratio. Thorough CFD verification and validation is necessary to ensure meaningful results can be captured from CFD simulations.

In this study, for the first time, 3D CFD models are developed in OpenFOAM (OpenFOAM Version 7, 2019) using the k-ω-SST turbulence model for the Queensferry Crossing bridge in Edinburgh, United Kingdom, containing wind shields and a sample bus and truck. OpenFOAM uses the finite volume method (FVM) to convert partial different equations, which utilise Navier-Stokes equations, the mass and energy conservation equations and the turbulence equations, into a system of algebraic equations. Then, these equations can be solved over discrete control volumes (Hu, 2012). The CFD models accurately assess the wind effects from a range of yaw wind angles of attack and subsequently validate the aerodynamic coefficients. The models are verified by means of a mesh sensitivity study and a domain sensitivity study, and comparisons with wind tunnel test results. The models are then validated by using the same modelling process with a different type of wind shield, and again comparing results with wind tunnel test data for the same configuration.
2. Description of the Queensferry Crossing bridge

The Queensferry Crossing bridge shown in Figure 1 (formerly the Forth Replacement Crossing) connects the traffic across the Firth of Forth between Edinburgh, at South Queensferry, and Fife, at North Queensferry, in the United Kingdom. The bridge was opened in August 2017. The bridge’s 2.7 km length makes it used to be the longest three-tower cable-stayed bridge in the world and also the largest to feature cables which cross mid-span. The bridge deck is 41.6 m wide and carries two lanes of traffic in each direction alongside hard shoulders. The distance between the towers is 650 m. The bridge carries about 24 million vehicle journeys a year (Queensferry Crossing, 2014).

3. Description of the wind tunnel tests

Extensive wind tunnel tests on scaled models of the Queensferry Crossing were carried out in the Politecnico di Milano Boundary Layer Wind Tunnel in Italy. The wind tunnel is 36 m in length, 14 m wide and 4 m high. The main goals of the wind tunnel tests were to determine suitable cross-sectional shapes for the deck and the towers, determine the aerodynamic properties, identify effective wind shields, and confirm the aeroelastic stability of the bridge. In this paper, the authors focused on the Wind Shield Studies of the wind tunnel test report. Figure 2 illustrates the scaled bridge deck cross-section used in wind tunnel testing. The 1:40 scaled bridge deck was 6 m long and 1.042 m wide and included guard rails and wind shields. The vertical posts of the guard rail (refer to Figure 3) are equally spaced at 5 cm longitudinally along the deck. The wind shield has a longitudinal spacing between posts of 10 cm. Given that the purpose of wind shields is to protect vehicles, some scaled models of a truck and a bus were included and are depicted in Figure 4 (Politecnico di Milano, 2010). The bus and truck model were placed in Lane 1 which is 200 mm away from the windward edge. The bus was located 1.125 m along the 6 m span and the truck was located at 3.55 m along the 6 m span (refer to Figure 5). This was deemed to be a sufficient distance between vehicle models to avoid interference (Politecnico di Milano, 2010). A small force balance system (Ruag SG-Balance) was fitted inside each vehicle model to measure all six components of wind force and moment (refer to Figure 6).

The location of the Queensferry Crossing bridge is an area with prevailing south-westerly winds. Therefore, the action of winds perpendicular to the bridge was assessed for several yaw wind angles (-30°, -15°, 0°, and +15°). The first two represent the prevailing south-westerly winds, 0° represents wind perpendicular to the bridge, and +15° represents north-westerly winds. The tests have been carried out in calibrated uniform turbulence. The simulation of the natural wind turbulence in the test section has been achieved by using passive turbulence generators (spires) placed at the beginning of the test chamber. The profile of wind velocity and wind turbulence have been measured by using four cobra probes. According to the wind tunnel report (Politecnico di Milano, 2010), the mean inlet velocity was 4 m/s and the mean turbulence intensity was 0.05.
In the Wind Shield Studies wind tunnel tests, a vertical shaped post and a kinked shaped post, both of which were approximately 3 m high with similar porosities were analysed. The aim was to identify whether the post shape, which affects the ease with which wind shields can be climbed, had a significant influence on the cross-wind force on vehicles. It was found that the post shape did not significantly affect the performance of the wind shields.

4. CFD Model

4.1 Geometry

The geometry used in this study was a 3D model of the Queensferry Crossing bridge and is an exact match of the wind tunnel test model (scale 1:40, Figure 6.). The geometry was created in AutoCAD (AutoCAD, 2020) and exported to Stereolithography (STL) files in American Standard Code for Information Interchange (ASCII) format. The bus and truck vehicle models were included in simulations, along with the guard rails and the vertical post wind shield as can be seen in Figure 7. The geometry was imported into the CFD program OpenFOAM.

4.2 Domain

The extents of the computational domain are shown in Figure 8. The domain is 10 m long, 5 m wide and 2 m high (referred to as Domain 1). The blockage ratio for this domain is 0.033. These dimensions are smaller than the size of the wind tunnel to save on computation time and effort. A domain sensitivity study in Section 5.2 demonstrates that the Domain 1 size is sufficient. In Figure 8, the red face, which is the left side of the domain, is the inlet patch and the yellow face, which is the right side if the domain, is the outlet patch. The vehicle-bridge geometry is considered as a wall patch, as are the other four faces of the domain, to replicate the wind tunnel.

4.3 Mesh scheme

The BlockMesh and SnappyHexMesh utilities in OpenFOAM (OpenFOAM Version 7, 2019) were used to create the mesh (Figure 9), which for the remainder of this paper, is referred to as the Medium mesh. Placing the first cell in the near wall region in the viscous sublayer achieves the greatest accuracy in the model (Cochran and Derickson, 2011). This results in a cell size of 0.00004 m, which brings the normalized cell height to 0.00003839 m, based on having a $y^+$ less than 1 in the wall distance Equation 1 (Cochran, L. and Derickson, R., 2011):

$$ y = \frac{y^+ \mu}{\rho u^*} \quad (1) $$

where $\mu$ is the dynamic viscosity, $\rho$ is the fluid density and $u^*$ is the wall shear stress. The overall thickness of the boundary layer is 0.0005 m. The overall mesh quality is deemed to be of high quality as there are no distorted cells in the mesh, the average non-orthogonality is approximately 7°, the maximum skewness is below 4 and the $y^+$ value of the bridge surface is less than 1 (see Table 1 and Figure 10). It should be noted that the $y^+$ value in the vicinity of the vehicles is much higher, however the overall average for the model is 6.57 which is reasonable for Reynolds-averaged Navier–Stokes (RANS) simulations. The mesh has a cell count of around 81 million cells.
4.4 Wall functions
Further addition of layers and reduction of the mesh size did not reduce the y+ value of the vehicles below 1. Therefore, wall functions were applied to interface the region between the wall and the region where turbulence is fully developed. In this case, the first cell can be placed in the logarithmic region of $30 < y+ < 200$ (Cochran, L. and Derickson, R., 2011). In the OpenFOAM model, existing wall functions were modified so that the first cell was located in the viscous sublayer ($y+ < 5$) or the buffer layer ($5 < y+ < 30$) (Liu, 2016). The $kqrWallFunction$ was used on the vehicles to provide a zero-gradient turbulence kinetic energy boundary condition. The $OmegaWallFunction$ was used to provide a constraint on the turbulence specific dissipation. This function can automatically switch between the viscous and logarithmic equation depending on the value of y+, and the buffer layer value can be calculated through blending the viscous and logarithmic sublayer values. The $nutUSpaldingWallFunction$ was used to provide a turbulence viscosity condition based on the y+ value and the friction velocity.

4.5 Boundary conditions
The boundary conditions are summarised in Table 2. All the inlet parameters were set based on the wind tunnel test conditions. The pressure (p) was set as 0 and the condition $InletOutlet$ was set for the other outlet parameters in order to avoid backward flow. On the remaining boundaries, the pressure (p) was set as $zeroGradient$ so that the quantity is developed in space and its gradient is equal to zero in the direction perpendicular to the boundary. The velocity (U) was set as $noSlip$ to create smooth flow conditions.

4.6 Solution Scheme
The $SimpleFoam$ solver was used with a convergence tolerance set to $1e^{-4}$ for p and $1e^{-6}$ for U, k and omega, respectively. Convergence of drag and lift forces around the vehicles was achieved once aerodynamic forces stabilized around a constant value.

5. Verification of CFD Model
5.1 Mesh Sensitivity Study
The sensitivity of the mesh size was investigated for all cases of yaw angle. The Medium mesh was refined (Fine) and made coarser (Coarse) as shown in Table 3. The overall mesh quality of each mesh was deemed to be of high quality as there are no distorted cells, the average non-orthogonality is approximately $7^\circ$ and the maximum skewness is below 4. A visual comparison of the three mesh schemes is illustrated in Figure 11. The feature refinement is the most pivotal process of the mesh building as it relates to precision of the geometry generation. As it shows in the table, one level increase of feature refinement on the bridge leads to a significant promotion on y+ value. The number of cell and running time are drastically changed as well.

Aerodynamic coefficients are non-dimensional numbers that are used to determine the aerodynamic characteristics of an object in a fluid environment. Generally, aerodynamic coefficients are equal to the corresponding force divided by the product of velocity pressure and frontal area. More specifically, in this paper, the aerodynamic coefficients, especially drag coefficient, lift coefficient and overturning moment coefficient, were calculated from the
Accepted manuscript doi: 10.1680/jbren.21.00095

Aerodynamic forces determined in CFD simulations of all three mesh sizes, using Equation 2 and 3 respectively (Politecnico di Milano, 2010):

\[ C_{Fi} = \frac{F_i}{\bar{q}_H L J} \quad i = x, y; \quad j = B, T \]  \hspace{1cm} (2)

\[ C_{Mi} = \frac{M_i}{\bar{q}_H L J H} \quad j = B, T \]  \hspace{1cm} (3)

where, \( F_x \) is the drag force, \( F_y \) is the lift force, \( M_x \) is the moment, \( L \) and \( H \) are the characteristic length and height of the vehicle, respectively, \( \bar{q}_H \) is the mean dynamic pressure at the reference height which can be calculated by \( \bar{q}_H = \frac{1}{2} \rho U_H^{-2} \), \( \rho \) is the density of the air (1.225 kg/m\(^3\)), and \( U_H \) is the mean velocity at the reference height. The reference height was 0.9 m which is 0.04 m above the deck surface. The \( U_H \) was set as 3.75 m/s according to the simulated mean velocity. The aerodynamic coefficients for each of the three mesh density schemes are presented in Table 4 along with the corresponding values determined in the wind tunnel tests.

From Table 4 it is evident that the fine mesh produced results with best agreement with the aerodynamic coefficients found from wind tunnel testing. The Fine mesh density model was repeated for yaw angles of -30\(^\circ\), -15\(^\circ\) and +15\(^\circ\) which resulted in models of over 200 million cells each. Figure 12 illustrates the results of a comparison between wind tunnel test results and results of CFD models with a variety of mesh densities for the four yaw wind angles, for both the truck and the bus. Results indicate excellent agreement of the aerodynamic coefficients for the fine mesh and thus represents verification of the model.

The comparison of results between CFD models and wind tunnel tests is shown in Table 5. For the fine mesh case, the mean percentage difference of drag coefficient is only 1.3% lower compared with the medium mesh. However, the mean percentage difference of the overturning moment coefficient is 3.4% compared to the medium cases and the mean difference in lift coefficients is significantly greater. Moreover, the max results difference suggests that outcomes from fine mesh cases show significantly better stability than the others, especially when it comes to the lift coefficient. The max lift coefficient difference from medium and coarse cases can be around five times larger than the fine mesh. Besides, the overturning moment coefficient is the relatively best predicted aerodynamic coefficients from the perspective of mean and max results difference compared to the wind tunnel results. Overall, the mean percentage difference for the fine mesh cases is 11.9%, the percentage difference for the medium mesh cases is 19.9% and the coarse mesh cases is 31.3%.

Therefore, a sufficiently refined mesh scheme is essential for accurate simulations.

5.2 Domain Sensitivity Study – larger domain

Tominaga et al., (2008) have shown for a single building model, that the lateral and top boundary should be set 5H or more away from the building, (where H is height of the target building). They also showed that the outflow boundary should be set at least 10H behind the building. In this study, a larger domain (Domain 2) and a full-size domain (Domain 3) which is the size of the unscaled wind tunnel were used in the 0\(^\circ\) yaw angle case to investigate if the
size of the domain could affect the turbulence and subsequent simulation accuracy. The fine mesh scheme was applied to all three domains. The details of domains are shown in Table 6. As shown in Figure 13, the recirculation zone for Domain 1 occurs within the domain size and the domain size is adequate to capture the development of turbulence. Results in Figure 14 indicate that the aerodynamic coefficients are similar for all three domains to those determined from wind tunnel tests. A smaller domain can obtain results that are closer to the experimental results, especially for the lift coefficient. However, a certain amount of computational power is saved by reducing the domain size so that near wall region layers are able to be further refined. Thus, the domain size used in modelling (Domain 1) was found to not significantly affect the simulation results and is appropriate considering the need to be efficient with computational power.

6. Validation – Assessment of a Kinked Post Wind Shield
To validate the CFD model, a different shape wind shield was modelled and compared with wind tunnel test results (Politecnico di Milano, 2010). Details of the kinked post are given in Figure 15(a). The wind shield has a spacing between posts along the deck axis equal to 10 cm at the model scale. Figure 15(c) illustrates the geometry of the kinked post wind shield created within the 3D CFD model. All other parameters of the 3D CFD model remain unchanged.

The comparison of aerodynamic coefficients for the bus and truck between CFD simulations and wind tunnel experiments are shown in Figure 16. To assist comparison, the same scale is used in Figure 16 as in Figure 14. Results show similar trends and good agreement. Although there are some inaccurate results, especially for the lift coefficient prediction, other results show good match with experimental results. The average result difference for the drag coefficients is less than 10%. As for the overturning moment coefficient, the average difference is 11% and the max difference is less than 20%. The results suggest that the CFD model accurately captures the force coefficients of the wind tunnel data.

7. Conclusion
This paper presents a validation and verification study comparing wind tunnel test results with results from CFD models of a bridge section containing a representative truck, bus and two types of wind shielding. The 3D CFD models used the RANS method with the k-ω-SST turbulence model. Well-structured mesh schemes were created and the findings from the simulations were as follows:

- a mesh sensitivity study showed that a fine mesh scheme was required to capture the drag and lift coefficients of the vehicles on the bridge.
- a domain sensitivity study showed that results were independent of the domain sizes investigated and therefore a smaller domain is appropriate to balance computational power demands of mesh refinement.
the developed 3D CFD model was successfully validated by comparing two different types of wind shield data with wind tunnel data. Results demonstrate that the 3D CFD can accurately determine the aerodynamic coefficients of lift, drag and overturning moment, to a similar level of accuracy to that of wind tunnel tests.

Acknowledgments

The authors would like to acknowledge the Ph.D. scholarship awarded jointly by University College Dublin and the China Scholarship Council (No.201908300012). The authors would like to acknowledge the support given by Transport Scotland and Arup Consulting Engineers to this work. The authors also wish to acknowledge the Irish Centre for High-End Computing (ICHEC) for the provision of computational facilities and support.

Notation

\( \mu \) dynamic viscosity
\( \rho \) fluid density
\( u^* \) wall shear stress
\( F_x \) drag force
\( F_y \) lift force
\( M_x \) overturning moment

\( L \) characteristic length of the vehicle
\( H \) characteristic height of the vehicle
\( \bar{q}_H \) mean dynamic pressure at the reference height
\( U_H \) mean velocity at the reference height
\( P \) air pressure
\( U \) wind velocity
\( k \) turbulent kinetic energy

References


OpenFOAM Version 7 (2019) *The OpenFOAM Foundation*. Available at: https://openfoam.org/release/7/.

*Queensferry Crossing* (2014) *THE FORTH BRIDGES.* Available at: https://www.theforthbridges.org/queensferry-crossing/.


**Table 1. Summary of y+ value in all fine mesh simulations**

<table>
<thead>
<tr>
<th></th>
<th>$y_{max}^+$</th>
<th>$y_{mean}^+$</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>-30° case</td>
<td>-15° case</td>
</tr>
<tr>
<td>Bridge</td>
<td>26.74</td>
<td>3.82</td>
</tr>
<tr>
<td>Bus</td>
<td>33.80</td>
<td>4.33</td>
</tr>
<tr>
<td>Truck</td>
<td>29.98</td>
<td>3.89</td>
</tr>
</tbody>
</table>

Downloaded by [UNIVERSITY COLLEGE DUBLIN - JIL] on [13/07/22]. Copyright © ICE Publishing, all rights reserved.
Table 2. Summary of boundary conditions use in the model

<table>
<thead>
<tr>
<th>k-Omega SST RANS</th>
<th>Patch</th>
<th>Parameter</th>
<th>Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>nut</td>
<td>p</td>
<td>zeroGradient</td>
</tr>
<tr>
<td></td>
<td></td>
<td>U</td>
<td>fixedValue (U= 4 m/s), the same as the wind tunnel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>k</td>
<td>fixedValue (k = 0.06 $m^2/s^2$), calculated by $k = \frac{3}{2}(UI)^2$, where I=0.05, the same as in the wind tunnel</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Omega</td>
<td>fixedValue ($\omega = 0.35 \text{ s}^{-1}$), calculated by $\omega = 0.09 \frac{1}{4} \frac{\sqrt{k}}{l}$ where mixing length is 0.7 m.</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td>The actual values of $k$ and $\omega$ will be re-calculated by transport equations in the turbulence model.</td>
</tr>
<tr>
<td>Outlet</td>
<td>nut</td>
<td>p</td>
<td>fixedValue (p=0)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>U</td>
<td>inletOutlet</td>
</tr>
<tr>
<td></td>
<td></td>
<td>k</td>
<td>inletOutlet</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Omega</td>
<td>inletOutlet</td>
</tr>
<tr>
<td>Wall</td>
<td>nut</td>
<td>p</td>
<td>zeroGradient</td>
</tr>
<tr>
<td></td>
<td></td>
<td>U</td>
<td>noSlip</td>
</tr>
<tr>
<td></td>
<td></td>
<td>k</td>
<td>kqRWallFunction on vehicles fixedValue (1e-12) on the bridge surface (to avoid float point problem in OpenFOAM, k is set to be a very small number)</td>
</tr>
<tr>
<td></td>
<td></td>
<td>Omega</td>
<td>omegaWallFunction</td>
</tr>
</tbody>
</table>
Table 3. Summary of mesh details in mesh sensitivity study

<table>
<thead>
<tr>
<th>Mesh Density</th>
<th>Coarse</th>
<th>Medium</th>
<th>Fine</th>
</tr>
</thead>
<tbody>
<tr>
<td>BlockMesh Cell Size</td>
<td>0.25 m</td>
<td>0.25 m</td>
<td>0.25 m</td>
</tr>
<tr>
<td>Refinement Box Level</td>
<td>Bridge</td>
<td>5</td>
<td>5</td>
</tr>
<tr>
<td></td>
<td>Vehicle</td>
<td>-</td>
<td>6</td>
</tr>
<tr>
<td>Surface Refinement Level</td>
<td>Bridge</td>
<td>5</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>Vehicle</td>
<td>5</td>
<td>6</td>
</tr>
<tr>
<td>Feature Refinement Level</td>
<td>Bridge</td>
<td>6</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>Vehicle</td>
<td>5</td>
<td>6</td>
</tr>
<tr>
<td>Minimum Cell Size</td>
<td>0.00004 m</td>
<td>0.00004 m</td>
<td>0.00004 m</td>
</tr>
<tr>
<td>Total Number of Cells</td>
<td>68,963,213</td>
<td>80,745,362</td>
<td>98,537,504</td>
</tr>
<tr>
<td>% Difference in Cell Numbers</td>
<td>-14.6%</td>
<td>-</td>
<td>+22.0%</td>
</tr>
<tr>
<td>% Difference in Run Time</td>
<td>-25.0%</td>
<td>-</td>
<td>+37.5%</td>
</tr>
<tr>
<td>$y_{max}$</td>
<td>160.32</td>
<td>109.52</td>
<td>51.43</td>
</tr>
<tr>
<td>% Difference in $y_{max}$</td>
<td>+46.4%</td>
<td>-</td>
<td>-53.0%</td>
</tr>
<tr>
<td>$y_{mean}$</td>
<td>10.01</td>
<td>6.57</td>
<td>4.01</td>
</tr>
<tr>
<td>% Difference in $y_{mean}$</td>
<td>+52.4%</td>
<td>-</td>
<td>-38.9%</td>
</tr>
<tr>
<td>$y_{min}$</td>
<td>1.67</td>
<td>0.26</td>
<td>0.007</td>
</tr>
<tr>
<td>% Difference in $y_{min}$</td>
<td>+542.3%</td>
<td>-</td>
<td>-97.3%</td>
</tr>
</tbody>
</table>

Table 4. Comparison of aerodynamic coefficients

<table>
<thead>
<tr>
<th>Mesh</th>
<th>$C_{D,bus}$</th>
<th>$C_{D,truck}$</th>
<th>$C_{L,bus}$</th>
<th>$C_{L,truck}$</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>3.05</td>
<td>2.43</td>
<td>0.50</td>
<td>0.28</td>
</tr>
<tr>
<td>Medium</td>
<td>1.59</td>
<td>1.22</td>
<td>0.39</td>
<td>0.17</td>
</tr>
<tr>
<td>Fine</td>
<td>0.99</td>
<td>0.85</td>
<td>0.22</td>
<td>0.09</td>
</tr>
<tr>
<td>Wind Tunnel Testing</td>
<td>0.95</td>
<td>0.89</td>
<td>0.16</td>
<td>0.15</td>
</tr>
</tbody>
</table>

Table 5. Percentage difference in results between CFD and Wind Tunnel Tests

<table>
<thead>
<tr>
<th>Average Mean Results Difference</th>
<th>Drag Coefficient Difference</th>
<th>Lift Coefficient Difference</th>
<th>Overturning Moment Coefficient Difference</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mean</td>
<td>Max</td>
<td>Mean</td>
<td>Max</td>
</tr>
<tr>
<td>Fine mesh</td>
<td>11.9%</td>
<td>8.8%</td>
<td>20.3%</td>
</tr>
<tr>
<td>Medium mesh</td>
<td>19.9%</td>
<td>10.1%</td>
<td>19.7%</td>
</tr>
<tr>
<td>Coarse mesh</td>
<td>31.3%</td>
<td>16.8%</td>
<td>38.4%</td>
</tr>
</tbody>
</table>
Table 6. Comparison of domain size (all with fine mesh scheme)

<table>
<thead>
<tr>
<th>Domain</th>
<th>Domain Length</th>
<th>Domain Width</th>
<th>Domain Height</th>
</tr>
</thead>
<tbody>
<tr>
<td>Domain 1</td>
<td>10 m</td>
<td>25H</td>
<td>5 m</td>
</tr>
<tr>
<td>Domain 2</td>
<td>20 m</td>
<td>50H</td>
<td>7 m</td>
</tr>
<tr>
<td>Domain 3</td>
<td>36 m</td>
<td>90H</td>
<td>14 m</td>
</tr>
</tbody>
</table>

**Figure captions**

Figure 1. Photograph of Queensferry Crossing bridge (Queensferry Crossing, 2014)
Figure 2. Scaled bridge deck cross section (dimensions in m) (Politecnico di Milano, 2010)
Figure 3. Scaled guard rail and wind shield dimensions (mm) (Politecnico di Milano, 2010)
Figure 4. Vehicle Model Details (dimensions in mm) (Politecnico di Milano, 2010). (a) Side view of Bus, (b) Elevation of Bus, (c) Side view of Truck, (d) Elevatio of Truck
Figure 5. Schematic aerial view of the vehicles on the bridge deck (not to scale)
Figure 6. Photograph of double deck bus frame and force balance sensor (Politecnico di Milano, 2010); (a) with cover removed; and (b) with cover in place
Figure 7. Geometry of CFD model
Figure 8. Computational domain containing the geometry of the bridge deck and vehicles
Figure 9. Slice view of the Medium mesh scheme around the geometry (0° yaw angle). ((a) Full Mesh, (b) Innermost refinement regions
Figure 10. An overview of the y+ value of the geometry for the 0° yaw angle case
Figure 11. Mesh visualization comparison with; (a) general comparison; and (b) closer comparison among coarse mesh (1), medium mesh (2) and fine mesh (3)
Figure 12. Comparison of wind tunnel results with CFD models of different mesh densities for different wind yaw angles; (a) coefficient of drag for the bus; (b) coefficient of drag for the truck; (c) coefficient of lift for the bus; (d) coefficient of lift for the truck; (e) coefficient of overturning moment for the bus and (f) coefficient of overturning moment for the truck.
Figure 13. Flow visualization for Domain 1; (a) overview of the domain; and (b) near model view
Figure 14. Comparison of wind tunnel results with CFD models of different domain size; (a) coefficient of drag for the bus; (b) coefficient of drag for the truck; (c) coefficient of lift for the bus; (d) coefficient of lift for the truck; (e) coefficient of overturning moment for the bus and (f) coefficient of overturning moment for the truck.

Figure 15. Details of the kinked post wind shield (Politecnico di Milano, 2010); (a) dimensions; (b) model used in wind tunnel; and (c) CFD model

Figure 16. Comparison of wind tunnel results with CFD models for a kinked-post wind shield for a variety of wind yaw angels; (a) coefficient of drag for the bus; (b) coefficient of drag for the truck; (c) coefficient of lift for the bus; (d) coefficient of lift for the truck; (e) coefficient of overturning moment for the bus and (f) coefficient of overturning moment for the truck.
Accepted manuscript doi:
10.1680/jbren.21.00095
fig6
Accepted manuscript doi: 10.1680/jbren.21.00095

fig8
Accepted manuscript doi: 10.1680/jbren.21.00095

(a) Full mesh

(b) Innermost refinement regions

fig9
fig12
fig13
Accepted manuscript doi: 10.1680/jbren.21.00095

fig14
fig16