



Trinity College Dublin

Coláiste na Tríonóide, Baile Átha Cliath

The University of Dublin

School of Engineering

Department of Mechanical & Manufacturing Engineering

Vortex-Induced Vibrations of Bluff Decks on Cable-Stayed Bridges Using Computational Fluid Dynamics

by

Marko Đuranović

Supervised by Dr. Craig Meskell

August 2024

A dissertation submitted to Trinity College Dublin in partial fulfilment of the
requirements for the degree of

Doctor of Philosophy

Declaration

I, the undersigned, declare that this work has not previously been submitted as an exercise for a degree at this, or any other University, and that unless otherwise stated, is my own work.

I, the undersigned, agree to deposit this thesis in the University's open-access institutional repository or allow the library to do so on my behalf, subject to Irish Copyright Legislation and Trinity College Library conditions of use and acknowledgement.

I consent to the examiner retaining a copy of the thesis beyond the examining period, should they so wish (EU GDPR May 2018).



Marko Đuranović

Date:

Abstract

For long and flexible bridges, dynamic wind loading is an important aspect of the design which, if not adequately considered, can impact not just the comfort of the bridge users but also their safety. The standards dealing with adverse wind effects include an amount of uncertainty which is mitigated by prescribing wind tunnel tests for the cases outside the standards' scope. In this research, a number of areas where the design standards need further development were identified and the work towards their improvement was undertaken.

While wind tunnel tests are an essential part of the majority of long span bridge designs, they can adversely impact both the project timeline and expenditure. By gaining prior knowledge of what to expect from these tests, engineers can reduce the time spent on their execution. Computational fluid dynamics has been used in bridge engineering research in the past decade, but in practice, it is not routinely used in design. Furthermore, most of the published work in this field dealt with streamlined bridge deck sections, while bluff, i.e., aerodynamically more complex sections, have received little attention.

The research work described here was complemented by the results from the wind tunnel testing campaigns conducted as part of the design of three cable-stayed bridges.

The first part of this research explores the level of accuracy and practical applicability of computational models built in commercial CFD software when applied in the bridge design industry for bluff bridge decks. A selection of physical and geometric modelling choices was evaluated against the data available from the wind tunnel tests. The CFD models developed were of such size that a desktop workstation can provide sufficient computational power while delivering results in an acceptable time. It was found that 2D RANS models can be successfully used to deliver the static force coefficients and the vortex-induced vibration response. Extending the models to 3D did not provide improved results, despite the much greater computational effort. However, it was found that having a simplified geometry which includes a representation of the deck barrier is essential, although this can be greatly simplified as long as the gross blockage is comparable to the target geometry. The flow structure interaction model which allowed the bridge deck to deflect dynamically in response to the aerodynamic loading was able to predict the VIV lock-in range, but the oscillation amplitude was overestimated by approximately 40%. When a parametric study of structural damping is of interest, a prescribed motion model in combination with an energy balance approach can provide results in a computationally lighter manner.

The second part of this work dealt with the effects of the overhang length on the bridge deck VIV response. The overall goal is to instigate further development of the bridge standards dealing with aerodynamics to include the overhang as a parameter for evaluation of the bridge's susceptibility to VIV. For this purpose, two sets of generic decks were developed, each comprising five geometries with different lengths of the overhang. Deck shapes with and without the barriers were considered. In the case of the bridge deck without the barrier, it was found that with increasing overhang length the maximum displacement amplitude drastically decreases. For two geometries with the largest overhang, an additional LCO region was observed which was attributed to the motion-induced vortices. It was shown that the system can enter this LCO only with an initial kick in the displacement amplitude. When it comes to the width of the LCO region, it experienced a decrease with the increasing overhang length. A somewhat different behaviour was observed for the geometries with the barriers. Initially, the displacement amplitude increased but with further increase of the overhang length, the displacement amplitude reduced however it remained above the initial one. The width of the LCO region experienced similar behaviour with the increasing overhang length as in the case of the deck without the barrier. It was noted that for the case with the barrier, the second LCO region was not clearly visible although some similar characteristics were observed.

It was shown that CFD can be used as a preliminary design tool to provide information valuable at the onset of the wind tunnel testing. Furthermore, the research showed that the design standards might be conservative when estimating the VIV response of the bridge decks with relatively long overhangs.

Acknowledgements

I would like to sincerely thank you, Craig, for all the guidance and support during the past, almost, four years. Thank you for having patience with me and believing in the project despite my lack of knowledge in CFD and general aerodynamics at the start of it. Thank you for the humour and words of encouragement when things were bumpy, I am truly grateful for that. You made me feel I am not just a mentee to a mentor but your colleague as well. I hope you enjoyed this journey as much as I did.

A big thank you to Roughan & O'Donovan for sponsoring this research, especially to Tony Dempsey and Joe Kelly who supported the idea of me doing the PhD and staying with the company. To my friends and all the amazing people that I met in Dublin, who made me feel I am never alone, Walter, Karlo, Alen, Rebekkah, Luca, Barbara, Nicholas, Rob, Cliona, Caitríona, Róisín, Conor. Special thanks to my dearest friends Ilaria and Inês whose friendship I will forever treasure – Swanward office rules them all! Thank you to my Irish bros, Iomhar, Lorcan and Ciarán for always making me feel like I am at home in Ireland. Thank you to Sunčica, Janko and Luka for being my family in Ireland. Thank you to my friends in Montenegro who never made me feel I left.

Thank you, Milica, for being by my side wherever I was, for giving me strength and motivation when I needed them the most, and for believing in me from the day we met. *Volim te mišu.*

Finally, thank you to my family. To my father, Nebojša, for always pushing me to strive for more. To my mother, Zorica, for her unconditional support and love. To my three sisters, Jelena, Marija and Ksenija, whose love and friendship mean the world to me. To my nieces, Julija and Petra, who brought so much joy and happiness into my life. To all of you, I hope to repay at least a bit of everything you gave me. *Volim vas sve puno.*

To my grandfather, *dede Pero* it is about bridges, you would have loved it.

Table of Contents

Declaration.....	i
Abstract.....	iii
Acknowledgements.....	v
List of Figures	xi
List of Tables.....	xvii
Nomenclature	xix
Chapter 1 Introduction.....	21
1.1 Problem Definition	21
1.2 Thesis Objectives	25
1.3 Thesis Structure.....	25
Chapter 2 Vortex-Induced Vibration in Context	27
2.1 Design Codes for Aerodynamics of Bridges.....	28
2.1.1 Design Code in Ireland and the UK.....	28
2.1.2 Other Design Codes	31
2.1.3 Opportunities for Further Refinement of Design Codes.....	32
2.2 Flow Around Bluff Bodies	32
2.2.1 General	32
2.2.2 Flow-Induced Vibrations.....	34
2.2.3 Vortex-Induced Vibrations.....	35
2.2.4 Divergent Amplitude Response	39
2.2.5 Circular Sections	40
2.2.6 Rectangular Sections	41
2.2.7 Splitter Plate Effect	44
2.2.8 Bridge Deck Sections	45
2.3 Vortex-Induced Vibrations of Bridge Decks.....	47
2.3.1 Background.....	47
2.3.2 Historic Cases.....	47
2.3.3 Reynolds Number Dependency	48
2.3.4 Turbulence Effect.....	48
2.3.5 Spanwise Correlation.....	49
2.3.6 Influence of Structural and Non-Structural Elements and Mitigation of VIV in Bridges	50

2.4	Galloping and Flutter of Bridge Decks	52
2.5	CFD in Bridge Aerodynamics	53
2.5.1	Basics of Fluid Dynamics.....	53
2.5.2	Background to CFD modelling	55
2.5.3	Applicability in Bridge Engineering and Limitations	57
2.6	Overview	60
Chapter 3	Available Wind Tunnel Data.....	63
3.1	Northern Spire Bridge	63
3.2	Clyde Bridge	67
3.3	Narrow Water Bridge	70
3.4	Geometric Considerations.....	72
3.5	Discussion.....	73
Chapter 4	Methodology	75
4.1	Northern Spire Bridge CFD Model.....	75
4.1.1	Modelling Choices	75
4.1.2	Meshing Methodology and Grid Sensitivity	77
4.1.3	Boundary Conditions	78
4.1.4	Turbulence Modelling	79
4.2	Overhang Length Effects Study	79
4.2.1	Modelling Choices	79
4.2.2	Meshing Methodology and Grid Sensitivity	80
4.2.3	Boundary Conditions	82
4.2.4	Turbulence Modelling	82
4.3	Fluid-Structure Interaction Model	82
4.4	Prescribed Motion Model and the Energy Balance Approach	83
Chapter 5	CFD as an Early Aerodynamic Design Tool.....	87
5.1	Fixed Model.....	87
5.1.1	Approximation of Geometry as 2D Profile	87
5.1.2	3-Dimensional Models.....	90
5.1.3	Turbulence Modelling	92
5.1.4	Transient Analysis.....	94
5.2	Free-to-Oscillate Model (FSI)	95
5.3	Prescribed Motion Model (PM).....	98
5.4	Application to Other Case Studies.....	102
5.5	Synthesis	103

Chapter 6	Overhang Length and Bridge Deck VIV	105
6.1	Introduction.....	105
6.2	Deck Without the Barrier	106
6.2.1	Geometry S0, S10 and S25.....	106
6.2.2	Geometry S40 and S50	107
6.2.3	Discussion	110
6.3	Deck With the Barrier.....	112
6.3.1	Geometry SB0.....	112
6.3.2	Geometry SB10 and SB25.....	113
6.3.3	Geometry SB40 and SB50.....	114
6.3.4	Discussion	116
6.4	Synthesis.....	116
Chapter 7	Conclusions.....	118
7.1	Summary	118
7.2	Future Work	120
Bibliography		123
Appendix A	Effects of Turbulence Intensity on the VIV of Bridges	131
Appendix B	UDF Functions.....	134

List of Figures

Figure 1.1. Bridges over river Vltava, Czech Republic	21
Figure 1.2. Hong Kong–Zhuhai–Macau Bridge	21
Figure 1.3. Vortex shedding process behind a bridge deck and some of the elements found on a typical deck.....	22
Figure 1.4. Indicative bridge design workflow – aerodynamic aspects.....	22
Figure 1.5. A boundary layer wind tunnel and its components (adapted from Roberts, 2013).....	23
Figure 1.6. Example of a wind tunnel test setup in bridge engineering.....	23
Figure 1.7. Disadvantages and advantages of undertaking wind tunnel tests in bridge engineering	24
Figure 1.8. Disadvantages and advantages of using CFD in bridge engineering	24
Figure 1.9. Example of the flow visualisation around a bridge deck obtained in CFD	25
Figure 2.1. Bridge deck types as defined in PD 6688-1-4	29
Figure 2.2. Strouhal number for different deck types as defined in PD 6688-1-4; see Figure 2.1 for b^* and d_4 and deck types.....	30
Figure 2.3. Edge detail geometry defined in PD 6688-1-4	31
Figure 2.4. Particle pathlines in one-dimensional laminar and turbulent flows (adapted from Pritchard, 2011).....	33
Figure 2.5. Airfoil submerged in a moving fluid: (a) pressure, (b) shear/viscous stresses, (c) drag and lift forces and pitching moment (adapted from Gerhart et al., 2016)	34
Figure 2.6. Classification of flow-induced vibrations	34
Figure 2.7. Schematic representation of wind-induced vibrations vs wind velocity (adapted from Fujino & Siringoringo, 2013)	35
Figure 2.8. Karman vortex street above Tristan da Cunha island caused by the wind (photograph taken from NASA MODIS Rapid Response System, 2010)	36
Figure 2.9. Effects of turbulence in the boundary layer on the flow separation location; a) smooth ball and b) ball with rough surface (Gerhart et al., 2016).....	36
Figure 2.10. Vortex induced vibrations of a circular free-to-oscillate cylinder (adapted from Feng, 1968); Y – displacement amplitude; D – section diameter; f_{vs} – shedding frequency; f_n – natural frequency	37
Figure 2.11. Lock-in range width for different mass-damping parameters for a circular cross-section (adapted from Chen, 1985)	37
Figure 2.12. Ranking of the edge shapes based on their aerodynamic performance as defined by Shiraishi & Matsumoto, (1983)	38
Figure 2.13. Dependence of the Strouhal number on the Reynolds number for a circular cross-section in a low-turbulent flow (adapted from Païdoussis et al., 2010)	40
Figure 2.14. Results for a circular cross-section forced to oscillate in a moving fluid compared to free-to-oscillate model (adapted from Staubli, 1983)	41
Figure 2.15. Different classes of vortex shedding formation (adapted from Deniz & Staubli, 1997).....	41
Figure 2.16. Strouhal number vs B/D ratio (adapted from Nakamura et al., 1991)	42
Figure 2.17. Strouhal number vs Reynolds number for a rectangular section (adapted from Schewe, 2013)	42
Figure 2.18. Correlation coefficient for three flow velocities; δ – correlation length; D – rectangle depth; Y – oscillation amplitude; U_r – reduced velocity (adapted from Sun et al., 2019)	43
Figure 2.19. Sections equipped with splitter plates tested by a) Matsumoto et al., (1993), and b) Sahu et al., (2019)	44
Figure 2.20. Vortex shedding process developing behind a circular cylinder (adapted from THINK Fluid Dynamix, 2017)45	

Figure 2.21. Typical bridge deck cross-section (Narrow Water Bridge deck)	45
Figure 2.22. Full aeroelastic model and surrounding terrain for the Megami Bridge (Fujino et al., 2012).....	46
Figure 2.23. Hexagonal box section tested by Matsumoto et al., (1993); dimensions in full scale and in [model scale] ...	49
Figure 2.24. Some of the models [at model scale] investigated by Kawatani et al., (1993)	49
Figure 2.25. Spanwise correlation of VIF for the varying angle of attack: a) trapezoidal cross-section, b) streamlined bridge girder (adapted from M. Li et al., 2020); δ – correlation length; B – section width	50
Figure 2.26. Trapezoidal bridge deck showing the fairing angle (β).....	51
Figure 2.27. Cable anchorages protruding the deck of the William Dargan Bridge, Dublin, Ireland	51
Figure 2.28. Guide vanes installed on the Great Belt Bridge (Larsen et al., 2000).....	51
Figure 2.29. Temporary water-filled barriers placed on the Humen Pearl River Bridge (Y. Ge et al., 2022)	52
Figure 2.30. Flow spoilers installed on the Humen Pearl River Bridge (Y. Ge et al., 2022).....	52
Figure 2.31. Typical velocity measurement in turbulent flow (adapted from Versteeg & Malalasekera, 2007)	56
Figure 2.32. Representation of resolved turbulent scales at a point in a turbulent flow	57
Figure 2.33. Original deck shape analysed by Nieto et al. (2009).....	58
Figure 2.34. The Normandy Bridge deck section analysed by Bruno & Mancini, (2002)	58
Figure 2.35. The Great Belt Bridge section analysed by Bruno & Khris, (2003) and Bruno & Mancini, (2002).....	58
Figure 2.36. Trapezoidal deck section [model scale] analysed by Mannini et al., (2010)	59
Figure 2.37. Stonecutters Bridge deck section analysed by Goering & Ramponi (2019).....	59
Figure 2.38. Twin box deck analysed by Jeong et al., (2019).....	59
Figure 2.39. VIV results (at model scale) of the analyses done by Zhan et al., (2014).....	59
Figure 3.1. The Northern Spire Bridge in Sunderland, UK	64
Figure 3.2. Northern Spire Bridge deck section: dimensions at full scale and [model scale]; adopted sign convention, deck width B and deck depth D	64
Figure 3.3. Dynamic model assembly and the deck section model (Flamand & Knapp, 2018).....	64
Figure 3.4. Static aerodynamic coefficients for the Northern Spire Bridge deck section shown in Figure 3.2 (adapted from Flamand & Knapp, 2018)	65
Figure 3.5. Vertical deck response at full scale for the Northern Spire Bridge deck section shown in Figure 3.2 (adapted from Flamand & Knapp, 2018).....	66
Figure 3.6. Spectrogram of the deck oscillation for a range of wind velocities for the Northern Spire Bridge deck section with the Strouhal law curve marked in red (adapted from Flamand & Knapp, 2018)	66
Figure 3.7. The Clyde Bridge in Scotland in the open position (left) and closed (right)	67
Figure 3.8. Clyde Bridge deck section: dimensions at full scale and [model scale]; adopted sign convention, deck width B and deck depth D	67
Figure 3.9. Test setup for the Clyde Bridge deck model (Carusone & Flamand, 2022).....	68
Figure 3.10. Static aerodynamic coefficients for the Clyde Bridge deck section with the fairing plates as shown in Figure 3.8 (adapted from Carusone & Flamand, 2022).....	68
Figure 3.11. Vertical deck response at full scale for the Clyde Bridge deck section without the fairing plates as shown in Figure 3.8 (adapted from Carusone & Flamand, 2022).....	69
Figure 3.12. Spectrogram of the deck oscillation for a range of wind velocities for the Clyde Bridge deck section with the Strouhal law curve marked in red (adapted from Carusone & Flamand, 2022)	69
Figure 3.13. Spectrogram of the flow fluctuations measured in the Clyde Bridge deck wake for a range of wind velocities with the Strouhal law curve marked in red (adapted from Carusone & Flamand, 2022)	70

Figure 3.14. Future Narrow Water Bridge in Louth Co., Ireland	70
Figure 3.15. Narrow Water Bridge deck section: dimensions at full scale and [model scale]; adopted sign convention, deck width B and deck depth D	70
Figure 3.16. Model assembly for one of the tested deck sections of the Narrow Water Bridge (Svend Ole Hansen ApS, 2023)	71
Figure 3.17. Static aerodynamic coefficients for the Narrow Water Bridge deck with the wind shielding as shown in Figure 3.15 (adapted from Svend Ole Hansen ApS, 2023)	71
Figure 3.18. Vertical deck response at full scale for the Narrow Water Bridge deck, with and without the wind shielding, as shown in Figure 3.15 (adapted from Svend Ole Hansen ApS, 2023).....	72
Figure 3.19. Wind shielding and traffic barriers installed on the Narrow Water Bridge model (Svend Ole Hansen ApS, 2023)	73
Figure 3.20. The barrier installed on the Northern Spire Bridge model (Flamand & Knapp, 2018)	73
Figure 4.1. Different modelling choices, arranged based on the computational effort needed	76
Figure 4.2. Multi-body domain division and grid details (model SM03)	77
Figure 4.3. Geometry S0 at full scale and [model scale]	80
Figure 4.4. Geometry S50 at full scale and [model scale]	80
Figure 4.5. Barrier detail for SB0-SB50 geometries; dimensions in full scale and [model scale]	80
Figure 4.6. Domain division and grid details (geometry SB25)	81
Figure 4.7. Single degree of freedom dynamic model	83
Figure 5.1. Bare deck geometry at model scale [mm] – model SM01	87
Figure 5.2. Static force coefficients for three deck models compared to the wind tunnel tests (full deck).....	88
Figure 5.3. Effect of the number of barrier openings as a change in error with respect to the no-barrier case, for a) lift, b) drag and c) moment coefficient (models SM11-SM14).....	89
Figure 5.4. 2.5D model (model SM21)	90
Figure 5.5. 3D model (model SM22)	90
Figure 5.6. (a) The 3D model with applied double symmetry boundary conditions (model SM22); (b) polyhedral mesh available in Fluent – model SM22 and in a similar way developed for model SM21	91
Figure 5.7. Results of the 2.5D model data (SM21) compared to the wind tunnel and 2D data (SM32)	91
Figure 5.8. Results of the 3D model (SM22) compared to the wind tunnel and 2D data (SM32)	91
Figure 5.9. 3D model (model SM22); Y -Vorticity contour (top) and Y -velocity streamlines (bottom)	92
Figure 5.10. Comparison of turbulence models for a) lift, b) drag and c) moment coefficient.....	93
Figure 5.11. Results of the 2D models: with high inflow turbulence intensity (model SM41), and with smooth inflow conditions (model SM03) compared to the wind tunnel data	93
Figure 5.12. Results of the 3D models: with high inflow turbulence intensity (model SM42), and with smooth inflow conditions (model SM22) compared to the wind tunnel data	94
Figure 5.13. Time history of the static coefficients (model TM01)	94
Figure 5.14. Frequency domain analysis of the lift coefficient from the transient simulation (model TM01)	94
Figure 5.15. Deck response with SST (model DM01) and TSST (model DM02) turbulence models compared to the wind tunnel data	95
Figure 5.16. Reduced shedding frequency for SST (model DM01) and TSST (model DM02) turbulence models compared to the Strouhal law	96

Figure 5.17. Spectrogram of the deck displacement obtained in the wind tunnel – generated from the data acquired from CSTB (reduced values) (Flamand & Knapp, 2018)	96
Figure 5.18. Time record of the lift coefficient and deck displacement (DM02) for Ur of a) 17 and b) 20	97
Figure 5.19. Frequency domain of the lift coefficient and deck displacement (DM02) for Ur of a) 17 and b) 20	97
Figure 5.20. Vorticity magnitude in the lock-in region for $Ur = 17$	97
Figure 5.21. Vorticity magnitude at the lock-out for $Ur = 20$	97
Figure 5.22. Fluid (ΔEF) and damping (ΔES) energy increments per oscillation cycle at $Ur = 16.8$	98
Figure 5.23. System net energy ($\Delta ENET$) per oscillation cycle for $Ur = 16.8$; arrows mark the tendency of the system amplitude change depending on the net energy	99
Figure 5.24. 3D plot of energy brought in and energy lost per oscillation cycle for a range of Yr_{RMS} and Ur compared to the “displacement vs flow velocity” plot obtained from the FSI model (model DM02); a) Isometric view, b) top view $\zeta = 0.45\%$ c) top view $\zeta = 0.55\%$	100
Figure 5.25. Time history of the work done by the fluid force on the oscillating body per oscillation cycle for two amplitudes at $Ur = 16.8$	100
Figure 5.26. Aerodynamic (solid line) and damping (dashed line) energy increments per oscillation cycle for a range of flow velocities for $Yr, 1_{RMS} = 6\%$, $Yr, 2_{RMS} = 7\%$ and $Yr, 3_{RMS} = 10\%$	101
Figure 5.27. Static coefficients for the Clyde Bridge deck with the fairing plates as shown in Figure 3.8 compared to the wind tunnel test results	102
Figure 5.28. VIV response curve for the Narrow Water Bridge deck without the wind shielding ($Sc = 47$) as shown in Figure 3.15 compared to the wind tunnel test results	102
Figure 6.1. Prescribed motion model results (geometry S0) in 3D view; dot and cross markers – data points obtained from the CFD simulations; blue surfaces – damping energy calculated using Eq. 4-9 and Eq. 4-12	105
Figure 6.2. Prescribed motion model results for geometry S0	106
Figure 6.3. Prescribed motion model results for geometry S10	107
Figure 6.4. Prescribed motion model results for geometry S25	107
Figure 6.5. Prescribed motion model results for geometry S40	108
Figure 6.6. Prescribed motion model results for geometry S50	109
Figure 6.7. Vorticity field around geometry S50 at the phase angle $\pi/2$ for (Ur, Yr) equal to (8.1,0.4%) in bottom and (9.1,3.4%) in top figure	109
Figure 6.8. Vorticity field around geometry S50 at the phase angle π for (Ur, Yr) equal to (8.1,0.4%) in bottom and (9.1,3.4%) in top figure	110
Figure 6.9. Vorticity field around geometry S50 at the phase angle $3\pi/2$ for (Ur, Yr) equal to (8.1,0.4%) in bottom and (9.1,3.4%) in top figure	110
Figure 6.10. Time history for the three PM-FSI simulations for $Ur = 9.14$ and $Sc = 100$; in blue – $Yr = 3.4\%$ sustained for $0.25T$; in orange – $Yr = 3.4\%$ sustained for $10T$; in green – $Yr = 3.4\%$ sustained for $0.25T$;	112
Figure 6.11. Prescribed motion model results for geometry SB0	113
Figure 6.12. Prescribed motion model results for geometry SB10	114
Figure 6.13. Prescribed motion model results for geometry SB25	114
Figure 6.14. Prescribed motion model results for geometry SB40	115
Figure 6.15. Prescribed motion model results for geometry SB50	115

Figure 6.16. Maximum displacement amplitude (left) and lock-in region width (right) for the deck geometry without the barrier (S0-S50) normalised with values for geometry S0; solid line – lower LCO region; dashed line – upper LCO region.....	116
Figure 6.17. Maximum displacement amplitude (left) and lock-in region width (right) for deck geometry with the barrier (SB0-SB50) normalised with values for geometry SB0	117
Figure 7.1. Methodology for CFD application in bridge engineering at the preliminary design stage.....	119
Figure A - 1. Northern Spire Bridge deck model response for three levels of turbulence intensity; VIV response in the top and spectrogram of the deck displacement in the bottom (adapted from Flamand & Knapp, 2018)	132
Figure A - 2. Clyde Bridge deck model response for two levels of turbulence intensity; VIV response in the top and spectrogram of the deck displacement in the bottom (adapted from Carusone & Flamand, 2022)	133
Figure A - 3. Narrow Water Bridge deck model response for two levels of turbulence intensity with the wind shielding; VIV response in the top and spectrogram of the deck displacement in the bottom (adapted from Svend Ole Hansen ApS, 2023)	133

List of Tables

Table 3.1. Summary deck sections tested in three wind tunnel testing campaigns	63
Table 3.2. Model and full-scale properties (first heave mode) of the tested Northern Spire Bridge deck section.....	65
Table 3.3. Model-scale and full-scale properties (first heave mode) of the tested Clyde Bridge deck section.....	68
Table 3.4. Model and full scale properties (first heave mode) of the tested Narrow Water Bridge deck section.....	71
Table 4.1. Simulation Matrix.....	76
Table 4.2. Grid sensitivity (model SM03)	78
Table 4.3 Sub-domain properties of the medium mesh	78
Table 4.4. Model-scale dimensions of the geometries studied for the overhang length effects.....	79
Table 4.5. Grid sensitivity (model SB0)	82
Table 4.6. Sub-domain properties of the medium mesh	82
Table 5.1. Proposed modelling approach for the preliminary design stage in bridge engineering.....	103
Table A - 1. Summary deck sections tested in three wind tunnel testing campaigns	131

Nomenclature

Roman symbols

A	Area / Projected area
B	Section width (equivalent to b)
B_{eff}	Effective width
B^*	Width between the box girders (equivalent to b^*)
b	Deck section width (PD 6688-1-4)
b^*	Width between the box girders (PD 6688-1-4)
C_D	Drag force coefficient
C_L	Lift force coefficient
C_M	Moment coefficient
c	Leading edge geometry factor (PD 6688-1-4)
D	Section depth (equivalent to d_4)
d_4	Deck section depth (PD 6688-1-4)
E_F	Fluid energy
E_{NET}	Net energy
E_p	Potential energy
E_S	System energy
E_{TOT}	Total energy
E_k	Kinetic energy
E^*	Reduced energy
f_c	Prescribed oscillation frequency
F_D	Drag force
F_L	Lift force
f_n	Natural frequency (equivalent to n_b)
f_{vs}	Vortex shedding frequency
g	Gravity
h	Parapet height (PD 6688-1-4)
I	Moment of inertia
I_u	Turbulence intensity
k	Stiffness
L	Spanwise length
m	Mass
M	Pitching moment
M_a	Mach number
n_b	First natural frequency in bending (PD 6688-1-4)
n_i	Frequency of the i^{th} natural mode of oscillation (PD 6688-1-4)
p	Pressure

P_b	Aerodynamic susceptibility parameter (PD 6688-1-4)
Q	Quality factor
r	Polar radius of gyration (PD 6688-1-4)
Re	Reynolds number
S	Overhang length
Sc	Scruton number
St	Strouhal number
t	Time
t_r	Reduced time
U	Wind velocity / Free stream flow velocity
U_{vs}	Vortex shedding velocity for a fixed model
U_r	Reduced velocity
v_m	Mean wind velocity (PD 6688-1-4)
$V_{crit,i}$	Critical wind velocity for vortex shedding
Y	Displacement amplitude
Y_r	Reduced displacement amplitude
y	Displacement
y_r	Reduced displacement

Greek symbols

α	Angle of attack
β	Fairing angle
δ	Correlation length
δ_s	Damping (logarithmic decrement)
ρ	Air density
$\rho_{i,j}$	Spanwise correlation ratio
μ	Air dynamic viscosity
σ	Normal stress
τ	Shear stress
ϕ	Parapet solidity ratio (PD 6688-1-4)
ω	Natural angular frequency
ω_d	Natural angular frequency of a damped system

Chapter 1

Introduction

1.1 Problem Definition

Bridges, like any other civil structure, are inherently exposed to unpredictable and variable environmental conditions. The crosswind on a bridge can exert significant loads with different characteristics compared to either the self-weight or the traffic load. For stiff and heavy bridges (see for example, Figure 1.1), static wind loads are of bigger concern than dynamic ones, while for lighter and more flexible designs (see for example, Figure 1.2), the latter becomes the limiting issue. Perhaps the most famous example of adverse effects caused by wind is the Tacoma Narrows bridge collapse that occurred in 1940 at a relatively low wind speed. There are two main mechanisms of concern related to wind effects: (i) vortex-induced vibrations, caused by the periodic aerodynamic forces resulting from the wake instability, and (ii) divergent amplitude response, such as galloping and flutter and both were present at Tacoma Narrows.



Figure 1.1. Bridges over river Vltava, Czech Republic¹



Figure 1.2. Hong Kong–Zhuhai–Macau Bridge²

When a bluff body (here a bridge deck) is submerged in a moving fluid (air i.e., the wind) the part of the flow downstream of the body becomes disturbed and as such is called the *wake*. Instability in the wake causes the formation of vortex structures in the flow that become detached (i.e. shed) from either side of the body. As the vortices are shed, the pressure difference on the two sides generates a fluctuating force acting on the deck in the direction perpendicular to the flow direction. For a given cross-section, vortex shedding occurs at a frequency that depends on the flow velocity. As the flow velocity changes, the shedding frequency varies and may match the natural frequency of the structure. When this coincidence happens, the shedding frequency *locks onto* the structural frequency and remains constant for a range of flow velocities. During the lock-in, resonance may occur and high amplitude displacement oscillations may be observed. This phenomenon is called vortex-induced vibrations (VIV), it is self-limiting and occurs for a range of flow velocities outside of which it is manifested with a negligible amplitude. Vortex-induced vibrations in most cases primarily affect user comfort but can also affect the fatigue life of a structure. Figure 1.3 depicts the VIV process behind a typical bridge deck along with some of the main structural and non-structural elements

¹ Source: <https://en.lesso.com/case-study/hong-kong-zhuhai-macao-bridge.html>

² Source: <https://blog.ansharphoto.com/city/aerial-view-vltava-river-prague/>

normally found on bridge decks. As seen, the vortices in the wake are arranged in a spatially periodic pattern that convects downstream. This spatial periodicity is what causes temporal periodicity in the fluctuating force. Vortex shedding is a wake instability which originates from the interaction of the vortices from two sides of the bluff body. This means that other than the deck geometry, what happens in the downwind region also affects the VIV response. By placing a plate (also called a splitter plate) in the wake of a bluff body, the shedding process can be disrupted and so its effects mitigated. As seen in Figure 1.3, bridge decks may have deck ends that extend past the main girders, forming elements which will be referred to as *overhangs*. It might be expected that the overhangs can produce effects similar to those created by the splitter plates.

To avoid adverse effects caused by the wind (static or dynamic loading), bridge engineers follow design standards (i.e. codes) and guidelines developed based on research and past experience in the industry. In Chapter 2, it will be shown that some design codes dealing with bridge aerodynamics recognise the possible existence and impact of the overhangs. The design standard used in Ireland and the UK justifies a factor in an equation used for estimating the deck displacement caused by the VIV by the fact that some decks can have overhangs (TRL, 2002). This is only specified in the background document to the main standard text. In the Japanese design standard, the overhang is, to some degree, better recognised however the underlying effects it has on the shedding process are not explained (Japan Road Association, 2007).

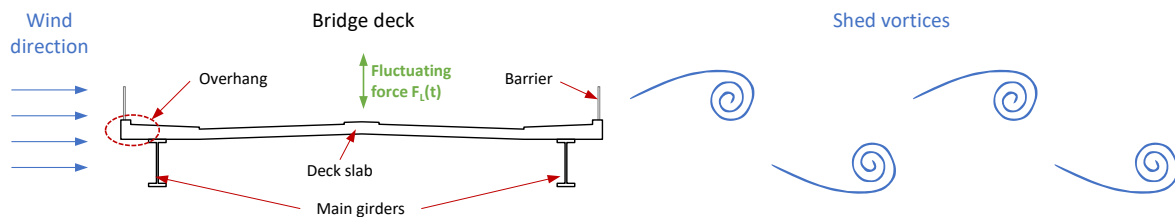


Figure 1.3. Vortex shedding process behind a bridge deck and some of the elements found on a typical deck

Based on their level of development, different design standards deal with wind actions with different levels of complexity. The more developed ones prescribe hand calculations for an initial evaluation of the wind effects on the structure. Depending on the results from these, conservative calculations, the code may require the wind tunnel tests to be included in the design process. While every design is different, most are split into a number of stages that progress from when the initial idea is developed (concept stage) to when the final design is delivered (construction stage). Furthermore, each stage comprises subtasks dealing with design elements characteristic of that stage. Figure 1.4 outlines stages commonly present in most new-build bridge projects along with the subtasks related to aerodynamics.

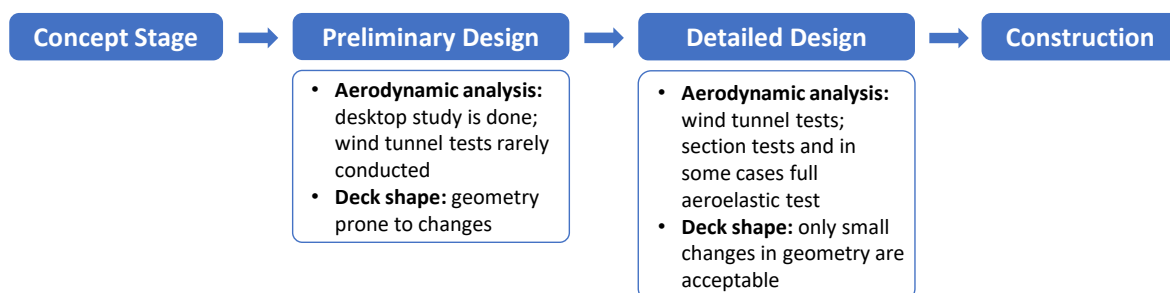


Figure 1.4. Indicative bridge design workflow – aerodynamic aspects

At the preliminary design stage, the aerodynamics are covered as desktop/hand calculations, while commissioning the wind tunnel tests at this stage is rare. The preliminary design stage also comprises decision-making on the bridge deck shape, so the geometry changes are common and acceptable. This means that an aerodynamically suitable deck shape can be implemented with little effort. However, desktop analyses provide limited information on the aerodynamic properties so it is not uncommon that the wind tunnel tests conducted at the detailed design stage identify issues. Furthermore, wind tunnel tests are usually done in parallel with the structural design, hence any changes in the geometry impact the design tasks and can further delay their delivery.

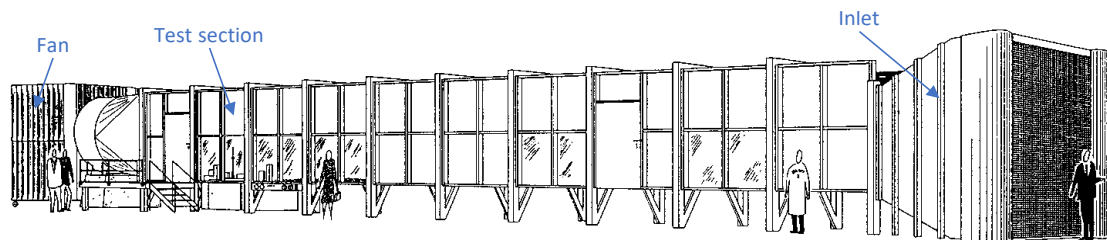


Figure 1.5. A boundary layer wind tunnel and its components (adapted from Roberts, 2013)

Wind tunnel tests are part of the design process for the majority of medium and long-span bridges (spans longer than 100 m). Early wind tunnel tests (1950s) were done in smooth uniform flows in aeronautical wind tunnels (Davenport, 2002). The first *boundary layer* wind tunnel facility built for testing in civil engineering was designed in 1965 by Alan G. Davenport who can be regarded as one of the *fathers* of this field. The name *boundary layer* signifies that the wind tunnel can mimic the wind energy present in the first kilometre of the atmosphere measured from the earth's surface (Roberts, 2013). Figure 1.5 shows a sketch of a boundary layer wind tunnel. Some of the most notable bridges that have been tested in Davenport's wind tunnel are Golden Gate Bridge, Millau Viaduct in France and the proposed Messina Straight Bridge in Italy.



Figure 1.6. Example of a wind tunnel test setup in bridge engineering¹

Figure 1.6 shows an example of a modern wind tunnel setup in bridge engineering. These tests are state-of-the-art methods for evaluating the likelihood of unacceptable aerodynamic loads and response and are recognised by a majority of design codes. However, wind tunnel tests are costly and add to the project programme timeline and are often an iterative process. Furthermore, these tests are always done at model

¹ Source(27/03/2024): <https://twitter.com/RWDIbridges/status/1166330025081262080/photo/1>

scale which may introduce issues with Reynolds number similitude. Because they are done at scale replicating site-specific wind conditions is often an impossible task. Points that go against and in favour of wind tunnel tests are summarised in Figure 1.7.

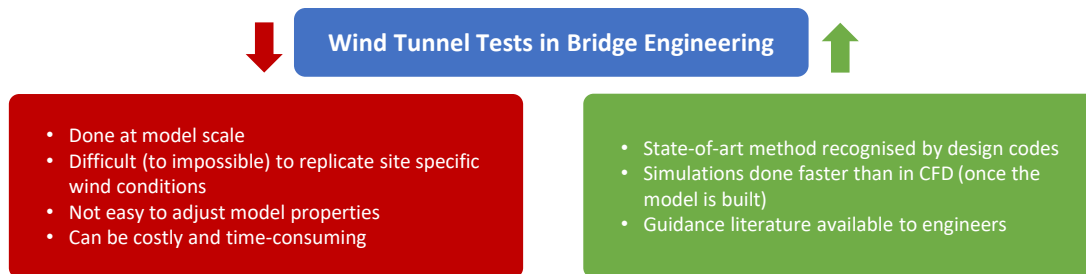


Figure 1.7. Disadvantages and advantages of undertaking wind tunnel tests in bridge engineering

By reducing the time spent in the wind tunnel, engineers can positively impact the overall design delivery, both cost-wise and time-wise. This can be achieved by choosing appropriate *starting points* for the tests and by having a better knowledge of what to expect from the test results. While hand calculations can provide useful information, they often employ equations developed empirically which do not always capture all designs. Computational fluid dynamics (CFD) can provide additional or complementary information to desk-based studies. CFD has been used in research across many applications in the past decades but it has not been present in the bridge industry at the same level. This is because of the complexities revolving around the usage of CFD. Figure 1.8 outlines some of the points that go against and in favour of using CFD in bridge engineering.

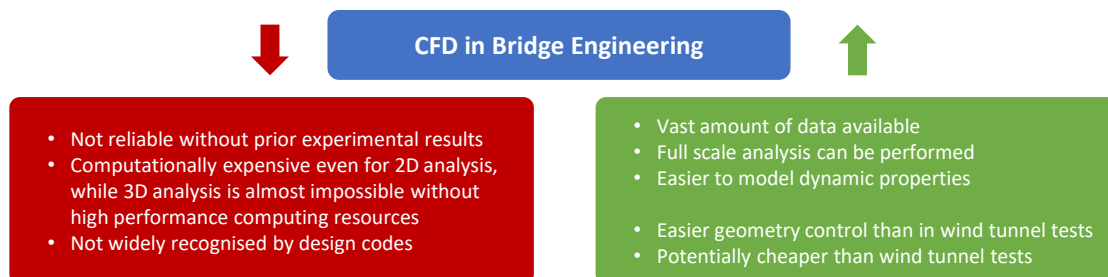


Figure 1.8. Disadvantages and advantages of using CFD in bridge engineering

It is generally accepted that the CFD results are unreliable without prior experimental results of a similar system. This is due to simplifications present in CFD analyses needed so that they can be performed with reasonable computational effort. Even though they are simplified, CFD models still require much larger computational and storage resources than what is required for other modelling tasks in bridge engineering, e.g., Finite Element Modelling. Because of these limitations, CFD is not widely recognised by bridge design codes, hence most design endeavours tend to neglect it. However, when properly used, CFD can provide a vast amount of useful data, flow visualisation (see for example Figure 1.9) and enable an easier understanding of the bridge's aerodynamic properties. Furthermore, in some cases due to scaling, it is easier to control the geometry and dynamic properties in CFD than in the wind tunnel tests, and the results may be obtained in a faster manner.

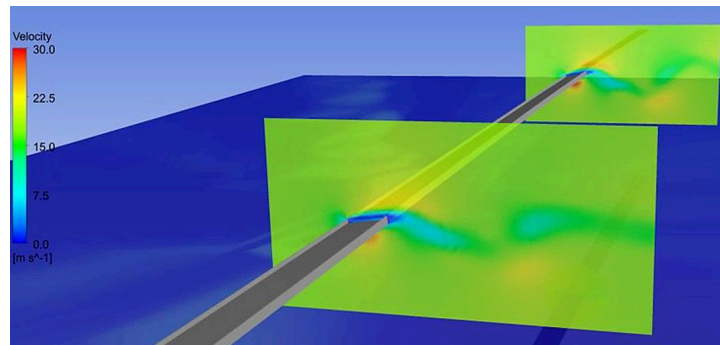


Figure 1.9. Example of the flow visualisation around a bridge deck obtained in CFD¹

1.2 Thesis Objectives

As already noted, the use of CFD in the bridge engineering field has been mainly confined to the research endeavours while the bridge industry saw a limited number of attempts in using CFD as a design tool. Furthermore, as will be discussed in Chapter 2, the CFD in bridge engineering is primarily used for aerodynamically streamlined bridge decks, while bluff decks received little attention.

Although the design standards are continuously being updated, their level of development directly affects the efficiency and accuracy of the design solutions provided by the engineers who use them. The gaps and deficiencies identified in the standards used in Ireland and the UK are discussed in Chapter 2. This research will facilitate further development of these standards.

The first objective of this thesis is to contribute to the CFD representation in the bridge design process with the main focus on bluff bridge decks. This will be achieved by exploring different modelling choices and understanding the peculiarities involved in undertaking these CFD analyses on bluff bridge decks. In order to facilitate the impact on the bridge design industry, the goal is to avoid the usage of high-power computing facilities but rather perform the analyses on a desktop workstation with computing power easily acquired in the bridge industry.

The second objective is to investigate the effects of the deck overhang on the VIV response of bridge decks. This will be achieved by undertaking CFD analyses on a number of deck geometries with varying overhang lengths and developed based on three case study bridges described in Chapter 3. A separate study of the decks with and without the barriers will be presented to outline the effects of the barrier.

1.3 Thesis Structure

Practical industry knowledge guiding the academic capacities towards the problems occurring in everyday engineering is the approach that has fuelled this research from its inception. A synergy between research practices and practical constraints set by the industry is present in every chapter of this thesis. When conducting aerodynamic analyses, the engineers are only interested in the end results of these studies such as static force coefficients, maximum displacement amplitudes and wind velocity at which they occur. The reasons behind a particular behaviour are not of concern. On the other side of the spectrum, in academia, researchers are interested in understanding what causes a certain phenomenon but they often tend to study

¹ Source (27/03/2024): <https://www.mechhead.com/tacoma-bridge-cfd-simulation-7th-year-remembrance/>

cases that are from the practical perspective non-feasible. The work presented here will try to find a middle point between these two sides.

Chapter 2 provides conceptual tools needed for understanding the wider problem of vortex-induced vibrations. It starts with an overview of bridge aerodynamics from the industry's point of view by introducing bridge design standards for aerodynamics. This is followed by an overview of academic literature on the main properties that describe flows around bluff bodies, flow-induced vibrations and more specifically, vortex-induced vibrations of bridge decks. The chapter concludes by introducing computational fluid dynamics and its presence in bridge engineering.

Chapter 3 introduces three case study bridges suggested by the industry partner of this research. The results from their respective wind tunnel testing campaigns are given and discussed with an academic tone but within the practical engineering lane. These case studies are used at various points throughout the research both as a benchmark and as a guide for different research assumptions and decisions.

Chapter 4 summarises these decisions and describes the employed methodologies for each of the research objectives. The theoretical background is outlined with consideration of the practicality and applicability to realistic bridge prototypes.

The substantive results are organised in two chapters which broadly align with the two objectives specified above. Chapter 5 deals with the practicalities around the use of CFD as a tool in the bridge design process with an emphasis on the bluff bridge decks. In Chapter 6, the second objective of the thesis is covered by providing the results of the overhang length effects on the VIV bridge deck response. The implications of the results are discussed through a research lens after which their possible impact on the industry field is outlined.

Chapter 7 concludes the thesis by providing a synthesis of the results and giving an outlook on the possible future work in this field.

Chapter 2

Vortex-Induced Vibration in Context

Chapter 2 provides context for the main topic of this thesis, vortex-induced vibrations of bridges. Firstly, the problem of vortex-induced vibration of bridges is introduced from the design standard perspective. This, along with a short introduction to design standards, what they are, and how they are developed is given in the first part of Section 2.1. After that, a more focused commentary on the code dealing with wind actions in Ireland and the UK and its approach to consideration of the VIV phenomenon for bridges is included. It is then briefly compared to some of the codes used throughout the world. The section concludes with a discussion of what is common to most of these design codes and what they lack.

Section 2.2 gives the basics of fluid mechanics. The main goal of the section is to introduce the vortex-induced vibrations from the theoretical point of view and explain the physics behind it. The section starts with a description of flows around different bluff bodies, and how the flow affects the submerged body and vice versa. Then it introduces different flow-induced vibration phenomena that can occur for general cross-sections. It describes the most important characteristics of vortex induced vibrations and explains how they are different from the divergent aerodynamic behaviour. Circular and rectangular sections are then identified as the most commonly analysed sections in the literature. Finally are defined structural and non-structural elements normally found on a bridge that can affect the flow around it.

Section 2.3 gives a detailed discussion of vortex-induced vibrations of bridge decks. It outlines common issues related to this phenomenon and gives an overview of the research done up to date. Some of the most notable historical occurrences of the vortex-induced vibrations in bridges are presented. The section includes a discussion on some of the parameters affecting the VIV in bridges such as Reynolds number dependency, turbulence intensity in the oncoming flow and the spanwise correlation. A discussion on how different structural and non-structural elements found on a bridge deck can affect the VIV is given at the end of this section, along with the VIV mitigation strategies normally employed by the bridge engineers.

Next, Section 2.4 talks about divergent phenomena with the main purpose of establishing the difference between them and the vortex-induced vibrations.

The process of numerically modelling the flow is introduced in Section 2.5. It provides the theoretical background needed for understanding the issues related to computational fluid dynamics. Then a short background to CFD is given and the most important terms when employing it are introduced. This includes a discussion on mesh generation, boundary conditions, and turbulence models available. The last part of this section gives an overview of the CFD's presence in bridge engineering, clearly showing its limited extent when it comes to studying bluff bridge decks.

The research objectives are defined in Section 2.6. It comprises two topics: the presence of CFD in bridge engineering, and a more specific one which aims to contribute towards design code development.

2.1 Design Codes for Aerodynamics of Bridges

The first known written building code was part of the *Code of Hammurabi* dated circa 1770 BC (Harper, 1904) and since then, world civilisations have been recognising the need for standardisation and regulation in construction. The role of design codes (or standards) is twofold: (i) to give guidance to engineers in their design endeavours, and (ii) to regulate how the design and construction are done. Design standards regulate a broad range of elements involved in engineering, from defining material properties and load actions to prescribing structural analysis and design procedures. In this thesis, the standards dealing with load actions, or more precisely, with wind actions are of interest.

As the industry is modernising (e.g., new materials are being discovered, new construction techniques are being developed, and new loads are being introduced), the design standards must also be constantly updated and developed. Often, the development of the standards is done through dedicated research studies, but also it can be part of big *first-of-its-kind* engineering projects whose design largely falls outside the scope of current codes hence the need for more developed ones.

Normally, countries follow standards tailored to suit the requirements governed by their local geographical and environmental conditions, socioeconomic factors, construction techniques commonly practised, availability of the construction materials and others. However, it is not uncommon for countries to share standards if those requirements have negligible differences (e.g., within the European Union, all countries use the so-called Eurocodes, supplemented by the National Annexes). This thesis refers to the wind effects standards used in Ireland and the UK, with a commentary included in Section 2.1.1, while a summary of several other standards used in countries throughout the world is given in Section 2.1.2.

2.1.1 Design Code in Ireland and the UK

The Design Code applicable in Ireland and the UK, which deals with wind loads on structures is *Eurocode 1: Actions on structures – Part 1-4: General actions – Wind Actions* (EN 1991-1-4) + *National Annex* (NA to IS EN 1991-1-4). Within the *National Annex*, many of the parameters related to the dynamic behaviour of structures under the wind actions refer to *Published Document – Background information to the National Annex to BS EN 1991-1-4 and additional guidance* (PD 6688-1-4, 2015). It should be noted that both standards mentioned here deal with the static and dynamic behaviour of all structures under wind actions. However, the present research is concerned only with the dynamic behaviour of the bridges under wind actions as it is a more complex problem.

The code defines the *aerodynamic susceptibility parameter*, P_b (see Equation 2-1) which informs whether a bridge is susceptible to aerodynamic excitation or not. Depending on the value of P_b , bridges are classified into three groups:

- a) When $P_b < 0.4$, the bridge is **not susceptible** to any form of aerodynamic excitations,
- b) When $0.4 \leq P_b \leq 1.0$, the bridge is **susceptible** to aerodynamic effects but is within the scope of the design code, and,
- c) When $P_b > 1.0$, the bridge is **very susceptible** to aerodynamic excitations and should be subject to wind tunnel tests.

$$P_b = \left(\frac{\rho b^2}{m} \right) \left(\frac{16 v_m^2}{b L n_b^2} \right) \quad 2-1$$

where b, L, m and n_b are deck width, span length, deck mass per unit length and first natural frequency in bending, respectively; ρ and $v_m(z)$ are air density and mean wind velocity, respectively. EN 1991-1-4 defines the *mean wind velocity* as a 10-minute mean wind velocity with a 50-year return period. Bridges that are susceptible to aerodynamic excitations i.e., groups b) and c) are the subject of this thesis. It is worth noting that, at the first look, Equation 2-1 appears counterintuitive because with the increased length (L) the susceptibility parameter decreases. However, with the increased span length, the natural frequency of a bridge (n_b) drops, which gives rise to an increase in the susceptibility parameter.

2.1.1 (a) Vortex shedding velocity

Vortex shedding is a narrowband fluid flow excitation whose frequency depends on the flow speed. The resulting vibration response is self-limiting as the structural motion modifies the excitation force. Although EN 1991-1-4 (Annex E) provides guidelines for assessing the susceptibility of structures to vortex shedding, the National Annex prescribes that PD 6688-1-4 should be used to quantify the VIV response. A brief overview of the procedure for assessing the critical vortex shedding velocity for bridges is given here.

Based on their geometry, bridge decks are divided into nine groups (types) as shown in Figure 2.1. Critical vortex shedding wind velocity for the i^{th} natural mode of oscillation is defined in Equation 2-2, and it should be higher than 1.25 times the mean¹ wind velocity at the deck level. The value of 1.25 is introduced to allow for sustained gust speeds however the exact derivation of this value is not clear (TRL, 2002).

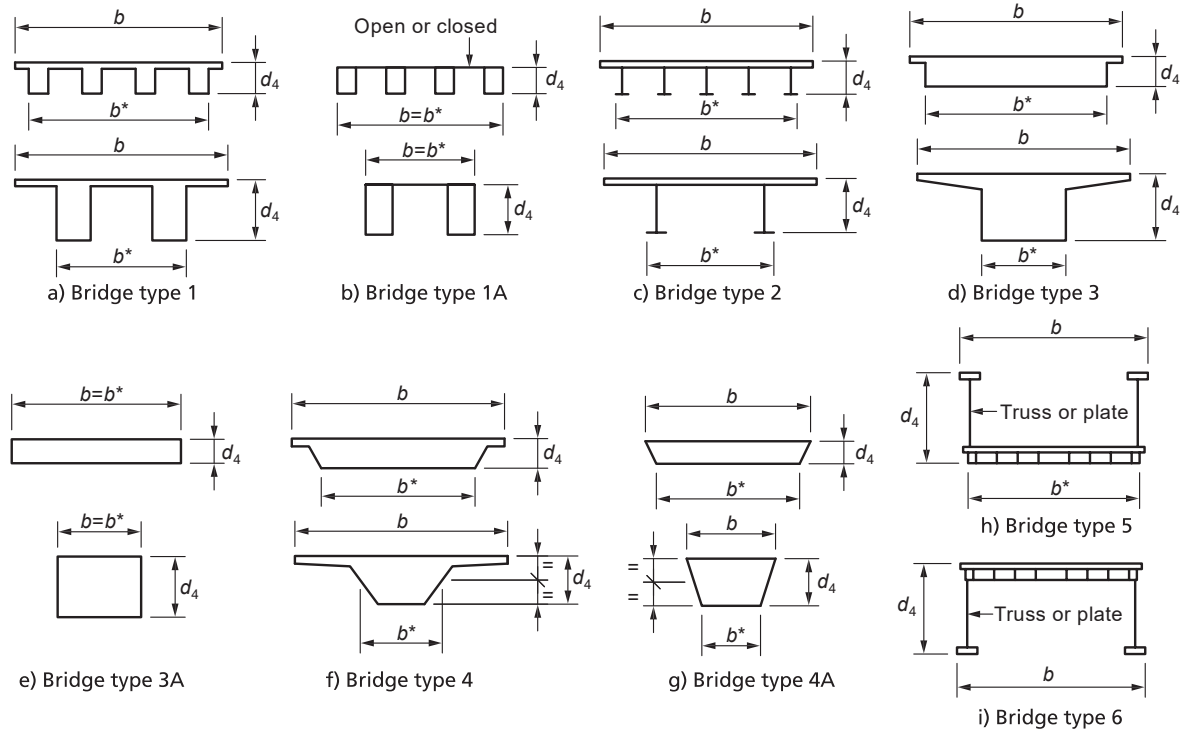


Figure 2.1. Bridge deck types as defined in PD 6688-1-4

¹ See Section 2.2.1 for the definition of *mean wind velocity*.

$$V_{crit,i} = \frac{d_4 n_i}{St} > 1.25v_m \quad 2-2$$

where d_4 is the deck depth as defined in Figure 2.1, n_b is the i^{th} natural frequency of the deck, and St is the Strouhal number obtained from Figure 2.2. The family of curves given for the Strouhal number has been developed from a comprehensive wind tunnel campaign conducted on a broad range of common bridge deck geometries. In essence, the curves envelope the values obtained from these tests. Further details on the tests done and the data behind these curves can be found in Flint & Neil & TRL, (1986).

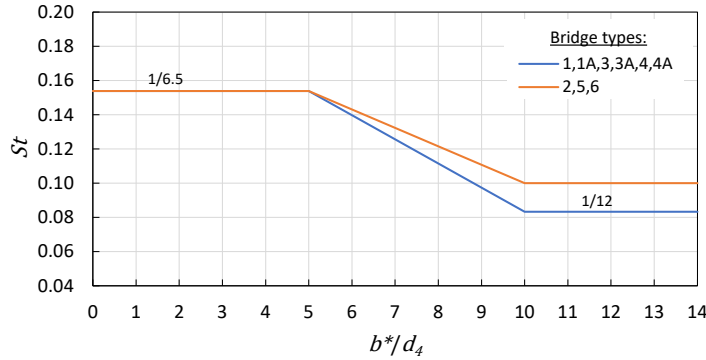


Figure 2.2. Strouhal number for different deck types as defined in PD 6688-1-4; see Figure 2.1 for b^* and d_4 and deck types

2.1.1 (b) Crosswind amplitude due to vortex shedding

For natural modes that do not satisfy the criteria given in Equation 2-2 (i.e., where VIV is likely), PD 6688-1-4 provides approximate formulae to estimate the vibration amplitudes. These formulae are valid for all bridge deck types defined in Figure 2.1, provided that certain conditions¹ on deck geometry and site topography are met. Equation 2-3 and Equation 2-4 give the estimate of the vertical and torsional amplitudes, respectively.

$$y_{max,vertical} = \frac{cb^{0.5}d_4^{2.5}\rho}{4m\delta_s} \quad 2-3$$

$$y_{max,torsion} = \frac{cb^{1.5}d_4^{3.5}\rho}{8mr^2\delta_s} \quad 2-4$$

where c is a factor taking into account the leading edge geometry and is equal to $3(k + h\phi)/d_4$ but not less than 0.5, δ_s is the structural damping (logarithmic decrement), r is the polar radius of gyration, k is the depth of the edge beam, ϕ is the parapet solidity ratio and h is the parapet height (see Figure 2.3).

The research leading to the development of Equation 2-3 and Equation 2-4 concluded that the geometry of the leading edge plays an important role in the maximum amplitude estimation, hence the existence of factor c . The background document to this study (TRL, 2002), justifies the factor of 3 in the equation for c by stating that the amplitudes for bridge decks without an overhang² are larger than those of the decks with an overhang. It does not however indicate what is the minimum overhang length which would allow for the reduction of this factor.

¹ Geometric conditions are related to the fascia beam, barrier, and parapet geometry while the conditions regarding the site topography require that the wind attack angle is not altered by more than 3° due to the terrain topography, see PD 6688-1-4 for more details.

² Part of the deck extending outside the width of the main girders. Overhang length for symmetrical decks is equal to $0.5(b - b^*)$.

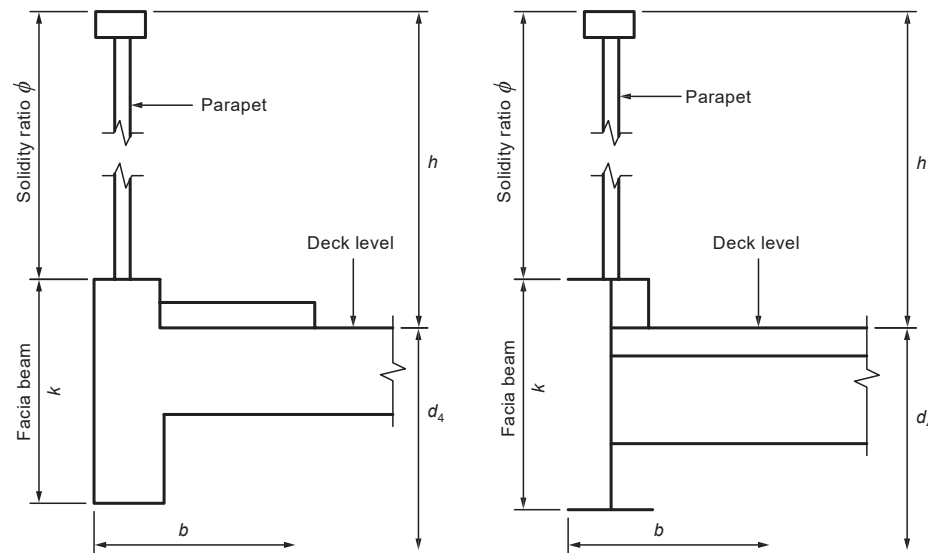


Figure 2.3. Edge detail geometry defined in PD 6688-1-4

2.1.2 Other Design Codes

The bridge design standard used in the USA (AASHTO, 2020), defines bridges as *wind-sensitive* when the span length to depth or width ratio is higher than 30. It is then stated that for wind-sensitive bridges the aeroelastic effects must be taken into consideration and it suggests a number of books and guidelines to be followed, e.g., ASCE, (1961), ASCE, (1992), Scanlan, (1975), along with the wind tunnel testing.

Indian standard for design loads (IS 875-3:2015) gives guidelines on assessing the vortex shedding critical velocity for circular and rectangular cross-sections only, while for bridges it prescribes specialist guidance to be sought (*specialist* implies wind tunnel testing). Similarly, the Australian design code (AS 5100-2:2017) provides guidelines for the static wind load acting on bridges but the dynamic behaviour of *wind-sensitive* bridges is again to be examined by a wind specialist.

One of the most developed standards for the design of long-span bridges for wind-induced vibrations is used in Japan. *Wind Resistant Design Manual for Highway Bridges* or *WRDM* (Japan Road Association, 2007) is developed based on comprehensive research and experience obtained during the design and construction of several long-span bridges built in Japan in the late 1900s and early 2000s, e.g., Seto-Ohashi Bridges (1978-1988), Akashi-Kaikyo Bridge (1998), Kurushima-Kaikyo Bridge (1999) and others. It defines bridges susceptible to wind-induced vibrations based on a number of factors, such as bridge type, material type, deck type, deck depth-to-width ratio, span length, design wind velocity, and local turbulence intensity. The standard also advises on the type of phenomena that may occur (divergent vibration in bending or torsion, or vortex-induced vibration) for a given bridge. It gives a formula to estimate the displacement due to vortex-induced vibrations. It suggests that for the deck sections with overhangs of length larger than a quarter of the deck depth, the estimated displacement is reduced by a factor of two (the background is not explained). It also includes a factor that is a function of the turbulence intensity (with increasing turbulence intensity, the estimated deck displacement decreases). Detailed guidance for the use of WRDM is given in Fujino et al., (2012). It is worth noting that the Japanese standard prescribes the use of wind tunnel testing for the new designs when the wind tunnel data of the existing bridges with similar geometry is insufficient for making an informed design decision.

2.1.3 Opportunities for Further Refinement of Design Codes

As seen, the level of development of the design standards for wind actions varies with each country. However, two similarities can be observed: (i) they always prescribe the use of wind tunnel testing for bridge designs that are susceptible to aerodynamic effects. An exception is the Japanese standard which suggests that wind tunnel testing should take place only when experience from past testing campaigns for similar designs cannot be utilised, and (ii) none include provisions for the use of CFD in the design process (from those considered in this thesis). One can argue that these two points go hand in hand. While wind tunnel testing remains a state-of-the-art procedure for assessing and understanding the complex aerodynamic behaviour of bridges, the use of CFD at an early design stage could provide valuable information which may relax the testing requirements for certain, marginally susceptible, cases.

Other than identifying whether a bridge will experience adverse aerodynamic effects, the more developed standards also provide formulae for the estimation of the magnitude of these effects. The equations evaluating the displacement amplitude that may be caused by the vortex-induced vibrations are given in British PD 6688-1-4 and Japanese WRDM. In both cases, a possible presence of the overhang in the bridge deck shape is acknowledged. As the overhang extends past the deck's full depth, it can play a stabilising role in the vortex shedding phenomena. Although this is recognised in both standards, only the Japanese one estimates smaller displacements due to VIV for cases when a critical threshold of the overhang length is reached. On the other hand, PD 6688-1-4 does not explicitly define the parameter dealing with the overhang but in its background document (TRL, 2002), the difference between the decks with and without an overhang is acknowledged.

Another parameter that experience of practice exposed as important when wind tunnel tests are conducted is the turbulence intensity in the oncoming flow. While turbulence is included in the calculations of the static wind forces, PD 6688-1-4 does not consider it when the aerodynamic response of the bridge is studied. The significance of this parameter is recognised in WRDM by evaluating the bridge's susceptibility to the aerodynamic effects and categorising the phenomena based on the turbulence intensity level.

Two topics identified to be of interest for this research are (i) the lack of presence of CFD in the bridge design process and (ii) consideration of the overhang length in the desk-top calculations for the vortex-induced vibrations of bridges. Furthermore, a discussion on the effect of turbulence intensity on the VIV response of bridges is included and identified as a path worth exploring in future research and standard developments.

2.2 Flow Around Bluff Bodies

2.2.1 General

Fluid flows can be divided into three general groups: laminar, transitional, and turbulent. Laminar flow is characterised by an undisturbed flow in the afterbody¹ region and a large boundary layer region where viscous effects are dominant (e.g., Gerhart et al., 2016). Pathlines of fluid particles in *laminar* flow are smooth and form a layer-like arrangement or laminas. In laminar flow, there is very little mixing and no multi-scale

¹ Part of the flow downstream and in the vicinity of the body.

eddies occur. *Turbulent* flow is characterised by a high level of mixing and the fluid particles moving in a chaotic, random, manner. The boundary layer in a turbulent flow is confined to a narrow region in the proximity of the body and the flow in the afterbody region is highly disturbed. Figure 2.4 shows particle pathlines for one-dimensional laminar and turbulent flows. *Transitional* flow is present when the flow is transitioning from laminar to turbulent.

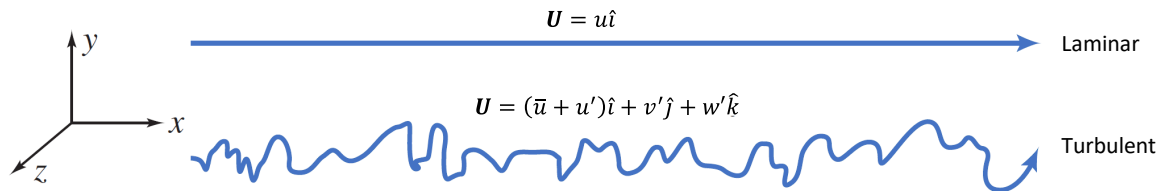


Figure 2.4. Particle pathlines in one-dimensional laminar and turbulent flows (adapted from Pritchard, 2011)

In one-dimensional laminar flow, the velocity vector $\mathbf{U}$ is specified only by u , while in turbulent flow the velocity is defined by the mean velocity $\bar{u}$ and randomly fluctuating velocity u' , v' and w' in three directions. It is the presence of these random, high-frequency fluctuations that make the analysis of turbulent flows difficult. It is important to note that the 3-dimensionality is not what defines a flow turbulent¹ but the randomness and mixing. More details on the flow analysis are given in Section 2.5.1.

Properties affecting the flow type are fluid velocity (U), characteristic dimension of the submerged body (D), density of the fluid (ρ) and dynamic viscosity of the fluid (μ). A dimensional analysis yields a parameter named *Reynolds number* (Re) given in Equation 2-5. It represents a ratio of inertial and viscous forces present in the fluid surrounding the body. The higher the Reynolds number, the more dominant inertial forces are which leads to flow separation and turbulent behaviour.

$$Re = \frac{\rho U D}{\mu} \quad 2-5$$

It is generally accepted that laminar flow occurs when $Re < 100$, while fully turbulent flow develops at $Re > 10^5$. This can depend on many factors, such as the shape of the body and the roughness of the body surface. In practice, for bridges, wind velocities range between 5 and 60 m/s, the bridge depth (see Figure 2.21) is taken as a characteristic length (ranging between 3 and 7 m for long-span bridges) and if air density and viscosity are taken as 1.225 kg/m³ and 1.802 × 10⁻⁵ kg/m-s, respectively, the Reynolds number ranges between 1 × 10⁶ and 30 × 10⁶. Thus, the typical flows around bridge decks are almost always turbulent. It is important to note that much of the early experimental studies on vortex-induced vibrations of basic cross-sections were done in a low $Re (< 1000)$ regime and so are predominantly laminar.

When a rigid body is submerged in a moving fluid, the fluid exerts forces on the body. These forces come as a result of pressure difference and the viscous stresses that are present on the *wet* faces of the body. The resultant forces can be obtained by integrating pressure over the body surface. The force in the direction of the flow is termed *drag*, and the force normal to the flow is termed *lift*. The third load that is of interest is the *pitching moment* acting about the body centroid (see Figure 2.5).

¹ Flow around a sphere at low Reynolds number is 3-dimensional but laminar.

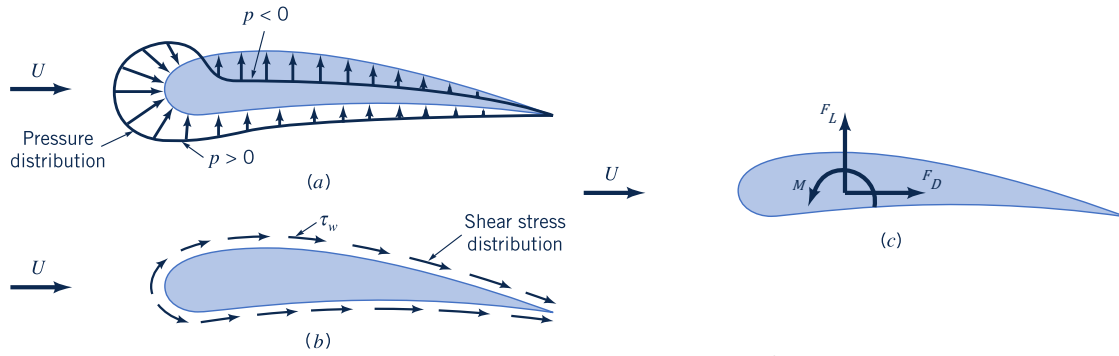


Figure 2.5. Airfoil submerged in a moving fluid: (a) pressure, (b) shear/viscous stresses, (c) drag and lift forces and pitching moment (adapted from Gerhart et al., 2016)

In practice, instead of the aerodynamic forces, their non-dimensionalised counterparts are used. This is done for a more convenient comparison between the cases¹. These are defined in Equation 2-6, Equation 2-7 and Equation 2-8.

$$C_D = \frac{F_D}{0.5\rho U^2 A} \quad 2-6$$

$$C_L = \frac{F_L}{0.5\rho U^2 A} \quad 2-7$$

$$C_M = \frac{M}{0.5\rho U^2 BA} \quad 2-8$$

where C_D , C_L and C_M are drag, lift and moment coefficients, respectively, F_D , F_L and M are the drag and lift forces and moment, respectively, A and B are the projected area and the width, respectively. The dimensions for reduction commonly used in bridge engineering are introduced in Section 2.2.8.

2.2.2 Flow-Induced Vibrations

In practice, the fluid loads in Equation 2-6, Equation 2-7 and Equation 2-8 are time-dependent, leading to the possibility of structural vibration. Figure 2.6 identifies different types of flow-induced vibrations that may occur in a structure submerged in a moving fluid: *Buffeting* – coming from the vibrations caused by the turbulence in the oncoming flow; *Vortex-induced vibrations* – characterised by the self-limited amplitude; *Gallop* and *flutter* – are divergent amplitude oscillations related to aeroelastic instability phenomena.

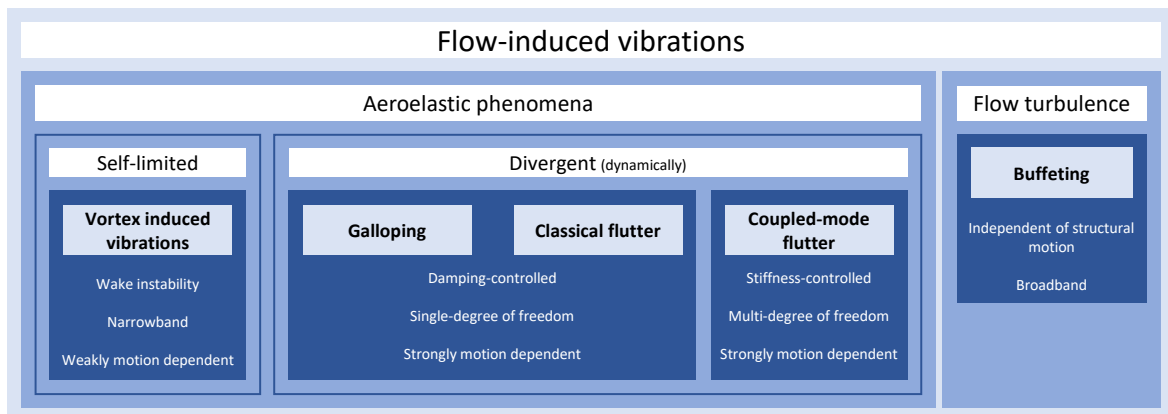


Figure 2.6. Classification of flow-induced vibrations

¹ E.g., forces measured in the wind tunnel experiments can be related to forces exerted on the prototype.

Each of these phenomena is characterised by the magnitude of the oscillations they are causing, the self-limiting or divergent behaviour of the oscillations, and the wind velocity band at which they occur. Some of these features may be identified in the schematic representation of the amplitude against wind velocity shown in Figure 2.7.

Buffeting oscillations are characterised by relatively small oscillations, small enough so that they do not alter the surrounding flow. They occur purely due to fluctuations present in the oncoming flow. Buffeting is independent of structural motion and is broadband.

Vortex-induced vibration occurs due to the formation of the separation bubble on one or two sides of the bluff body and is commonly referred to as a *wake instability phenomenon*. Although the vibrations due to vortex shedding occur for a wide range of wind velocities, large excitation occurs only in a narrow band. They are weakly dependent on the body motion. VIVs, usually, do not cause a collapse of the structure but can considerably shorten its fatigue life and induce undesirable effects for the serviceability limit state (PD 6688-1-4).

Divergent flow-induced vibrations cause oscillations with no-limit amplitudes and, in most cases, cause a collapse of the structure. They typically occur at much higher flow velocities than VIVs (with some exceptions – see Section 2.2.4 for further details). The fluid forces do not exist if the body is stationary.

When studying flow-structure interaction, it is commonly adopted that the flow velocity, frequency, and oscillation amplitude are non-dimensionalised (reduced) using the system's natural oscillation frequency (f_n) and the characteristic length (D). This reduction of variables allows for an easier comparison between different analysis cases.

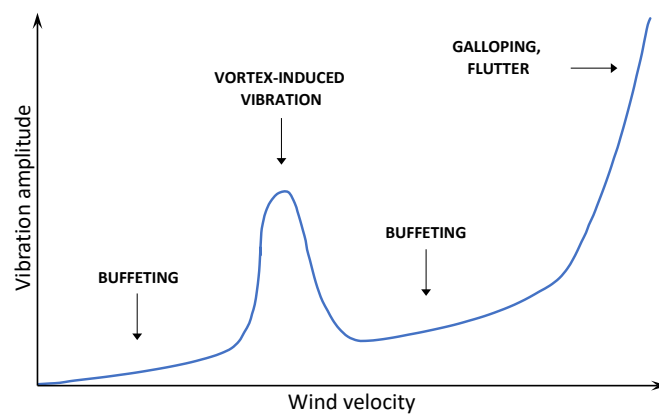


Figure 2.7. Schematic representation of wind-induced vibrations vs wind velocity (adapted from Fujino & Siringoringo, 2013)

2.2.3 Vortex-Induced Vibrations

As previously noted, vortex shedding is a wake instability. At high values of the Reynolds number, when the flow separation occurs, vortices form in the wake region behind the bluff body. These vortices are being detached (shed) from the body at a frequency named the *shedding frequency* (f_{vs}). The cross-section shape and size, or more precisely, leading, and trailing edge shapes along with the width-to-depth (B/D) ratio, play a crucial role in characterising the vortex shedding pattern i.e., cross-wind and along-wind vortex spacing (see Figure 2.8). The ratio of these two values is nominally constant, so with the changing location of the separation points, the cross-wind vortex spacing varies which in turn alters the along-wind spacing which

governs the shedding frequency. Vortex shedding can occur at different scales, from shedding behind the oil risers caused by the sea current to shedding in the wake of islands caused by the atmospheric winds (Figure 2.8). The parameter describing the shedding behind a bluff body is named *Strouhal number* (St), defined in Equation 2-9, where D and U are the characteristic cross-flow dimension of the body and free stream flow velocity, respectively.

$$St = \frac{f_{vs}D}{U} \quad 2-9$$

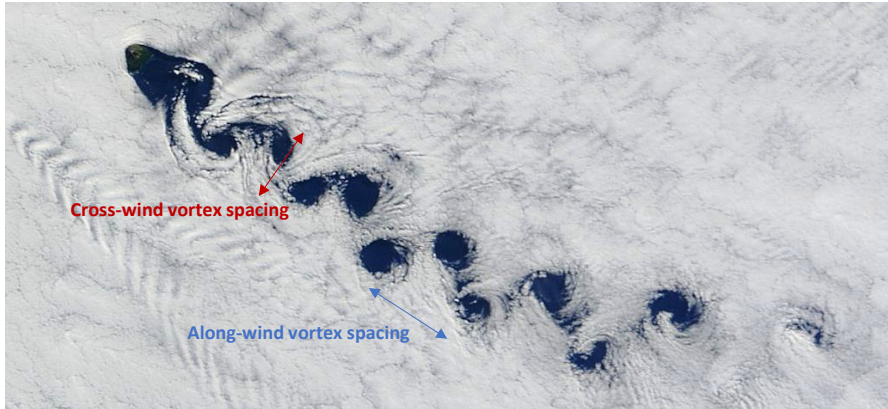


Figure 2.8. Karman vortex street above Tristan da Cunha island caused by the wind (photograph taken from NASA MODIS Rapid Response System, 2010)

As previously noted, the location of the separation points affects the frequency at which the vortex-induced vibrations may occur (by affecting f_{vs}). For bodies with rounded leading edges (e.g., a circular cylinder, or an airfoil), the locations of separation points vary with time and are highly dependent on the Reynolds number. With increased turbulence in the boundary layer, the flow has more kinetic energy which allows it to *travel* further across the body surface before it is separated (see Figure 2.9). A classic example of this behaviour is seen in a golf ball moving through the air. The indents on the surface of the golf ball increase the turbulence in the boundary layer enabling it to experience smaller drag and thus fly further than a smooth-surfaced ball.

Having the location of the separation points vary leads to a dependency of the Strouhal number on the Reynolds number. For bodies with sharp edges, it is generally accepted that for high enough values of the Reynolds number $O(> 10^4)$, the Strouhal number can be considered constant. This is because the separation points are well-defined by the sharp corners of the body.

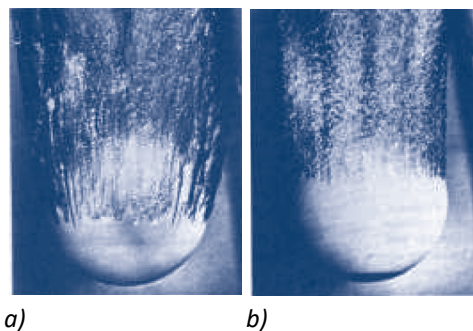


Figure 2.9. Effects of turbulence in the boundary layer on the flow separation location; a) smooth ball and b) ball with rough surface (Gerhart et al., 2016)

As the vortices are being shed from the bluff body, they exert a fluctuating load on it. With varying flow velocity, the shedding frequency varies (see Equation 2-9) and, at a certain point, it may match the first natural frequency of the body. At the coincidence, the shedding frequency matches the oscillation frequency and stays constant for a range of flow velocities. This phenomenon is called *lock-in* of the shedding frequency. Within the lock-in region, large amplitude vortex-induced vibrations may occur. Figure 2.10 shows an example of this phenomenon for a circular cylinder. As previously noted, the VIV exists outside the lock-in region and this is clearly seen in Figure 2.10 where a non-zero VIV response is noticeable pre-coincidence and after the lock-out has occurred.

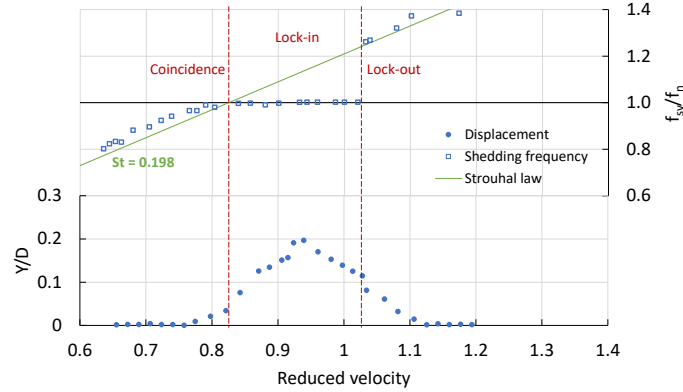


Figure 2.10. Vortex induced vibrations of a circular free-to-oscillate cylinder (adapted from Feng, 1968); Y – displacement amplitude; D – section diameter; f_{vs} – shedding frequency; f_n – natural frequency

The width of the lock-in region depends on the mass-damping ratio of the oscillating system. The parameter often used for describing the mass-damping properties of a system is named *Scruton number* (Sc) and is defined in Equation 2-10. Figure 2.11 shows the dependency of the lock-in region width on the mass-damping parameter of a circular free-to-oscillate cylinder on a linear elastic mount.

$$Sc = \frac{2\delta_s m}{\rho D^2} \quad 2-10$$

where δ_s and m are the structural damping (logarithmic decrement) and the mass, ρ is the air density and D is the characteristic dimension. Scruton number is a *twice-reduced* number, i.e., the logarithmic increment of the damping is a non-dimensional value and $\frac{m}{\rho D^2}$ is the reduced mass of the body, or the mass ratio. It is interesting to note that the mass ratio is also found in Equation 2-1 describing the aerodynamic susceptibility of a bridge.

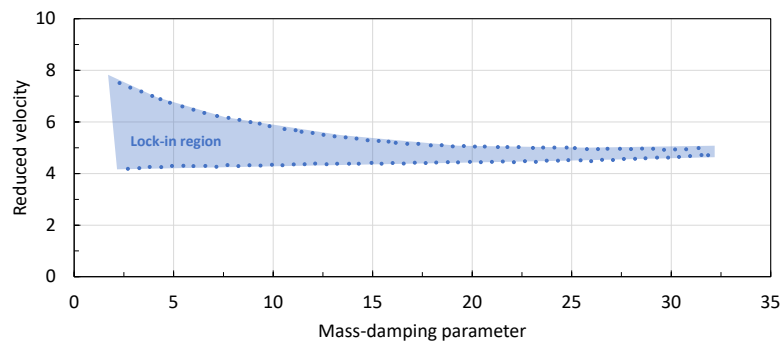


Figure 2.11. Lock-in range width for different mass-damping parameters for a circular cross-section (adapted from Chen, 1985)

Both, the leading and trailing edges of the bluff body affect the shedding process. This was examined in detail by Shiraishi & Matsumoto, (1983). It was concluded that the bodies with the leading edges shaped so that the flow separation is delayed away from the leading edge perform better under VIV. The contrary was found for the shape of the trailing edge where more rectangular ones provided a more stable response. They ranked different types of edge shapes from unstable to stable as shown in Figure 2.12.

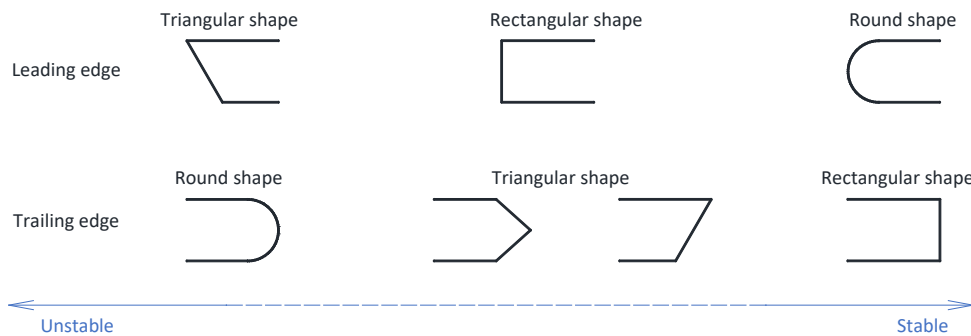


Figure 2.12. Ranking of the edge shapes based on their aerodynamic performance as defined by Shiraishi & Matsumoto, (1983)

Since vortex-induced vibration is a non-linear problem (the flow and the body motion affect each other), in their endeavours to study the phenomenon, researchers tried to simplify it. In principle, three types of models can be employed when studying the VIV:

- a) **Fixed model** – the simplest model of the three, consists of a stationary body submerged in a moving fluid. It is important to note that the results obtained differ from the results for the oscillating body because of the non-linearity present in the relationship between the body motion and the flow field. Nonetheless, the model can be used as a starting point for obtaining an estimate of the Strouhal number.
- b) **Forced oscillation model** – the body is forced to oscillate at a set frequency with a set amplitude. This is a slightly more complex model but without the need for modelling the dynamic properties of the structural system (mass, stiffness, and damping). The flow field around is forced to adjust to the motion of the body. Useful information on the fluid force and shedding frequency can be obtained if the motion amplitude and flow velocity are varied. The maximum amplitude can be obtained by combining it with a model of the structure.
- c) **Elastically supported model** – also referred to as **Fluid-structure interaction (FSI) model**. The most complex model of all three since all dynamic properties of the system are set. The flow velocity is varied and the information on the maximum VIV amplitude and the lock-in region are directly measured. For each flow velocity, the body motion and fluid flow adjust until an equilibrium state has been reached. The maximum amplitude obtained is only valid for the specific value of the Scruton number.

It is well known that the turbulence in the oncoming flow affects the vortex shedding. Matsumoto et al., (1993) studied vortex-induced vibrations caused by vortex shedding (Karman vortex) and motion-induced vortices (MIV) and their susceptibility to turbulence in the upstream flow. Motion-induced vortices are generated by, a relatively large, motion of the body; they should not be confused with vortices

generated by the vortex shedding which is a wake instability phenomenon and is independent of the body motion. Matsumoto et al., (1993) conclude that turbulence disrupts the shedding process thus reducing the oscillations that are governed by this type of excitation. However, they also show that Karman shedding reduces the strength of MIV. This means that for the oscillations primarily governed by the MIV, the introduction of turbulence in the oncoming flow can act adversely on the amplitude of the oscillations. Further details on the effects of the turbulence in the flow are given in the following sections, while bridge-specific issues are covered in Section 2.3.4.

2.2.4 Divergent Amplitude Response

The focus of this thesis is on vortex-induced vibrations, but as galloping and flutter are phenomena that should be part of all bridge aerodynamics work, a summary is given here and additional details, specifically related to bridges, are presented in Section 2.4.

Unlike VIV, galloping and flutter are phenomena with potential non-limiting¹ amplitude feedback and thus play an important role in the bridge design process. While VIV affects the serviceability limit state, galloping and flutter are in the domain of the ultimate limit state. The most famous example of how destructive they can be is that of Tacoma Narrows Bridge collapse in 1940.

Galloping is a single degree of freedom, velocity-dependent, damping-controlled instability occurring primarily in bending (translation) mode. It occurs when the negative damping from the fluid forces cancels out the structural damping present in the structure. Commonly, it occurs at velocities higher than the vortex shedding velocities, which prevents them from interacting. A less common type occurs before the vortex shedding velocity and is caused by the *flapping* of the detached shear layer. It is often manifested as motion-induced vortices (Païdoussis et al., 2010). Only high-velocity galloping will be discussed here. Equation 2-11 gives the necessary criterion for galloping to occur, where C_L and C_D are lift and drag coefficients, respectively and α is the angle of attack (Den Hartog, 1957). This criterion is derived from the equation of motion for an elastically supported body in a moving fluid with the condition that instability occurs when the net damping becomes negative. To use Equation 2-11, the quasi-steady fluid dynamics assumption must be imposed which for the case of slow² structural motion is valid. By quasi-steady fluid dynamics, it is assumed that the fluid force on the structure is defined by instantaneous relative velocity (between the motion and flow velocity). This allows for the fluid forces to be measured independently at different angles of attack for a stationary model in the wind tunnel tests (Blevins, 2001).

$$\frac{\partial C_L}{\partial \alpha} + C_D < 0 \quad 2-11$$

Classical flutter is also a 1-dof instability, occurring in torsion, however, the quasi-steady assumption cannot be imposed (Equation 2-11 is not applicable). The fluid forces depend on the angular velocity while the angle of attack varies along the length of the cross section. The last point makes the

¹ In certain cases, limit cycle oscillation can occur but is achieved by the structural non-linearity.

² *Slow* when compared to the flow velocity. There is no clear consensus in the literature on how slow the motion should be; however, it is generally accepted that when $U/f_n D > 10 - 30$, the structural motion can be considered as slow, and quasi-steady assumption valid. U , f_n and D are flow velocity, natural frequency, and cross wind body dimension, respectively (Païdoussis et al., 2010).

quasi-steady assumption invalid (Païdoussis et al., 2010). The analysis is further complicated by the fact that the phase of the fluid force (relative to the motion) changes with the flow velocity. The susceptibility of a cross-section to classical flutter can be estimated in the wind tunnel tests on an elastically supported model free to oscillate in torsional mode.

Other than 1-dof phenomena, a multi-degree divergence response can also occur. This instability is called *coupled-mode flutter* and it occurs in combined bending and torsion modes. Coupled-mode flutter is a stiffness-controlled phenomenon. Same as for the classical flutter, quasi-steady fluid dynamics assumption cannot be imposed, and a cross-section in the wind tunnel is tested as an elastically supported system with 2-dofs. Further details and a general formulation of the divergent response of bridge decks are presented in Section 2.4.

2.2.5 Circular Sections

Due to its simplicity and applicability to a great number of engineering problems (chimneys, bridge cable stays, oil risers, tubes in nuclear steam generators and others), the circular cross-section is one of the most analysed cross-sections in both wind tunnel testing and CFD analyses. Chen, (1985), and more recently Zdravkovich, (1997), provide a comprehensive overview of the research done on a single, and a series of circular cylinders in still and moving fluid (axially and crossflow).

As previously noted, due to the movement of the separation points, the Strouhal number is highly sensitive to the Reynolds number. Figure 2.13 shows this relation for a cylinder submerged in a low-turbulent flow.

For a cylinder with a circular cross-section, galloping/flutter cannot occur due to its axisymmetry. In this case, the resultant fluid force does not change magnitude and is always in the direction of the flow. However, if a circular cylinder loses its shape, it becomes prone to divergent amplitude behaviour. A common case in bridge engineering when this can occur is when ice is formed on the cable-stays, making them of a non-circular cross-section.

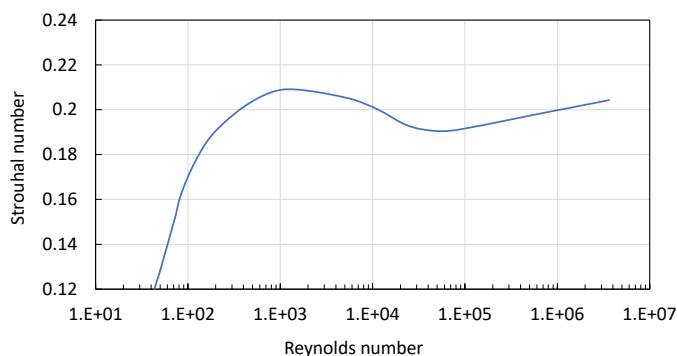


Figure 2.13. Dependence of the Strouhal number on the Reynolds number for a circular cross-section in a low-turbulent flow (adapted from Païdoussis et al., 2010)

As noted in Section 2.2.3, vortex-induced vibrations are often studied on prescribed motion models. Staubli, (1983) used the data gathered from an experiment done on a circular cylinder forced to oscillate in a moving flow at $Re = 2 \cdot 10^4 - 3 \cdot 10^5$. They were able to obtain amplitude responses that were

partially in good agreement with free-to-oscillate model results. The maximum amplitude was overestimated but it was observed at an isolated “island” which was questioned whether it was real or not.

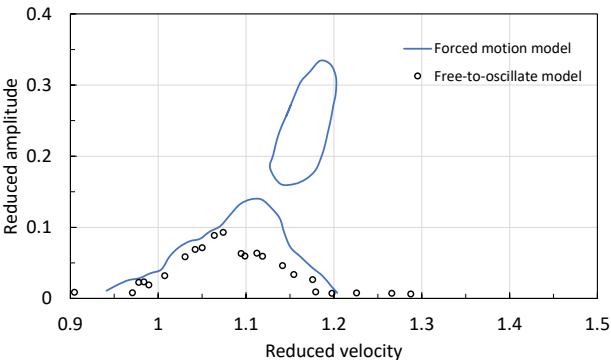


Figure 2.14. Results for a circular cross-section forced to oscillate in a moving fluid compared to free-to-oscillate model (adapted from Staubli, 1983)

Other similar analyses on forced motion circular cylinders were carried out by Akhter & Maysa, (2019), Jiao & Wu, (2018) and Placzek et al., (2009) but at low Reynolds number regime ($Re \approx 100$) thus are not discussed here.

When it comes to the effects of the turbulence in the flow, a series of tests were undertaken on a circular cross-section and results were reported by Goswami et al., (1993). They found that the added turbulence intensity did not have any effect on the elastically supported circular cylinder.

2.2.6 Rectangular Sections

The second most studied cross-section type is the rectangular section which in many cases can be used for studies of wakes caused by bridge decks. Flows around rectangular sections are characterised by well-defined separation points, at four corners. Unlike for circular sections, for certain shape sizes, reattachment of the boundary layers can occur. This phenomenon is primarily driven by the B/D ratio, where B is the width and D the depth of the rectangle. As defined by Deniz & Staubli, (1997), several types of vortex shedding can be distinguished based on the structure of vortices that are being shed. The type depends on B/D ratio and the angle of attack. For a small angle of attack ($< 6^\circ$), three types of vortex-shedding may occur; vortices being shed from the leading edge (LEVS), impingement of leading edge vortices (ILEV) and vortices being shed from the trailing edge (TEVS). An additional type, occurring when the angle of attack is relatively big ($> 6^\circ$), is named alternate-edge vortex shedding (AEVS), that is, the vortices are being shed alternately between the leading and trailing edge. This is summarised in Figure 2.15.

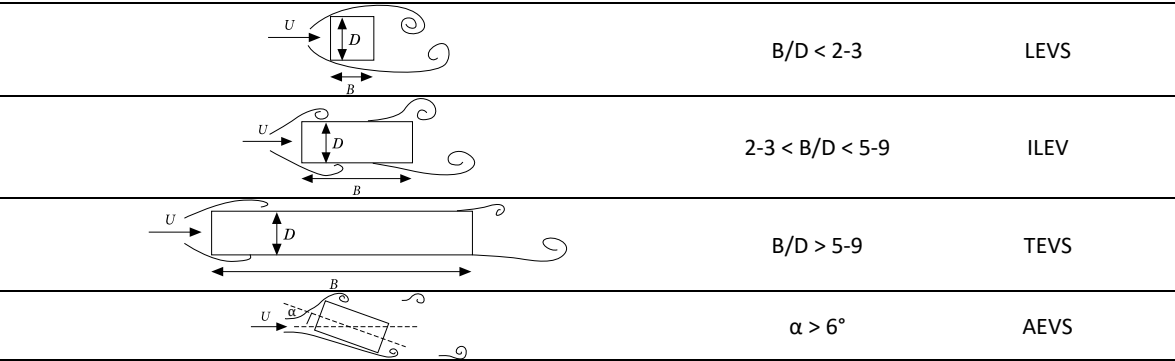


Figure 2.15. Different classes of vortex shedding formation (adapted from Deniz & Staubli, 1997)

The B/D ratio also affects the Strouhal number. Nakamura et al., (1991) conducted tests on a range of plates with different width-to-depth ratios. They found that the St varies with changing B/D in a stepwise behaviour (Figure 2.16). They found that at each step within the increasing B/D ratio, the dominant frequency of the velocity fluctuations measured in the wake gradually becomes less dominant. At the jump between two *steps*, two dominant frequencies were noticeable. These did not occur simultaneously but rather periodically, for different time periods observed. Finally, as the B/D ratio is further increased, the higher frequency becomes dominant while the lower one disappears from the spectrum. It is worth noting that the Strouhal number values for each *step* are approximately integral multiples of 0.6 ($n = 1$ to 4). Further analysis concluded that for a given B/D ratio, the number of vortices that are forming on the body's surface corresponds to n .

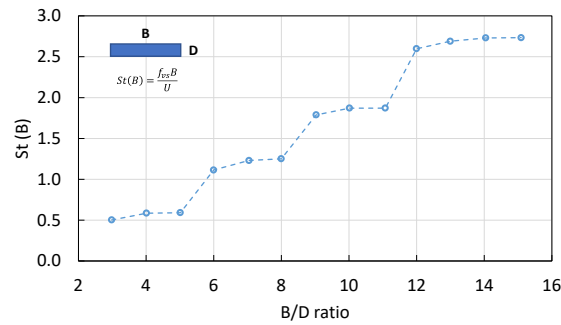


Figure 2.16. Strouhal number vs B/D ratio (adapted from Nakamura et al., 1991)

Nguyen et al., (2018) investigate the vortex-induced vibrations of a rectangular cylinder with B/D ratio equal to 5. In their study, they found that VIV originated from the vortices shed at the leading edge, while the increase in structural response was due to correlated flow features near the trailing edge.

As previously noted, due to well-defined separation points (at the corners), the Strouhal number for rectangular sections is less prone to variation with the changing Reynolds number. Schewe, (2013) conducted a study on a 1:5 rectangular section for a number of angles of attack. The results for the zero angle of incidence are shown in Figure 2.17. For an easier comparison with the results for a circular section, shown in Figure 2.13, the y-axes in both figures are of comparable range.

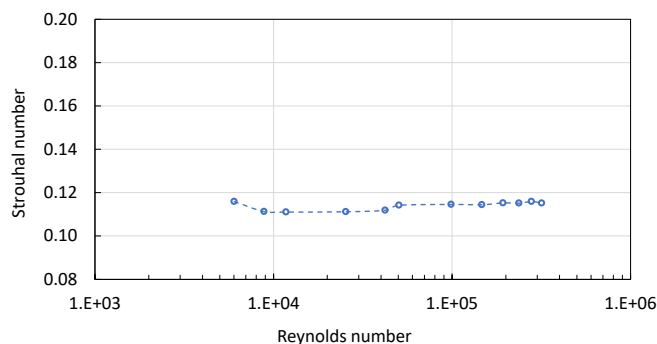


Figure 2.17. Strouhal number vs Reynolds number for a rectangular section (adapted from Schewe, 2013)

Another important feature that influences the aerodynamic behaviour of bluff bodies is the spanwise correlation of the vortex-induced forces. Highly elongated bodies cannot be subject to wind tunnel

testing in their full length and must be scaled down. If the test model is not of sufficient length¹, the results obtained may be valid only in the case of a fully correlated flow. This imposed assumption of 2D full correlation may render the test results invalid if the spanwise correlation length is not properly taken into consideration. Spanwise correlation is normally studied using forced oscillation models (see Section 2.2.2) which allows one to evaluate the spanwise correlation as a function of the vibration amplitude, vibration frequency or flow velocity. A common way of quantifying the spanwise correlation of flow variables e.g., vortex-induced forces, between two spanwise points, is by calculating the *spanwise correlation ratio* using Equation 2-12.

$$\rho_{ij}(z/D) = \frac{E \langle F_i(t) F_j(t) \rangle}{\sqrt{E \left(F_i^2(t) \right) E \left(F_j^2(t) \right)}} \quad 2-12$$

where i and j are two points for which the correlation is being evaluated, D and z are model depth and the spanwise distance between the points, respectively, $E \langle \rangle$ is the mathematical expectation and $F_n(t)$ is the variable for which the spanwise correlation ratio is being calculated at the time t (e.g., vortex-induced force). The angle of attack plays a role in the spanwise correlation of the vortex-induced forces. M. Li et al., (2020) found that for a rectangular section, the spanwise correlation decreases with increasing angle of attack. Sun et al., (2019) studied the spanwise correlation of vortex induced forces for a rectangular, trapezoidal, and streamlined section. The analyses were done for different reduced velocities and oscillation amplitudes. They found that with an increasing oscillation amplitude, the spanwise correlation decreases which tends to decrease the excitation force response. For a free-to-oscillate model, the spanwise correlation at the maximum oscillation amplitude is smaller than when the amplitude is still increasing with the flow velocity (i.e., rising stage of VIV). Figure 2.18 shows the correlation ratio for a 1:4 rectangular cross-section for locked-in and locked-out regimes.

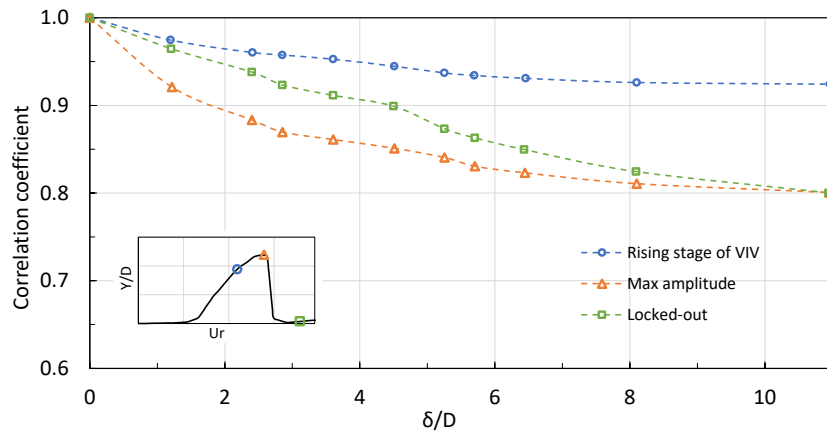


Figure 2.18. Correlation coefficient for three flow velocities; δ – correlation length; D – rectangle depth; Y – oscillation amplitude; U_r – reduced velocity (adapted from Sun et al., 2019)

Vickery, (1966) studied the behaviour of the fluctuating lift force on the square section under smooth and turbulent flows. They found that with the increased turbulence intensity in the oncoming flow, the lift is reduced by about 50%, while the spanwise correlation length is reduced from $5.6D$ to $3.3D$ (D being

¹ Length bigger than the phase of the spanwise deformation for the studied mode of oscillation.

the cylinder diameter). In the same study, a reduction in base pressure was observed for low angles of incidence, while for angles $>25^\circ$ the influence of turbulence was minimal. The author attributes this to the re-attachment of the separated flow, which does not occur for high angles of attack. The turbulence did not affect the value of the Strouhal number in the spectrum of the fluctuating lift. It was also noted that the amount of energy present in the 2% narrow bandwidth around Strouhal frequency decreased between 15% and 20% depending on the angle of attack. Similar findings are reported by Namiranian, (1982) with an additional statement that the turbulence length scale does not influence the fluctuating lift. The effects on heaving and torsional VIV of a rectangular cylinder (side ratio equal to 4) were similar to that found by Daniels et al., (2016). They studied the effect of turbulence intensity and the turbulent length scale. It was reported that increased turbulence intensity diminishes the oscillation amplitudes. However, for a stationary cylinder, the turbulence intensity increased the peak loading. With an increased turbulent length scale (to the same order of magnitude as the cylinder width), the vibration amplitude also increased, although it did not reach the value of the amplitude in the smooth flow. The increased length scale also enhanced the spanwise correlation.

2.2.7 Splitter Plate Effect

The geometric and dynamic properties of the body affect its response when subjected to vortex-induced vibrations. Other than the body itself, what happens in the wake also affects the VIV response. Matsumoto et al., (1993) studied the effect a splitter plate placed in the wake of a 1:4 rectangular cylinder has on its VIV amplitude. They found that the splitter plate eliminates Karman vortex shedding. Similar behaviour was reported by Sahu et al., (2019) who tested a circular section with a splitter plate of 3.5D in length. They found that the vibration amplitude dramatically decreased when compared to the bare cylinder.

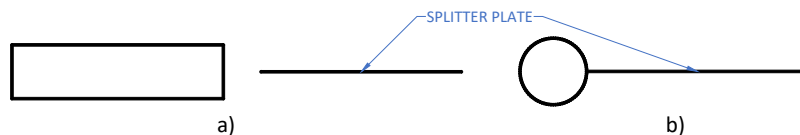


Figure 2.19. Sections equipped with splitter plates tested by a) Matsumoto et al., (1993), and b) Sahu et al., (2019)

Vortex shedding is a wake instability phenomenon meaning that the perturbations originating in the wake cause the shedding process to occur. By placing a plate in the region of the wake where these initial perturbations happen, a symmetry between top and bottom vortices is maintained which removes the likelihood of instability to occur. Figure 2.20 shows the time history of the vortex shedding process being developed behind a circular cross-section. It is seen that: initially, two shear layers that are formed at the top and bottom of the cylinder are symmetric and no oscillation in the lift coefficient is reported; then at approximately $t = 2.7s$, perturbations in the wake start to occur which is manifested as destroyed symmetry in the flow and oscillating lift coefficient; initially small instability begins to grow leading to a steady oscillatory vortex shedding process which is clearly visible at $t = 7.0s$. If a splitter plate is placed in the wake where initial perturbations are to occur, the whole vortex shedding process would be avoided. It should be noted however, that while the splitter plate neutralises Karman vortex shedding, motion-induced vortices are caused by the motion of the body and their effect can be increased by adding a splitter plate. Matsumoto et

al., (1993) showed that the splitter plate doubled the effect of MIV. At the end of Section 2.2.3, it was noted that the Karman shedding process attenuates MIV, hence by eliminating it the effect of the MIV is increased.

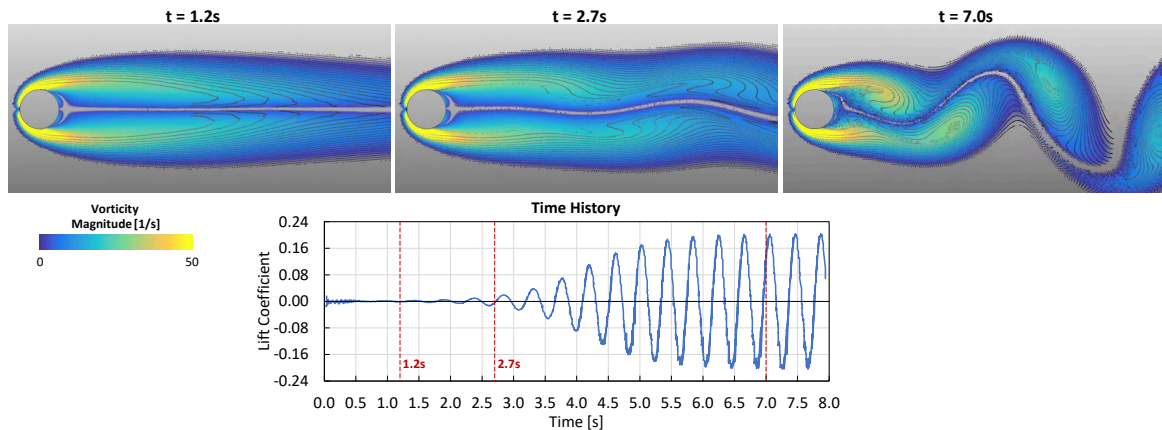


Figure 2.20. Vortex shedding process developing behind a circular cylinder (adapted from THINK Fluid Dynamix, 2017)

For a splitter plate to have an effect, its length can be relatively small (when compared to the size of the bluff body), however, the exact location of the initial perturbations, depends on the Reynolds number. So, a long splitter plate can neutralise vortex shedding for a range of flow velocities. It is expected that an effect similar to the one described in this section can be achieved with long overhangs sometimes found on bridge decks.

2.2.8 Bridge Deck Sections

The importance of correctly understanding the flow field in the vicinity of the deck has been justified numerous times throughout the history of bridge engineering (further details are given in Section 2.3.2). The flow field is characterised by the presence of complex flow structures that may be generated by the deck shape, the 3D features present on the deck (barriers, wind shielding, cable anchorages, road curbs, terrain, and other bridge furniture), micro-climate of the surrounding and other structural elements such as pylon(s) and abutments. Understanding of the flow field around the bridge decks is further complicated by the broad range of deck types, shapes and sizes that can exist (see Figure 2.1 and Section 2.5.3). Because of this, wind tunnel tests are still considered the state-of-the-art approach for assessing bridges for their aerodynamic behaviour.

A typical bridge deck section comprises structural elements (deck girders, deck slab, cable stays) and non-structural elements (wind shielding, barriers, light poles, and others). Both groups equally affect the aerodynamic profile of the bridge. Figure 2.21 shows a typical bridge deck section (Narrow Water Bridge deck section taken as an example) along with the characteristic dimensions such as deck width and depth.

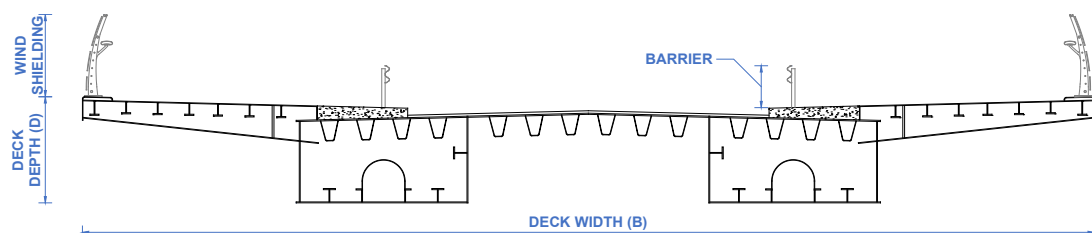


Figure 2.21. Typical bridge deck cross-section (Narrow Water Bridge deck)

Bridge aerodynamic analysis can provide a broad range of results important for the design process. It can comprise results on static force coefficients, susceptibility to VIV and divergent excitation, wind profile downstream of the deck (important for structures downstream of the bridge), wind shielding performance (important for the assessment of the users' comfort) and many more. In bridge engineering, a common practice for reducing the force coefficients, defined in Equation 2-6, 2-7 and 2-8, is to use bridge depth (D), bridge width (B) and spanwise length¹ (L), yielding a set of relations given in Equation 2-13.

$$C_D = \frac{F_D}{0.5\rho U^2 DL} \quad C_L = \frac{F_L}{0.5\rho U^2 BL} \quad C_M = \frac{M}{0.5\rho U^2 B^2 L} \quad 2-13$$

The wind response of a bridge is also affected by the surrounding terrain (see Figure 2.22). It can govern the level of turbulence and the angle of attack of the oncoming flow (see Section 2.3.4 for turbulence effect on VIV). Y. Li et al., (2017) studied the wind characteristics of a bridge site located in a deep-cutting gorge. They developed a model at the scale of 1:1000 (15 m in diameter) and conducted the tests in a wind tunnel. They conclude that due to the terrain configuration, the angles of attack considered when analysing bridge aerodynamics should be in the range of $\pm 6^\circ$. As seen in Section 2.2.6, the angle of attack influences the type of vortex shedding that occurs (see Figure 2.15).

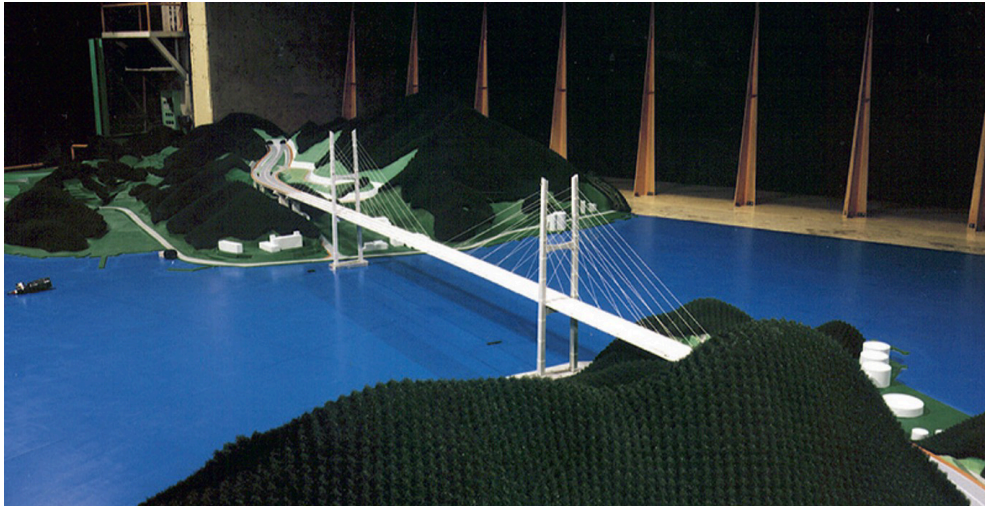


Figure 2.22. Full aeroelastic model and surrounding terrain for the Megami Bridge (Fujino et al., 2012)

Y. Zhang et al., (2023) conducted a CFD study on a full-scale 3D model of Rose Fitzgerald Kennedy Bridge in Ireland which included the surrounding terrain. To a satisfactory degree, they managed to replicate the readings from the anemometers installed in different locations on the bridge. They conclude that when analysing the wind characteristics of the surrounding terrain, the inclusion of the bridge geometry plays an important role. The proximity of the deck to the ground/water level below the deck also affects the bridge's aerodynamics. Yin et al., (2022) studied the effect waves have on aerodynamic force coefficients of a bridge deck above the water level. They found that the time histories of the aerodynamic forces change periodically, and the change is consistent with the wave periods. Zhou et al., (2019) analysed the flutter stability of a closed box girder when ground effects below the deck are introduced (see Section 2.4 for galloping and flutter of bridge deck sections). Models with clearance to deck width ratios ranging from 0.6 to 0.2 were analysed in

¹ In practice, the aerodynamic loads are defined per meter of spanwise deck length.

the wind tunnel tests and numerically. They found that flutter critical speed decreased with the reduced clearance level.

Other than increasing the blockage ratio¹, barriers and wind shielding also affect the flow field by introducing flow structures that are of significantly smaller scale than the ones produced by the bare deck. This introduces complexities when the flow around a bridge deck is analysed, numerically and in the wind tunnel experiments. More details on how the CFD analyses are affected by the introduction of barriers are given in Section 2.5.3.

2.3 Vortex-Induced Vibrations of Bridge Decks

2.3.1 Background

Although vortex-induced vibrations, in most cases, do not cause the collapse of the structure, they affect the bridge's fatigue life and the Serviceability Limit State (see e.g., EN 1991-1-4). Large amplitude vibrations can be managed with tuned mass dampers (TMD), as in the case of the bridge over the river Volga in Russia (Benicke & Butz, 2015); however, a more common approach is to eliminate the aerodynamic susceptibility at the design stage. This is done by investigating the aerodynamic profile of the bridge as a whole, but also its elements individually (e.g., deck, pylon, cables, wind shielding). Although many design codes and standards (e.g., EN 1991-1-4; PD 6688-1-4) give guidelines for the consideration of the aerodynamic effects on bridges, for long-span bridges, conducting wind tunnel tests is still integral to the design process.

2.3.2 Historic Cases

In 1980, the Rio Niteroi bridge in Brazil experienced VIV with excessive vertical deck motion. A comprehensive wind tunnel study determined that it was caused by VIV for wind velocities of approximately 14 m/s. The bridge remained closed until it was fitted with passive and active dynamic control devices (Battista & Pfeil, 2000). In 2010 a bridge over the river Volga, Russia experienced high amplitude vertical vibrations (approx. ± 400 mm) which forced authorities to close the bridge. To suppress future vibrations, tuned mass dampers were designed and installed (Benicke & Butz, 2015). In the late 90s during the construction of the Great Belt East Bridge in Denmark, low-frequency vertical oscillations were noticed (Larsen et al., 2000). Even though the vibrations were regarded as non-threatening from the structural point of view, they posed a considerable negative impact on user experience. The vibrations were mitigated by installing the guide vanes² at the corners of the deck. H. Li et al., (2014) investigated the behaviour of the Xihoumen Bridge in China by analysing the extensive monitoring data acquired from the bridge. They found that in the period between 2009 and 2014, there were 37 VIV occurrences with duration ranging from 30 minutes to 3.5 hours. They compared the full-scale data to the data acquired from the 1:25 scale wind tunnel model and concluded that, although the model predicts VIV, the amplitude of the oscillations and the lock-in region were smaller than those observed on the full-scale bridge. The discrepancy was attributed to the difference in the Reynolds number.

¹ Ratio of the frontal area of the model to the cross-sectional area of the wind tunnel.

² For guide vanes, see Figure 2-28 in Section 2.4.6.

2.3.3 Reynolds Number Dependency

As already noted, the value of the Strouhal number may depend on the Reynolds number at which it is measured. Although the dependency is more evident for the round cross-sections (see Figure 2.5), bridge deck sections are not immune to it either.

Larose & D'Auteuil, (2006) stressed the importance of respecting the Reynolds number similitude when investigating aerodynamic effects at model-scale for bluff bodies with sharp edges. The similitude cannot always be achieved, especially when the full-scale bridges are several tens or even hundreds of meters in size. The importance of the Reynolds number similitude was shown on several bridges in the past. They were subject to full-scale monitoring and the results compared to wind-tunnel tests with much smaller Reynolds numbers. Larose & D'Auteuil, (2006) also state that when studying the effect of the Reynolds number, turbulence in the oncoming flow is an important factor. Schewe & Larsen, (1998) examined the reason for the discrepancy in recorded and predicted wind velocity at which vortex shedding occurred for the Great Belt East Bridge in Denmark. The bridge consists of the main suspension span and two approach spans with the deck shapes less streamlined than that of the main span. The discrepancy was present only for the approach spans. They concluded that the main culprit is the difference in Reynolds number between the full-scale and model-scale decks and that a minimum Reynolds number of 500,000 (based on the deck width) should be respected in model-scale experiments. Another study on the Reynolds number sensitivity on aerodynamic effects was undertaken by Matsuda et al., (2001). It was done on twin box decks and it showed that the Reynolds number should be at least 10^6 for the results from the model-scale testing to be comparable to the full-scale effects.

2.3.4 Turbulence Effect

To the best of the author's knowledge, there have been a limited number of studies done towards a better understanding of the effect turbulence has on the VIV response of the bridge decks.

Matsumoto et al., (1993) investigated the effects of turbulence intensity on the VIV of the hexagonal box girder shown in Figure 2.23. They conducted tests for two levels of structural damping ($Sc = 12$ and $Sc = 60$). They found that, for the lower Sc number, the increased turbulence intensity ($I_u = 4.8\%$) did not affect the VIV amplitude but it increased the onset velocity by 18%. In the case of a higher Sc number, the VIV amplitudes increased by approximately a factor of 3.5 with no change in the onset velocity. They also report that the solidity ratio¹ of the barriers plays a significant role in the response due to VIV. For the solidity ratios above 70%, the introduction of turbulence in the oncoming flow decreases the VIV amplitude, while the opposite holds for the lower solidity ratios.

Ralph et al., (2016) have shown that the VIV amplitude of a twin box girder decreased linearly with increased turbulence intensity in the oncoming flow. They also show that the increase in the turbulent length scale did not affect the VIV response of the model.

¹ Effective area normal to the wind direction divided by the outline area of the barrier.

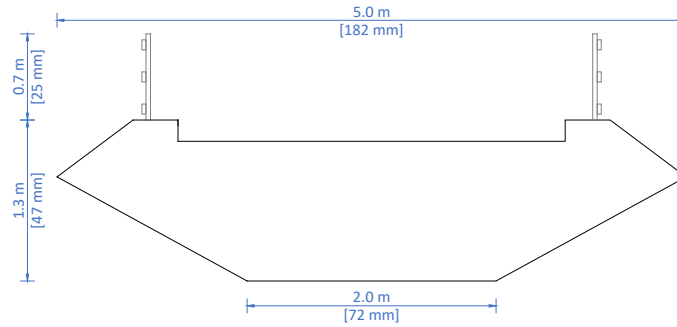


Figure 2.23. Hexagonal box section tested by Matsumoto et al., (1993); dimensions in full scale and in [model scale]

The effect of turbulence length scale was considered in the study done by Kawatani et al., (1993). They conducted wind tunnel tests on several bluff body shapes common in bridge engineering (see Figure 2.24). In their tests, they controlled turbulence intensity and turbulent length scale. They conclude that for all sections, the increased turbulence intensity decreases the vibration amplitude except for section AF1 which experienced an increase in the VIV amplitude for all three levels of turbulence intensity considered (6%, 12% and 24%). When it comes to the varying length scale size, similar to Ralph et al., (2016), they have found no effect on the VIV amplitude for any of the sections considered.

More cases of the effects caused by increased turbulence in the flow are given in Appendix A.

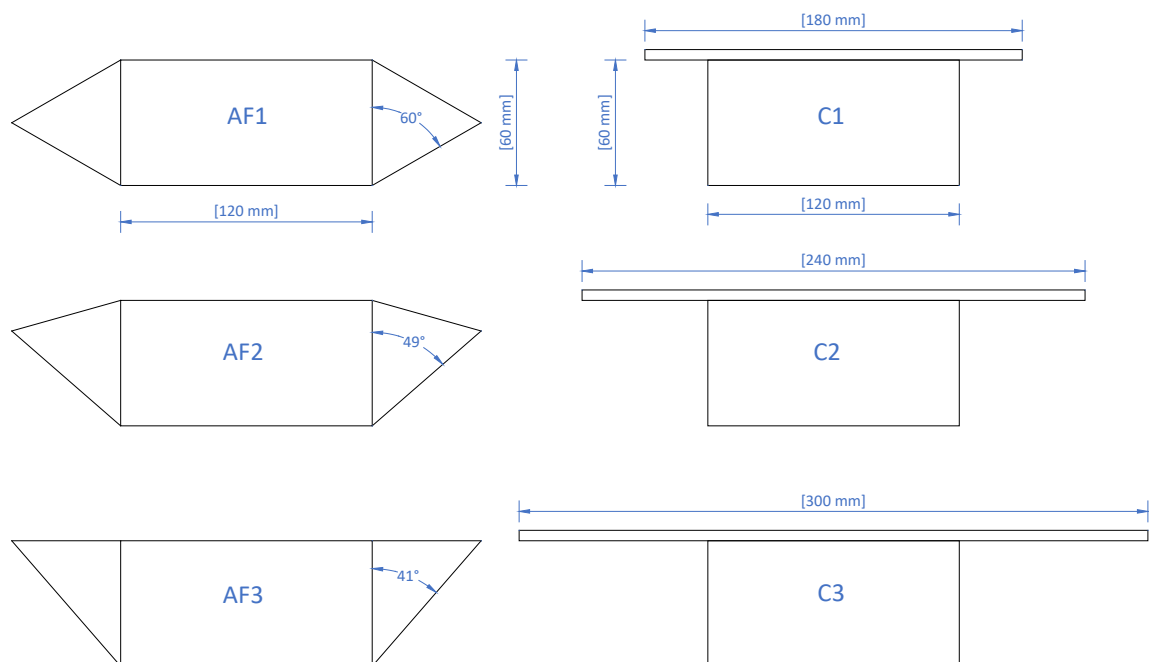


Figure 2.24. Some of the models [at model scale] investigated by Kawatani et al., (1993)

2.3.5 Spanwise Correlation

As noted in Section 2.2.6 for rectangular sections, the spanwise correlation of vortex-induced forces plays an important role in understanding the VIV phenomena. Reducing the model to two dimensions inherently assumes a full correlation of vortex-induced forces. This may be a sufficiently accurate assumption for a relatively low mode shape with the phase of spanwise deformation such that the change in sign occurs at intervals longer than the VIV correlation length. However, for higher natural frequencies, the assumption of full spanwise correlation may be wrong and lead to overestimated aerodynamic response.

M. Li et al., (2020) experimentally investigated the spanwise correlation of the vortex-induced forces on cylinders with 4:1 rectangular, trapezoidal, and streamlined bridge cross-sections. They studied the vortex-induced pressure distribution and the VIV amplitude for different angles of attack to identify the correlations of the total vortex-induced forces (VIFs) for stationary and oscillating cylinders. They found that, for trapezoidal cross-section, the spanwise correlation increases with increasing vibration amplitude. This is the opposite of what was recorded for a rectangular cross-section. For the streamlined bridge girder section, the correlation was high for all investigated vibration amplitudes and can be regarded as fully correlated. For the same cross-section, the spanwise correlation for a stationary cylinder was much smaller than for the oscillating one.

The same study showed that the variation of the angle of attack has a different effect on spanwise correlation for the trapezoidal and streamlined cross-sections. Although in both cases the correlation decreases with increasing angle of attack, the VIV for the streamlined cross-section can be considered as fully correlated even for the highest angle of attack (see Figure 2.25).

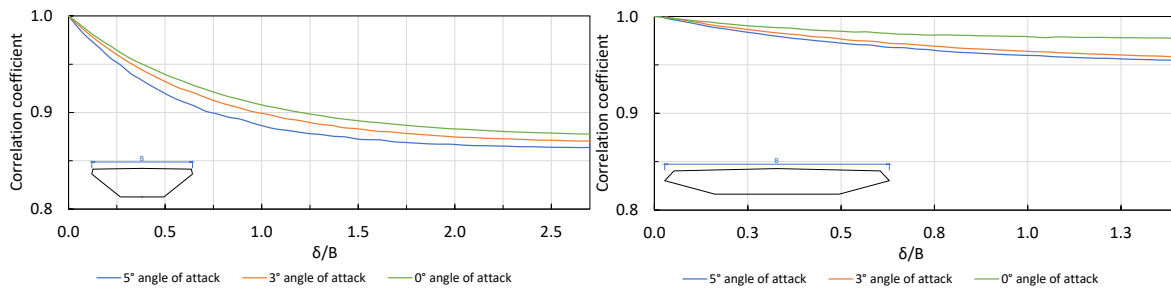


Figure 2.25. Spanwise correlation of VIF for the varying angle of attack: a) trapezoidal cross-section, b) streamlined bridge girder (adapted from M. Li et al., 2020); δ – correlation length; B – section width

Another thorough experimental campaign on spanwise correlation of VIFs was done by Zhu et al., (2017). They investigated a twin-box bridge deck section, which is considered quite susceptible to VIV. It is shown that correlation is the highest (correlation ratio close to 1.0) for the wind velocity corresponding to the biggest VIV amplitude. Again, the opposite was true for a rectangular section (see Figure 2.18).

2.3.6 Influence of Structural and Non-Structural Elements and Mitigation of VIV in Bridges

The aerodynamic behaviour of a bridge deck is mainly governed by its shape and size. However, the size and shape of elements such as cable-stay anchors, fairing plates, handrails, light poles, and other deck furniture, can affect the flow field around the deck. Matsumoto et al., (1993) studied the aerodynamic behaviour of a hexagonal box girder. They showed that the solidity ratio of the barriers at the side of the bridge affects the oscillation amplitude in the VIV response. The study also suggests that depending on the solidity ratio, the oscillation amplitude may increase or decrease when turbulence is introduced to the oncoming flow. Adding the handrails and the median divider increased the oscillation amplitudes dramatically for a streamlined bridge deck section studied by Larsen & Wall, (2012).

Fairings at the leading and trailing edges of bridge decks have been used extensively as a way of mitigating the issues related to the VIV. These plates are used to achieve more streamlined bridge deck sections and are considered as non-structural elements. The angle and size of these plates affect the VIV response of the bridge deck. Y. Le Li et al., (2018) showed that the smaller the fairing angle (see Figure 2.26),

the smaller the VIV response is expected. They also state that the wind velocity at the maximum displacement amplitude is also affected, however a clear conclusion could not be made.

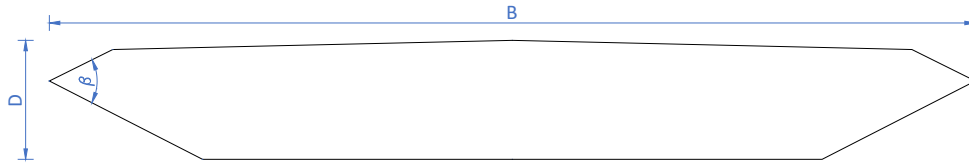


Figure 2.26. Trapezoidal bridge deck showing the fairing angle (β)

Having elements protruding from the bridge deck (see e.g., Figure 2.27) can affect the spanwise correlation of the vortex induced vibrations which in turn affects the oscillation amplitude. H. Zhang et al., (2019) showed that by adding 3D vortex generators on the bridge deck surface, vortex shedding in the wake of the deck can be reduced or even completely suppressed. A similar effect, although of a smaller magnitude, can be expected from the light poles placed on top of the deck.



Figure 2.27. Cable anchorages protruding the deck of the William Dargan Bridge, Dublin, Ireland

In cases when the original design fails to mitigate the issue related to the VIV, countermeasures must be developed and deployed to the bridge structure. In the late 1990s once it experienced the vortex-induced vibrations, the Great Belt Bridge was fitted with guide vanes at the bottom side of the deck (Larsen et al., 2000). Guide vanes were made of bent plates and installed with an offset of 60mm from the deck plates. In addition to mitigating VIV problems, the guide vanes increased the critical wind velocity for flutter instability by roughly 6%.

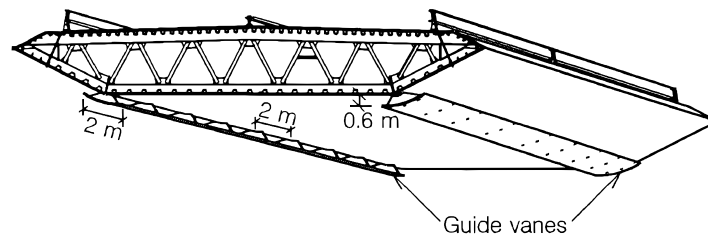


Figure 2.28. Guide vanes installed on the Great Belt Bridge (Larsen et al., 2000)

A more expensive solution was developed for the Volgograd Bridge in Russia after its VIV occurrence in 2010 (Benicke & Butz, 2015). Adaptive Tuned Mass Dampers were installed on the bridge which is an *intelligent* version of the more common TMD system. Based on the frequency of the excitation, adaptive TMD constantly adjusts damping (approximately every 0.01 second in the case of Volgograd Bridge).

Y. Ge et al., (2022) studied the vortex induced vibrations that occurred on the Humen Pearl River Bridge in China in May 2020. The excitations occurred after the temporary water-filled barriers during routine bridge maintenance were placed at the deck edges (see Figure 2.29). The barriers altered the aerodynamic

profile of the bridge which led to vortex induced vibrations. The study showed that by adding the flow spoilers (Figure 2.30) at the deck edges, the issue can be prevented from further occurring.

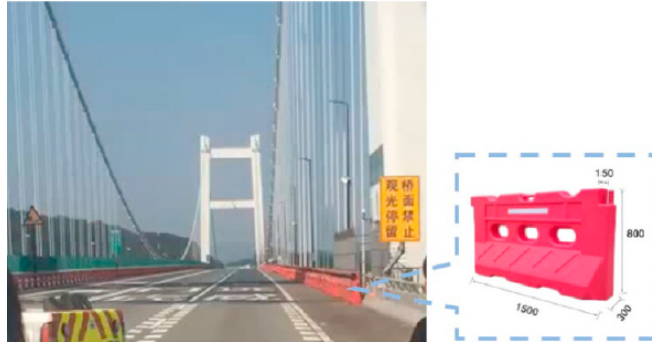


Figure 2.29. Temporary water-filled barriers placed on the Humen Pearl River Bridge (Y. Ge et al., 2022)

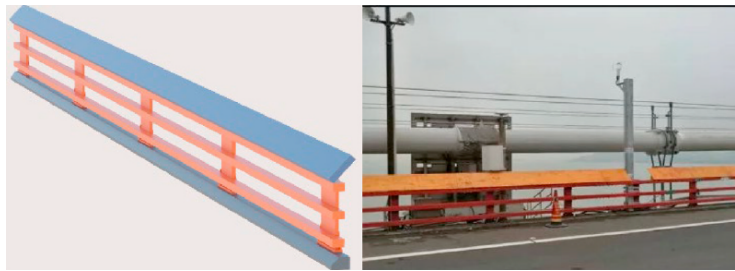


Figure 2.30. Flow spoilers installed on the Humen Pearl River Bridge (Y. Ge et al., 2022)

Larsen & Wall, (2012) studied the flow around different trapezoidal bridge deck sections and concluded that the best-performing shapes are the ones with an angle of 15 degrees between the bottom inclined face and the horizontal one. However, they state that this is only valid for the cases when open barriers are installed.

2.4 Galloping and Flutter of Bridge Decks

Section 2.2.4 describes different divergent response behaviours and outlines the differences between them. Here, a more general formulation of these phenomena as they relate to bridges is presented.

For a symmetric bridge deck cross-section with its elastic and mass centres coinciding (as is the case in the majority of the long-span bridges), the equations of motion for three degrees of freedom per meter spanwise length are defined as:-

$$m\ddot{y} + c_y\dot{y} + k_y y = F_L \quad 2-14$$

$$I\ddot{\alpha} + c_\alpha\dot{\alpha} + k_\alpha \alpha = M \quad 2-15$$

$$m\ddot{p} + c_p\dot{p} + k_p p = F_D \quad 2-16$$

where y , p and α are vertical and in-plane deck displacements and pitching rotation, m and I are mass and moment inertia, c and k are damping and stiffness for each degree of freedom; F_L , F_D , and M are lift and drag forces and pitching moment per meter of the deck. Derivatives with respect to time are denoted with a dot and a double dot. In the case of, relatively, in-plane-stiff bridges, Equation 2-16 may be neglected however, as found by Miyata, (2003) for the case of Akashi Kaikyo Bridge, flexible long-span bridges may undergo significant in-plane displacements giving importance to this term. Theodorsen, (1935) gave

theoretical solutions for L_f and M for the case of oscillating airfoils but, as pointed out by Paidoussis et al., (2010), this solution is not entirely valid for bridge decks because, unlike for airfoils, the flow separation occurs at the leading edge. Nevertheless, empirical expressions for, F_L , M , and F_D were developed (Equation 2-19 through 2-17). For details on their derivation see e.g., Simiu & Yeo, (2019).

$$F_L = \frac{1}{2}\rho U^2 B \left[KH_1^* \frac{\dot{y}}{U} + KH_2^* \frac{B\dot{\alpha}}{U} + K^2 H_3^* \alpha + K^2 H_4^* \frac{y}{B} + KH_5^* \frac{\dot{p}}{U} + KH_6^* \frac{p}{B} \right] \quad 2-17$$

$$M = \frac{1}{2}\rho U^2 B^2 \left[KA_1^* \frac{\dot{y}}{U} + KA_2^* \frac{B\dot{\alpha}}{U} + K^2 A_3^* \alpha + K^2 A_4^* \frac{y}{B} + KA_5^* \frac{\dot{p}}{U} + KA_6^* \frac{p}{B} \right] \quad 2-18$$

$$F_D = \frac{1}{2}\rho U^2 B \left[KP_1^* \frac{\dot{p}}{U} + KP_2^* \frac{B\dot{\alpha}}{U} + K^2 P_3^* \alpha + K^2 P_4^* \frac{p}{B} + KP_5^* \frac{\dot{y}}{U} + KP_6^* \frac{y}{B} \right] \quad 2-19$$

where B is the deck width; $1/2\rho U^2$ is the dynamic pressure; K is the non-dimensional oscillation frequency defined as $2\pi fB/U$, with f being the oscillation frequency; terms H_{1-6}^* , A_{1-6}^* and P_{1-6}^* are called *flutter derivatives* and they are functions of K . Again, for bridges with sufficient stiffness in the in-plane direction, Equation 2-19 and all terms dependent on p can be neglected (leaving A_{1-4}^* and H_{1-4}^*). Flutter derivatives provide information on the dominant type of divergent behaviour and the critical velocity at which it may occur. Terms $A_{1,2}^*$ and $H_{1,2}^*$ are the velocity-dependent damping terms and provide information on single-dof divergent behaviour while the other four are displacement-dependent stiffness terms and conversely provide information on coupled-mode flutter¹. For classical flutter to occur, term A_2^* must be positive, while for galloping, a requirement is the positive sign of the term H_1^* . Flutter derivatives strongly depend on the cross-section geometry and are identified in wind tunnel experiments. In recent years, CFD has been also used to identify flutter derivatives, see e.g., Brusiani et al., (2013); Ge & Xiang, (2008); Nieto et al., (2015). More on the application of CFD in bridge engineering is given in Section 2.5 below.

2.5 CFD in Bridge Aerodynamics

In this section, a summary of the fluid dynamics is given, including a description of Navier-Stokes equations, the complexities involved in solving them and the alternatives the researchers must resort to. The difficulties that come with CFD and the different options available for modelling fluid flows are also covered. Finally, a discussion on the presence of CFD in bridge engineering is included with some of the case studies found in the literature.

2.5.1 Basics of Fluid Dynamics

The derivation of the equations governing the fluid flow has been covered in numerous books and papers (see e.g., Pritchard & Mitchell, 2015). A summary, required for the context, is included here.

There are two methods for studying fluid dynamics problems. *Lagrangian method*, which is characterised by *following* the fluid particles as they move, and describing how the flow properties associated with these particles change in time. The second, more commonly used, is the *Eulerian method*. It describes the flow by defining the flow field variables as a function of space and time. Information about the flow is

¹ Galloping and classical flutter are damping-controlled instabilities, while coupled-mode flutter is stiffness-controlled phenomena. See Section 2.3.4.

obtained for fixed points in space and the properties are studied as the fluid particles flow through those points (Gerhart et al., 2016). This approach of describing properties is named *field representation* of the flow. One of the most important fluid variables is the *velocity field*, comprised of velocity vectors (Equation 2-20).

$$\mathbf{U} = u(x, y, z, t)\hat{\mathbf{i}} + v(x, y, z, t)\hat{\mathbf{j}} + w(x, y, z, t)\hat{\mathbf{k}} \quad 2-20$$

where u , v and w are the x , y and z components of the velocity vector. Flows for which the velocity, for a given location, is constant in time are called *steady flows*. The opposite holds for *unsteady flows* which can be subdivided into non-periodic flows, periodic flows, and truly random flows.

The *acceleration field* is derived as shown in Equation 2-21.

$$\mathbf{a} = \frac{\partial \mathbf{U}}{\partial t} + u \frac{\partial \mathbf{U}}{\partial x} + v \frac{\partial \mathbf{U}}{\partial y} + w \frac{\partial \mathbf{U}}{\partial z} \quad 2-21$$

where the first term is the *local derivative* and the other three are the *convective derivative*. Together, they form the operator named the *material derivative*, denoted as $\frac{D(\cdot)}{Dt} = \frac{\partial(\cdot)}{\partial t} + \mathbf{U} \cdot \nabla(\cdot)$.

The law of mass conservation states that *for any closed system, the mass of the system cannot change in time*. For a flow system, it is called the *continuity equation* and is expressed by Equation 2-22.

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad 2-22$$

The fluids with constant density are called *incompressible*. Air can be considered incompressible provided that the flow velocity is smaller than 100 m/s (Pritchard & Mitchell, 2015). Since ρ cannot change, for incompressible flows, the continuity equation (Equation 2-22) is simplified to Equation 2-23 (i.e., the velocity field is divergence-free).

$$\nabla \cdot \mathbf{U} = \frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad 2-23$$

Starting from Newton's second law, equations of motion for fluid can be derived, these are more commonly known as *Cauchy equations*. In convective form, these equations are given in Equation 2-24.

$$\frac{D\mathbf{V}}{Dt} = \frac{1}{\rho} \nabla \cdot \boldsymbol{\sigma} + \mathbf{g} \quad 2-24$$

where $\frac{D\mathbf{V}}{Dt}$ is the material derivative of the velocity vector field (acceleration field), ρ is the fluid density, $\nabla \cdot \boldsymbol{\sigma}$ is the divergence of the stress tensor and $\mathbf{g}$ is the acceleration vector caused by the body forces.

Fluids whose viscosity is not affected by shear rate are called *Newtonian fluids* (e.g., air and water). So, for incompressible, Newtonian fluids, the stresses are linearly related to the rate of deformation and can be expressed as in Equation 2-25 and Equation 2-26.

$$\sigma_{xx} = -p + 2\mu \frac{\partial u}{\partial x} \quad \sigma_{yy} = -p + 2\mu \frac{\partial v}{\partial y} \quad \sigma_{zz} = -p + 2\mu \frac{\partial w}{\partial z} \quad 2-25$$

$$\tau_{xy} = \tau_{yx} = \mu \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right) \quad \tau_{yz} = \tau_{zy} = \mu \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right) \quad \tau_{zx} = \tau_{xz} = \mu \left(\frac{\partial w}{\partial x} + \frac{\partial u}{\partial z} \right) \quad 2-26$$

where p is the pressure given by $p = -\frac{(\sigma_{xx} + \sigma_{yy} + \sigma_{zz})}{3}$ and μ is the *dynamic viscosity*. These equations can be combined with Cauchy equations and developed in three directions as shown below.

$$\text{x-direction} \quad \rho \left(\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} \right) = -\frac{\partial p}{\partial x} + \rho g_x + \mu \left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2} \right) \quad 2-27$$

$$\text{y-direction} \quad \rho \left(\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} \right) = -\frac{\partial p}{\partial y} + \rho g_y + \mu \left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2} \right) \quad 2-28$$

$$\text{z-direction} \quad \rho \left(\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} \right) = -\frac{\partial p}{\partial z} + \rho g_z + \mu \left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2} \right) \quad 2-29$$

The resulting set of equations (Equation 2-27 through 2-29) is called *Navier-Stokes equations*, and when combined with the continuity equation (Equation 2-23), form a complete mathematical description of the flow of incompressible Newtonian fluids (Gerhart et al., 2016). These four equations have four unknowns (u , v , w and p) and with a set of boundary conditions and defined initial conditions form a *well-posed* mathematical problem. However, being nonlinear, second-order, partial differential equations (PDEs), a closed-form solution is available only for a handful of cases.

2.5.2 Background to CFD modelling

As seen in the previous section, a flow field is described by a system of nonlinear second-order PDEs. Because of their complexity, the analytical solution can be obtained for a limited number of highly simplified, laminar flow cases. To overcome this problem, researchers and engineers must resort to experimental fluid mechanics when studying fluid flows. However, when the examined models must be scaled down (due to e.g., experimental equipment size), dynamic similitude becomes an important factor when evaluating the validity of the test results. To fully respect the similarity laws, the similitude of all non-dimensional parameters describing the system (e.g., Re , Sc , Ma^1) must be followed. However, this is not practical and, in some cases, not possible. For example, if the Reynolds number similitude is to be respected, the flow velocity in the wind tunnel would have to be increased (for example >100 m/s) which may cause issues with incompressibility assumption (Mach number similitude would be compromised). The issues related to the scaling in the wind tunnel test were the topic of many research studies. Stathopoulos & Surry, (1983) found that the local pressure coefficients for low buildings can be underestimated by up to 30% when tested in the wind tunnels. Larsen et al., (2008) tested the effectiveness of guide vanes as VIV mitigating devices mounted on a twin girder bridge deck. They concluded that the tests must be done at a sufficiently high Reynolds number for the guide vanes to be effective. At a lower Re number, the boundary layer grows to a thickness large enough to prevent a sufficiently high flow rate from entering the guide vanes. While typical wind tunnel tests are done at the model scale of 1:50 – 1:80, they would suffer from this deficiency and could provide inaccurate results on the efficacy of the guide vanes. Matsuda et al., (2001) did experimental tests on a 1:10 model scale of a bridge deck. They found that the Re number influenced the unsteady aerodynamic force coefficients and the Strouhal number.

¹ Mach number, defined as a ratio of the flow velocity and the velocity of sound.

Alternatively, flow field equations can be solved numerically using computational fluid dynamics. The development of the CFD codes intensified in the 1990s with the development of powerful computing machines. Today, there is a vast amount of commercial and open-source CFD codes that can successfully deal with the most complex fluid dynamics problems.

In Section 2.2, three types of flow were defined: laminar, transient, and turbulent. In practice, in bridge engineering, due to the size of the structures and wind velocities occurring, the flows are almost always turbulent. When the flow is in the turbulent regime, the velocity and other flow variables vary in time chaotically and randomly (see e.g., Figure 2.31).

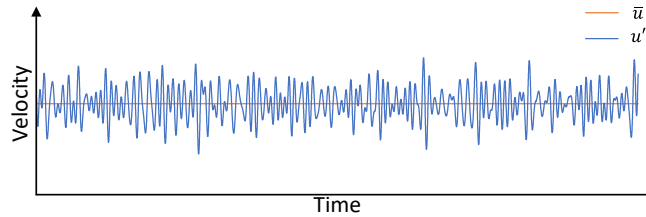


Figure 2.31. Typical velocity measurement in turbulent flow (adapted from Versteeg & Malalasekera, 2007)

For an easier description of the flow field, the velocity field variable can be decomposed into a steady mean value $\bar{u}$ and a fluctuating component $u'(t)$ resulting in $u(t) = \bar{u} + u'(t)$, named *Reynolds decomposition* (Versteeg & Malalasekera, 2007). These fluctuations can range in frequency from ≈ 0 to tens of kHz. The same approach (decomposition) can be applied to velocity in the other two directions and the pressure field variable. Once the Reynolds decomposition is applied to all four flow variables, a resulting set of equations is called *Reynolds-averaged Navier-Stokes (RANS)* equations. There is a great number of books and papers that describe the derivation of the RANS equations, here it is only noted that when the flow variables are decomposed, an additional six unknowns are added named *Reynolds stresses*. With these additional unknowns, and four previously defined (see Section 2.5.1), the system of equations becomes underdetermined. This is commonly referred to as the *turbulence closure problem* (Andersson et al., 2011). For most engineering purposes we are almost always satisfied with the information about the time-averaged properties of the flow (e.g., mean velocities and mean pressure) or, at most, the relatively slowly varying element of the fluctuations. Due to this, CFD simulations are often carried out with procedures based on the RANS equations (Versteeg & Malalasekera, 2007). When the fluctuations are averaged over a time step¹, the unsteady behaviour of the flow is captured in the mean quantities. This approach is named *Unsteady RANS* or *URANS*.

To be able to compute the turbulent flows using RANS equations, we must employ *turbulence models*. Turbulence models are used to *close* the underdetermined system of equations. Normally, the turbulence models are classified based on the number of additional equations that need to be solved. The most used turbulence models are $k - \varepsilon$ model, $k - \omega$ model and the *Shear Stress Transport (SST)* model, all have two additional equations. The $k - \varepsilon$ model is often used in flows with small pressure gradients (Launder & Sharma, 1974), while the opposite holds for the $k - \omega$ model (Wilcox, 1988). The SST model combines $k -$

¹ A relatively large time-step when compared to the frequency range of the flow fluctuations.

ε and $k - \omega$ models, by employing the former in the free-stream part of the domain, and the latter in the near-wall region where the pressure gradients are high (Menter, 1994).

A more complex way of dealing with the turbulence is by numerically resolving the flow, i.e., solving the four equations describing the flow. This approach in CFD is called *Direct Numerical Simulation* (DNS). To be able to resolve the small turbulence scales, the grid size requirements increase dramatically when compared to simulations employing the turbulence models and depending on the flow fluctuations frequency range, the time step size is decreased. With this, the processing power needed for running these, highly computationally demanding simulations, renders most of today's workstations unfit for the task. To avoid resolving the smallest eddies, *Large Eddy Simulation* (LES) uses a spatial filter to separate small from large eddies and numerically resolve only the later ones. Rodi, (1997) reported that the CPU time needed for the LES simulations was 35 times larger than that needed for the simulations employing RANS equations. The demand for well-refined grids largely comes from the regions of the high pressure gradients (wall boundary layer) where the small eddies are formed. Alternative simplifications are offered by *Delayed Detached Eddy Simulation* (DDES). DDES is a hybrid RANS-LES model which employs standard RANS turbulence modelling in the near wall region while in other regions it enables LES. Although this approach decreases the computational power needed, for the LES simulations, the domain should be modelled as 3D, instead of 2D, usually employed with turbulence models. 2D LES simulations produce significantly worse results than their 3D counterparts when vortex shedding is studied (Murakami & Mochida, 1995). Figure 2.32 shows the difference between these approaches when resolving different turbulence scales in a flow.

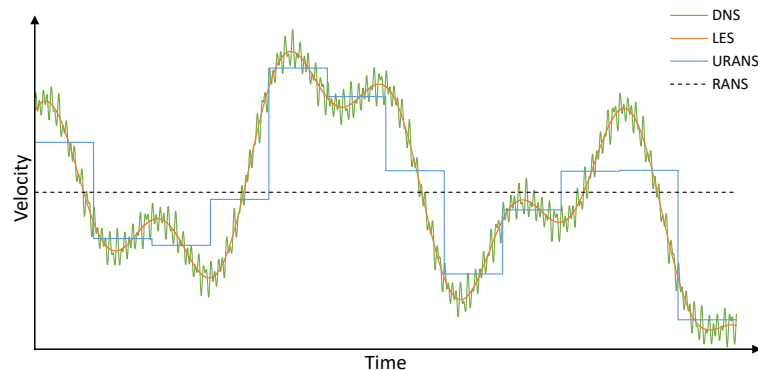


Figure 2.32. Representation of resolved turbulent scales at a point in a turbulent flow

2.5.3 Applicability in Bridge Engineering and Limitations

Although CFD has advanced dramatically, both through the development of sophisticated solvers and turbulence models and through the development of hardware capable of solving non-linear partial differential equations, experimental studies in wind tunnels are still the main pillar of the aerodynamic analysis of bridges. The uncertainties and computational requirements that come with CFD still put it out of reach for many non-research endeavours. It is noted however that in other industries, such as automotive or aerospace, CFD is a tool commonly used in the design process. Nevertheless, many studies have tried using CFD to replicate the results obtained in the wind tunnel tests of bridges. Zhang et al. (2021) modelled the Rose Fitzgerald Kennedy Bridge in Ireland using OpenFOAM. Their analysis consisted of developing 3D models and using the DDES method with the SST for turbulence modelling. The mesh they developed for the analysis

comprised approximately 33M cells and for running the analyses they used a supercomputer based in the Irish Centre for High-End Computing (ICHEC). Nieto et al. (2009) studied several deck shape options (see Figure 2.33) and provided the results for the static aerodynamic coefficients. Based on that, they made recommendations on the most favourable shape at the conceptual design stage. The models developed consisted of approximately 500,000 cells.

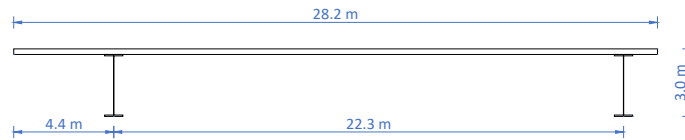


Figure 2.33. Original deck shape analysed by Nieto et al. (2009)

T. Zhang et al., (2020) used CFD analysis during the development of the VIV mitigation measures for the Yanpingba Yangtze River Bridge in China. Their models were 2D URANS using the SST turbulence model. They used the CFD results to identify the regions of the deck responsible for the generation of the vortices and to better understand the structure of the vortices forming in the wake. This allowed them to develop several VIV mitigation measures, later implemented in the wind tunnel testing. All their CFD simulations were done on a stationary deck model. Bruno & Mancini, (2002) analysed the importance of barriers on the aerodynamics of two semi-streamlined bridge decks by using URANS (with the SST model) and LES (see Figure 2.34).

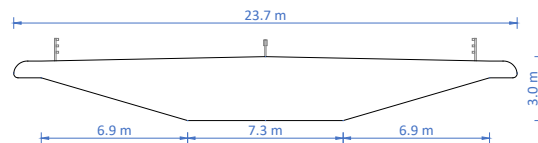


Figure 2.34. The Normandy Bridge deck section analysed by Bruno & Mancini, (2002)

The 2D LES analysis was used by Bruno & Khris, (2003) in their analysis of the Great Belt Bridge (Figure 2.35). They managed to obtain results comparable to those from the 3D LES analysis, however, they state that the methodology should not be applied to more complex flows before similar validation with the 3D LES is undertaken and that it might not yield the results of the same quality. Mannini et al., (2010) analysed a trapezoidal deck section (Figure 2.36) using URANS (with the SA¹ turbulence model). They stressed the importance of the radius of rounded edges and the effect it has on the flow field in the deck wake. They analysed the static lift coefficients and Strouhal number of the rigid deck model. Vairo, (2003) implemented the *Arbitrary Lagrangian Eulerian*² (ALE) method in solving the flow field. They used $k - \varepsilon$ turbulence model to calculate static load coefficients and flutter derivatives (see Section 2.4) for the Normandy Bridge and Great Belt Bridge. To simplify the simulation models, the barriers were neglected.

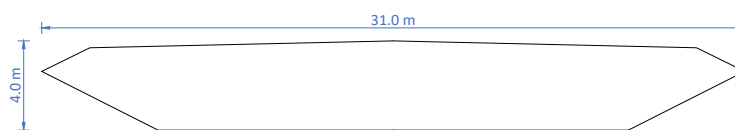


Figure 2.35. The Great Belt Bridge section analysed by Bruno & Khris, (2003) and Bruno & Mancini, (2002)

¹ Spalart-Allmaras model is a one-equation turbulence model developed for streamlined profiles (e.g., airfoils) (Spalart & Allmaras, 1992).

² In ALE, domain boundaries are also deforming compared to grid-only deformation occurring in the traditional Eulerian method. More on ALE formulation can be found in Hirt et al., (1974).

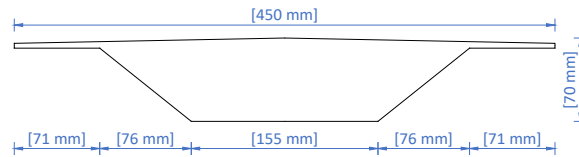


Figure 2.36. Trapezoidal deck section [model scale] analysed by Mannini et al., (2010)

Twin box deck sections were also subject to several CFD analyses. Goering & Ramponi (2019) used the SST turbulence model to identify flutter derivatives for galloping and static force coefficients for a range of angles of attack for the Stonecutters Bridge in Honk Kong (Figure 2.37). Jeong et al., (2019) used 2D URANS and the SST turbulence model to analyse a twin deck section (Figure 2.38) for angles of attack ranging between -10° to $+10^\circ$. Their model comprised approximately 300,000 cells, and the rigid deck section. The results included static force coefficients and the Strouhal number estimate which compared well to the wind tunnel test results.

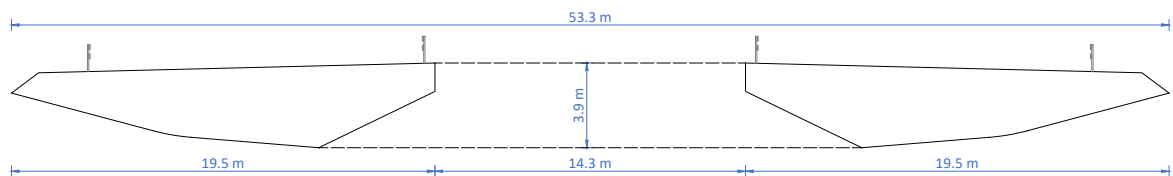


Figure 2.37. Stonecutters Bridge deck section analysed by Goering & Ramponi (2019)

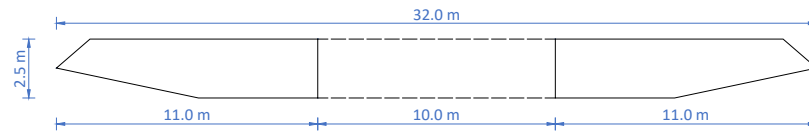


Figure 2.38. Twin box deck analysed by Jeong et al., (2019)

Zhan et al., (2014) used the 2D LES model to analyse the fluid-structure interaction problem of a twin box bridge deck (geometry similar to that of the Stonecutters Bridge in Figure 2.37). They obtained the results for the lock-in range and the displacement amplitude due to the VIV. The results are of the same order of magnitude as the results from the wind tunnel tests (see Figure 2.39). The discrepancy between the CFD and the wind tunnel results was attributed to the existence of nonlinear damping in the wind tunnel model.

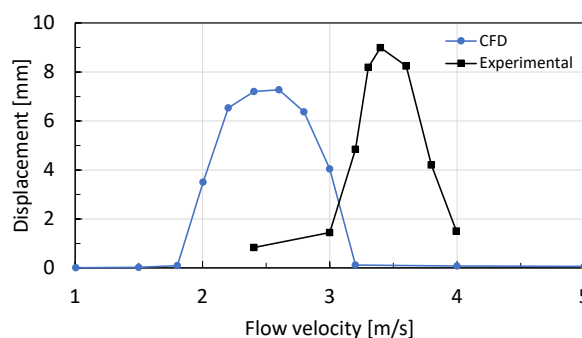


Figure 2.39. VIV results (at model scale) of the analyses done by Zhan et al., (2014)

As seen in this section, CFD has been used in bridge engineering in the past decades, however, most of the analyses have been done on streamlined and semi-streamlined deck sections. The flow field around streamlined sections is less disturbed than in the case of bluff sections, which makes mesh generation an easier task. An example of the bluff deck section (Figure 2.33) is that analysed by Nieto et al. (2009), who

reported that their model had approximately half a million cells, while the more streamlined sections (see e.g., Figure 2.38) were analysed by models comprising approximately 300,000 cells. Also, due to the complexity that the barriers introduce, most of the analyses omit them, while it was shown that the barriers play an important role in the aerodynamic behaviour of the bridge deck (Bruno & Mancini, 2002). Even fewer studies were done on the coupled FSI problem of the bridge decks. While the static force coefficients provide useful information on the stability of the deck, the lock-in range and the amplitude of the displacements are what govern the bridge design when it comes to the vortex-induced vibration phenomenon. Furthermore, to the best of the author's knowledge, only Nieto et al. (2009) and T. Zhang et al., (2020) used CFD as a tool to inform the design at the early, conceptual, stage.

Experience of practice suggests that the reasons for the lack of CFD presence at the design stage are twofold: (i) a detailed CFD analysis requires a large amount of computing power for it to be done in a timeframe acceptable for the design stage; (ii) the results achieved by CFD often bring uncertainties that can be removed only by committing to a wind tunnel testing campaign. Thus, engineers are still reluctant to choose CFD as a design tool in bridge engineering.

2.6 Overview

As seen in this chapter, flow-induced vibrations in bridges remain a concern for designers. Modern design standards coupled with wind tunnel tests allow bridge engineers to reach aerodynamically acceptable solutions. However, the path they have to follow is paved with many complexities and unknowns involving non-linear coupling of the bridge behaviour and the surrounding airflow.

It was shown that CFD is not a common tool in the bridge design process and that, with a couple of notable exceptions, it has been used only for research endeavours (discussed in Section 2.5.3). The reasons for this are primarily linked to (i) the high computing power needed for running CFD simulations and, (ii) the level of knowledge (and experience) a CFD user needs to have to be able to recognise flawed models and incorrect results they would yield. Nevertheless, as already noted several attempts were successfully made to intertwine CFD and bridge engineering. The focus of these efforts was primarily on streamlined deck sections while more aerodynamically intricate sections (i.e., bluff sections) have received little attention.

Although design standards dealing with bridge aerodynamics are constantly developing, they still are inherently limited (discussed in Section 2.1). These limitations are passed on to engineers who, in order to achieve a conforming design solution, are forced to commit to time-consuming and expensive wind tunnel testing campaigns. For example, turbulence intensity in the oncoming flow has been recognised by many to affect the vortex-induced vibrations (discussed in Section 2.2 and Section 2.3.4). The increased turbulence intensity disrupts the shedding process and, in some cases, can neutralise the resulting vibrations. However, it was shown that that is not always the case, see e.g., Matsumoto et al., (1993). The design code used in Japan (Japan Road Association, 2007), recognises this and includes the level of turbulence intensity as a parameter in the prediction of the aerodynamic phenomena type that may occur. The standard used in Ireland and the UK (PD 6688-1-4) does not consider turbulence intensity for the estimation of the VIV susceptibility.

Another area that can be further improved is the effect of the overhang length on the vortex induced vibrations. It was shown that by placing a splitter plate in the wake of a bluff body, the amplitude of the vortex-induced vibrations can be altered (discussed in Section 2.2.7). In certain cases, the overhang found on bridge decks can act as a splitter plate. This has been recognised only to a certain degree in the design codes (PD 6688-1-4 and to a higher degree in the Japanese code). Having an understanding of how the overhang affects the vortex shedding process can allow engineers to make more informed decisions on the deck shapes they are committing to in the early design stages.

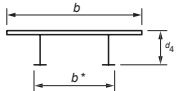
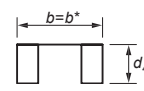
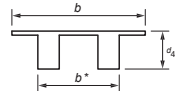
The gaps identified here are the context of this research campaign. Before providing the details on how this will be achieved, a series of wind tunnel testing results are presented in Chapter 3 in order to contextualise the aerodynamic response of bluff decks on cable-stayed bridges.

Chapter 3

Available Wind Tunnel Data

The main topic of this research is focused on simulating vortex-induced vibrations of bridge decks that are not streamlined. The reasons for the introduction of the wind tunnel data to this study are twofold: (i) to form a clearer picture of the aerodynamic behaviour of these types of bridge decks; (ii) to provide comparison data to evaluate various CFD modelling decisions taken throughout this study. The wind tunnel data used in this research was obtained from the testing campaigns conducted as part of the design process of three cable-stayed bridges. Their shapes, sizes and dynamic properties are summarised in Table 3.1 while detailed dimensions are given in the sections below. The results presented here are based on the raw results included in the factual reports provided by the subcontractors. They are not the final conclusions and recommendations of the testing campaigns. Details beyond that given here can be found in their respective factual reports, however, they are not available to the public.

Table 3.1. Summary deck sections tested in three wind tunnel testing campaigns

Property		Northern Spire Bridge		Clyde Bridge		Narrow Water Bridge	
Deck shape							
Deck type (PD 6688-1-4)		 Bridge type 2		 Bridge type 1A		 Bridge type 1	
Scale		1:1	1:50	1:1	1:30	1:1	1:35
Deck width (m)	b	24.4	0.488	12.3	0.41	16.8	0.480
Girder/Box distance (m)	b^*	21.6	0.432	12.3	0.41	9.6	0.274
Deck depth (m)	d_4	2.5	0.050	2.4	0.08	1.68	0.048
Parapet height (m)	h	1.25	0.025	1.25	0.05	1.4	0.040
Mass (kg/m)	m	29,325	11.73	3,933	4.37	9,395	7.67
Bending frequency (Hz)	f_n	0.41	4.71	1.51	7.08	0.75	3.61
Damping (log-dec)	δ_s	2.8%	2.8%	3.0%	3.0%	3.1%	3.1%
Scruton number	Sc	209	209	33	33	168	168

3.1 Northern Spire Bridge

The Northern Spire Bridge is a two-span cable-stayed bridge across the River Wear in Sunderland, UK, with spans of 148 m in length each (Figure 3.1). The wind tunnel campaign was executed in 2018 at Centre Scientifique et Technique du Bâtiment (CSTB), Nantes, France in their Boundary Layer wind tunnel facility. The data was presented in the Factual Report (Flamand & Knapp, 2018). The tests were done at a scale of 1:50 yielding a Reynolds number, $Re \approx 1.5 \times 10^4$, based on the deck depth (D), and at the velocity scale¹ of 4.35. Although a number of deck shape variations were tested, only the results for the deck section of interest to this research are presented here.

¹ Defined as a product of frequency and geometric scales.



Figure 3.1. The Northern Spire Bridge in Sunderland, UK¹

Figure 3.2 shows the deck shape dimensioned at the full scale and model scale, the adopted sign convention, the sign convention for the wind loads and the wind incidence angle. This deck section fits under *Bridge type 2* as defined in PD 6688-1-4 (see Figure 2.1). The spanwise length of the model was 1935 mm.

The tests were conducted in two stages: (i) tests on the fixed model which yielded aerodynamic force coefficients and their gradients²; (ii) the *free vibration* tests with the model supported by springs (see Figure 3.3) to obtain the response to periodic vortex shedding.

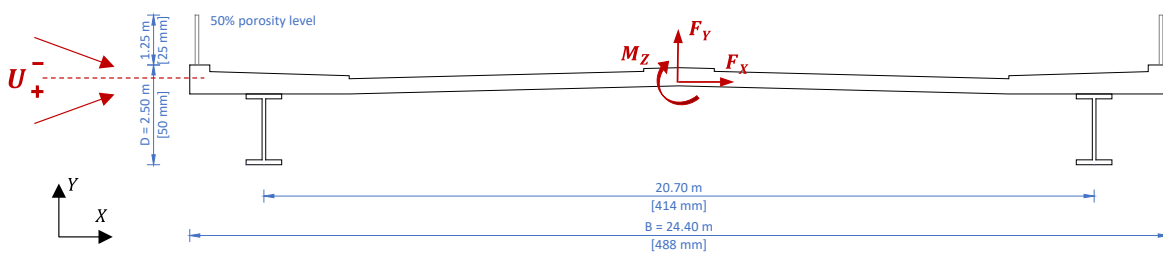


Figure 3.2. Northern Spire Bridge deck section: dimensions at full scale and [model scale]; adopted sign convention, deck width B and deck depth D

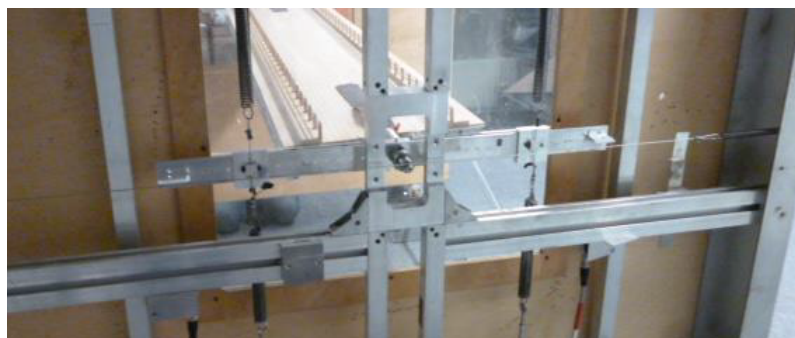


Figure 3.3. Dynamic model assembly and the deck section model (Flamand & Knapp, 2018)

The dynamic model was developed with two degrees of freedom, heaving – equivalent to the bending deformation of the full-scale elastic bridge and rocking – equivalent to the torsional deformation of a flexible deck. It should be noted that the deck model itself is rigid, while the two degrees of freedom are

¹ Source: <https://www.rod.ie/projects/northern-spire-bridge>

² The gradient of lift coefficient provides insight into galloping susceptibility of a cross section. See section 2.3.4.

modelled using springs at both ends (Figure 3.3). The computational models developed in this research neglect the latter degree of freedom as the heaving mode dominates the VIV of this structure (based on the results presented in the factual report). The geometric and dynamic properties of the tested deck section are reported in Table 3.2.

Table 3.2. Model and full-scale properties (first heave mode) of the tested Northern Spire Bridge deck section

Property	Description	Full scale	Model scale	Unit
-	Geometric scale	1:1	1:50	-
D	Depth	2.50	0.050	m
B	Width	24.40	0.488	m
m	Mass	29,325	11.70	kg/m
f_n	1 st natural frequency	0.41	4.71	Hz
δ_s	Damping (log-dec)	2.80 %	2.80 %	-
Sc	Scruton number	215	215	-

The aerodynamic coefficients for a range of angles of attack at a specific wind speed¹ are shown in Figure 3.4. Static force coefficients are defined as in Equation 2-13 in Section 2.2.8. By using the static coefficients in Figure 3.4 and Equation 2-11, it is straightforward to verify that the bridge deck will not experience galloping in the velocity range under consideration. Therefore, the response of the sprung model to aerodynamic forces will be dominated by vortex-induced vibration.

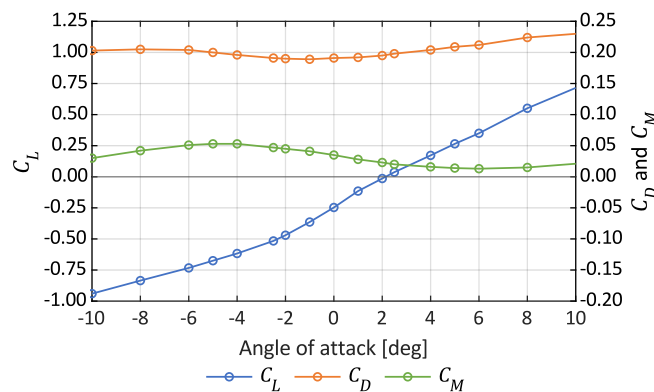


Figure 3.4. Static aerodynamic coefficients for the Northern Spire Bridge deck section shown in Figure 3.2 (adapted from Flamand & Knapp, 2018)

The experimentally obtained RMS of the amplitude response of the *free vibration* tests is shown in Figure 3.5. The amplitudes and velocities reported here are converted to the full scale as is normal practice in bridge testing. The maximum vertical displacement occurs at approximately 17 m/s suggesting a lock-in range between approximately 13.6 m/s and 21 m/s. The Strouhal number² is estimated as 0.075. These VIV results are for the tests undertaken under the turbulence intensity in the oncoming flow of 4%. Additional tests were done in flows with higher turbulence intensity and the results, along with a discussion, are presented in Appendix A.

¹ Defined based on the Reynolds number dependency test conducted for the static force coefficients.

² See Equation 2-9 in Section 2.3.3 .

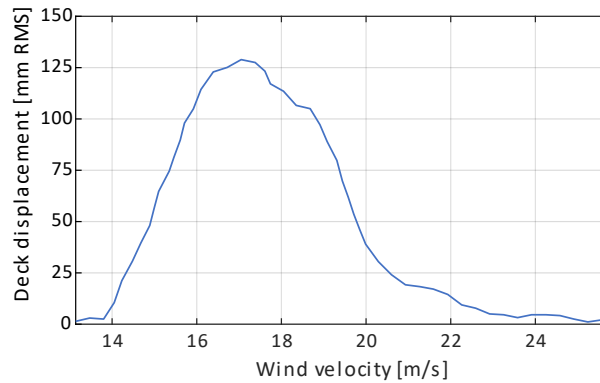


Figure 3.5. Vertical deck response at full scale for the Northern Spire Bridge deck section shown in Figure 3.2 (adapted from Flamand & Knapp, 2018)

Figure 3.6 shows the spectrogram of the deck displacement oscillation. Several observations can be made: the lock-in frequency is at approximately 0.40 Hz which corresponds to the bridge's natural frequency of 0.41 Hz (see Table 3.2); a strong lock-in occurs between approximately 14 m/s and 21 m/s which is consistent with results shown in Figure 3.5; there is no clear lock-out point. However, a narrow frequency component which is linearly dependent on the velocity is clearly visible away from the lock-in regions. This is likely to be the vortex shedding frequency. A Strouhal law line is plotted in red, based on this observation. Also, it can be noted that although the lock-in becomes significantly weaker after approximately 21 m/s, the non-zero amplitude displacement does not disappear until approximately 24 m/s (see Figure 3.5). Similar behaviour was seen for a circular cross-section results shown in Figure 2.10 in Section 2.2.3. A clearer picture of the vortex shedding frequency at different wind velocities could be provided if the flow fluctuations were measured in the deck's wake, however, this test was not conducted. It is interesting to note that the vortex shedding frequency appears to lock on to the second harmonic of the natural frequency in the range between 23 m/s and 30 m/s but this does not cause substantial deck displacement.

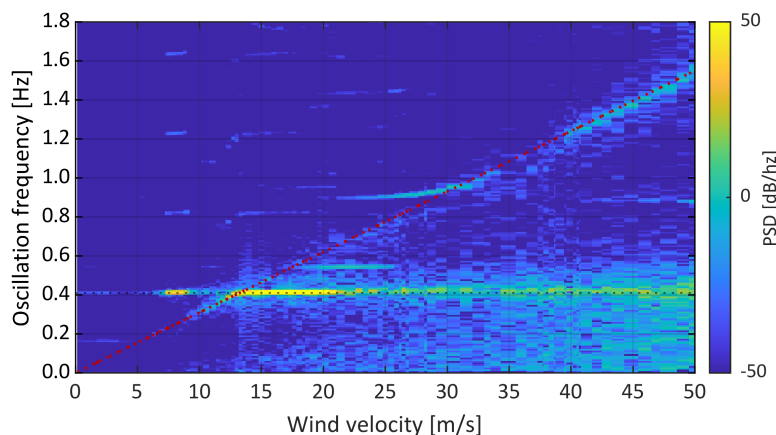


Figure 3.6. Spectrogram of the deck oscillation for a range of wind velocities for the Northern Spire Bridge deck section with the Strouhal law curve marked in red (adapted from Flamand & Knapp, 2018)

Additional tests were done on deck sections with horizontal and inclined plates mounted on the leading and trailing edges forming overhang-like geometry. The results showed the deck VIV response to be highly affected by these plates, in some cases attenuating and, in some cases, increasing the maximum response amplitude. The effects the overhang has on the deck VIV response are investigated in Chapter 6.

3.2 Clyde Bridge

The Clyde Bridge is a double-leaf single-span swing cable-stayed bridge crossing the River Clyde near the city of Glasgow, Scotland with a total length of 130 m (Figure 3.7). The wind tunnel testing was conducted in 2022 by CSTB, Nantes, France in their Boundary Layer wind tunnel facility. The results were presented in the Factual Report (Carusone & Flamand, 2022). Although a number of deck shapes were tested, the results for two deck sections of interest to this research are presented here: (i) deck with barriers and (ii) deck with barriers and fairing plates at the two edges. The wind fairings are non-structural elements and are only fitted to a deck to alter its aerodynamic profile. One of the aims of this wind tunnel testing campaign was to develop a geometry of the wind fairings such that the deck exhibits aerodynamically acceptable behaviour, from the design perspective. This is an iterative process that commences with an initial shape of the wind fairings suggested by the design team. The tests were done at a scale of 1:30 at $Re \approx 4 \times 10^4$ based on the deck depth (D) with the velocity scale¹ of 6.4.



Figure 3.7. The Clyde Bridge in Scotland in the open position (left) and closed (right)²

Figure 3.8 shows the deck shape dimensioned at the full scale and model scale, the adopted sign convention, the sign convention for the wind loads and the wind incidence angle. This deck section fits under *Bridge type 1A* as defined in PD 6688-1-4 (see Figure 2.1). The spanwise length of the model was 2000 mm.

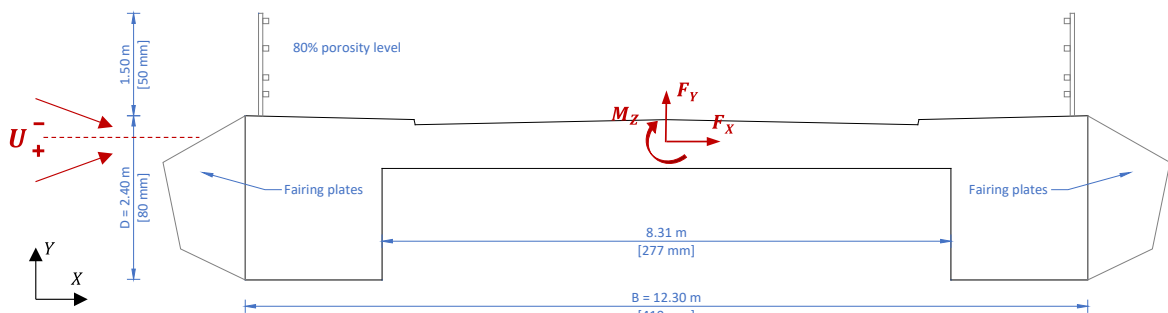


Figure 3.8. Clyde Bridge deck section: dimensions at full scale and [model scale]; adopted sign convention, deck width B and deck depth D

Again, the tests were conducted in two stages: (i) tests on the fixed model which yielded aerodynamic force coefficients and gradients³; (ii) the *free vibration* tests with the model supported by springs (as shown in Figure 3.9) to obtain the response to periodic vortex shedding.

¹ Defined as a product of frequency and geometric scales.

² Source: <https://www.rod.ie/about/news/rodhh-design-first-opening-road-bridge-over-river-clyde>

³ The gradient of lift coefficient provides insight into galloping susceptibility of a cross section. See section 2.3.4.



Figure 3.9. Test setup for the Clyde Bridge deck model (Carusone & Flamand, 2022)

The dynamic model was built with two degrees of freedom, heaving, or bending (translation in the Y -direction) and rocking or torsion (rotation about the Z -axis). The computational models developed in this research neglect the latter degree of freedom as the heaving mode dominates the VIV of this structure (based on the results presented in the Factual Report). The section properties of the tested deck section are reported in Table 3.3.

The aerodynamic coefficients for a range of angles of attack at a specific wind speed¹ are shown in Figure 3.10. Static force coefficients are defined as in Equation 2-13 in Section 2.2.8. By using the static coefficients in Figure 3.10 and Equation 2-11, it is noticeable that the section is stable when it comes to galloping.

Table 3.3. Model-scale and full-scale properties (first heave mode) of the tested Clyde Bridge deck section

Property	Description	Full scale	Model scale	Unit
-	Geometric scale	1:1	1:30	-
D	Depth	2.40	0.08	m
B	Width	12.30	0.41	m
m	Mass	3,923	4.36	kg/m
f_n	1 st natural frequency	1.51	7.08	Hz
δ_s	Damping (log-dec)	3.0%	3.0%	-
Sc	Scruton number	33	33	-

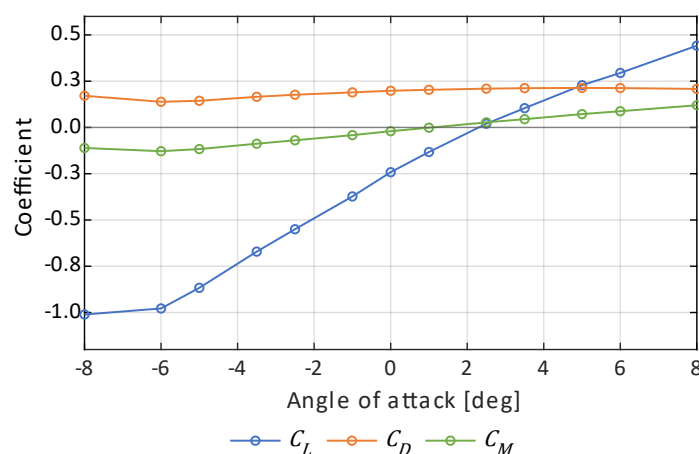


Figure 3.10. Static aerodynamic coefficients for the Clyde Bridge deck section with the fairing plates as shown in Figure 3.8 (adapted from Carusone & Flamand, 2022)

¹ Defined based on the Reynolds number dependency test conducted for the static force coefficients.

The experimentally obtained RMS of the amplitude response of the *free vibration* tests is shown in Figure 3.11. The amplitudes and velocities reported here are converted to the full scale as is normal practice in bridge testing. The maximum displacement occurs at approximately 38 m/s suggesting a lock-in range between approximately 33 m/s and 48 m/s. The Strouhal number¹ is estimated at 0.11. These VIV results are for the test undertaken with the turbulence intensity in the oncoming flow of 3.8%. Additional tests were done in the flows with higher turbulence intensity and the results, along with a discussion, are presented in Appendix A.

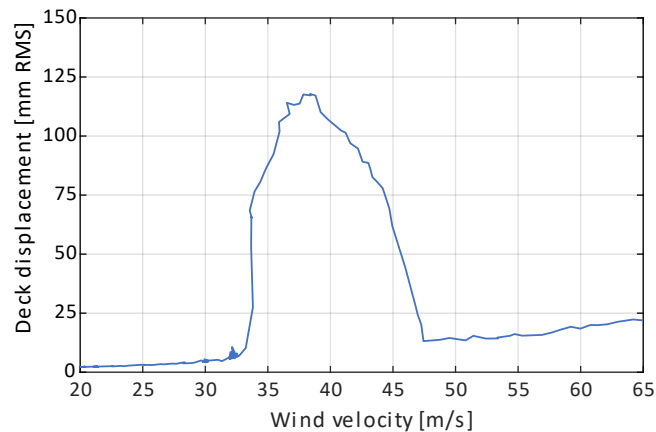


Figure 3.11. Vertical deck response at full scale for the Clyde Bridge deck section without the fairing plates as shown in Figure 3.8 (adapted from Carusone & Flamand, 2022)

Figure 3.12 shows the spectrogram of the deck displacement oscillations for a range of wind velocities. A very clear lock-in region can be observed for the same range of velocities shown in the amplitude response diagram (Figure 3.11). It occurs at approximately 1.5 Hz which is in line with the first natural frequency of the structure (1.51 Hz). Strouhal law curve is marked with the red dotted line but it is only weakly visible from the spectrogram (see range 20-30 m/s), suggesting a weak vortex shedding outside the lock-in region. This is substantiated in Figure 3.13 which shows the spectrogram of the flow fluctuations measured in the wake of the deck. Only at the lock-in region, a noticeable shedding process is observed. This suggests that the motion of the bridge deck reinforces vortex shedding which is in effect motion induced vortex shedding. It is also noteworthy that there is always some response at the natural frequency.

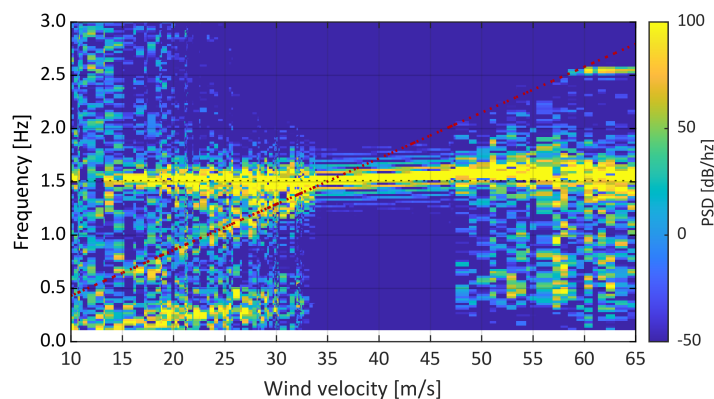


Figure 3.12. Spectrogram of the deck oscillation for a range of wind velocities for the Clyde Bridge deck section with the Strouhal law curve marked in red (adapted from Carusone & Flamand, 2022)

¹ See Equation 2-9 in Section 2.3.3 .

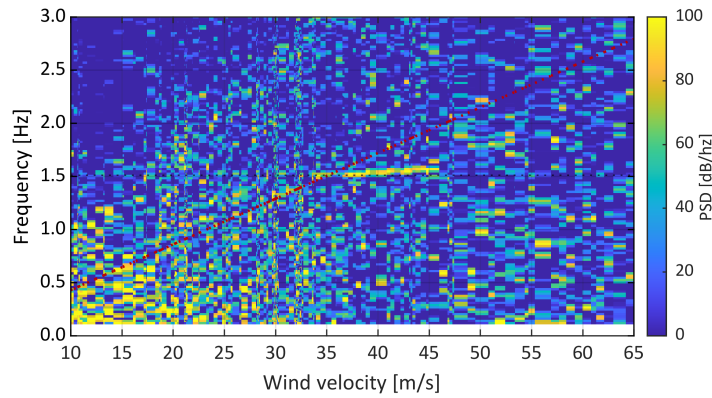


Figure 3.13. Spectrogram of the flow fluctuations measured in the Clyde Bridge deck wake for a range of wind velocities with the Strouhal law curve marked in red (adapted from Carusone & Flamand, 2022)

3.3 Narrow Water Bridge

The Narrow Water Bridge is a two-span cable-stayed bridge in Louth Co., Ireland (Figure 3.14) whose construction is expected to begin in the summer of 2024 (as of April 2024). It comprises a fixed span (138 m in length) and a rolling bascule span (56.8 m in length). The wind tunnel testing was done by Svend Ole Hansen ApS (SOH) in 2022 and early 2023 in their wind tunnels in Denmark and the USA. The results were presented in the Factual Report (Svend Ole Hansen ApS, 2023). The tests were done at 1:35 scale at $Re \approx 2 \times 10^4$ based on the deck depth (D) with the velocity scale¹ of 7.3.



Figure 3.14. Future Narrow Water Bridge in Louth Co., Ireland²

Figure 3.15 shows the deck shape at the full scale and model scale, the adopted sign convention, the sign convention for the wind loads and the wind incidence angle. This deck section fits under *Bridge type 1* as defined in PD 6688-1-4 (see Figure 2.1). The spanwise length of the model was 1680 mm.

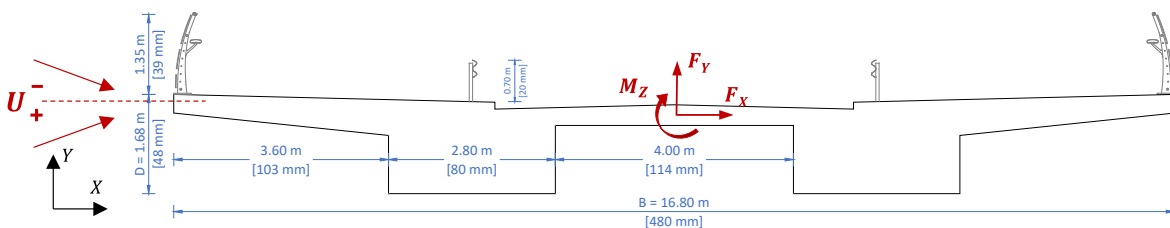


Figure 3.15. Narrow Water Bridge deck section: dimensions at full scale and [model scale]; adopted sign convention, deck width B and deck depth D

¹ Defined as a product of frequency and geometric scales.

² Source: <https://www.rod.ie/about/news/narrow-water-bridge-project-moves-tender-stage>

Again, the tests were conducted in two stages: (i) tests on the fixed model which yielded aerodynamic force coefficients and gradients¹; (ii) the “free vibration” tests with the model supported by springs (as shown in Figure 3.16) to obtain the response to periodic vortex shedding.

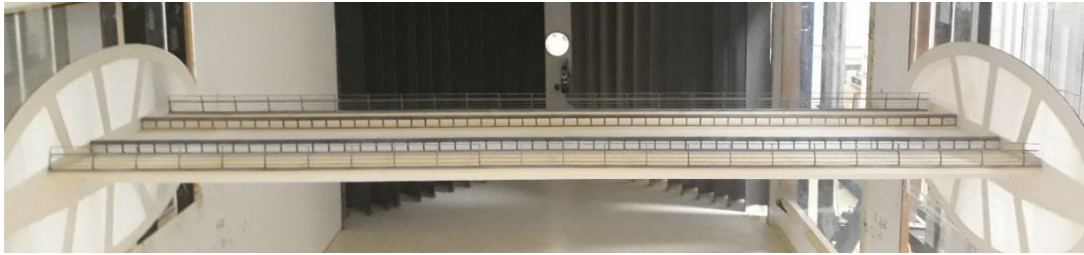


Figure 3.16. Model assembly for one of the tested deck sections of the Narrow Water Bridge (Svend Ole Hansen ApS, 2023)

The dynamic model was modelled with two degrees of freedom, heaving, or bending (translation in the Y -direction) and rocking or torsion (rotation about the Z -axis). The computational models developed in this research neglect the latter degree of freedom as the heaving mode dominates the VIV of this structure (based on the results presented in the Factual Report). The section properties of the tested deck section are reported in Table 3.4.

The aerodynamic coefficients for a range of angles of attack at a specific wind speed² are shown in Figure 3.17. The results are for the deck with the wind shielding included. Static force coefficients are defined as in Equation 2-13 in Section 2.2.8.

Table 3.4. Model and full scale properties (first heave mode) of the tested Narrow Water Bridge deck section

Property	Description	Full scale	Model scale	Unit
-	Geometric scale	1:1	1:35	-
D	Depth	1.68	0.048	m
B	Width	16.80	0.480	m
m	Mass	9,237	7.54	kg/m
f_n	1 st natural frequency	0.75	3.61	Hz
δ_s	Damping (log-dec)	various	various	-
Sc	Scruton number	various	various	-

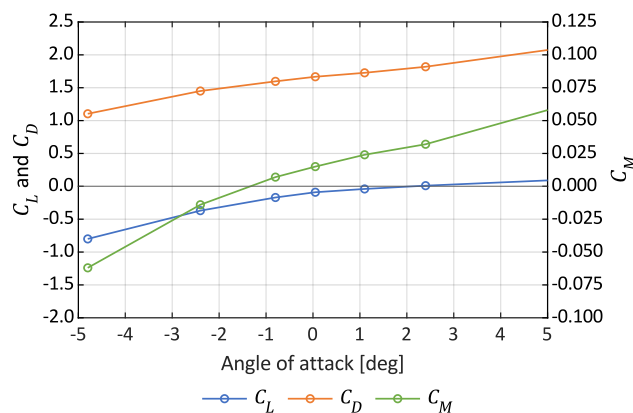


Figure 3.17. Static aerodynamic coefficients for the Narrow Water Bridge deck with the wind shielding as shown in Figure 3.15 (adapted from Svend Ole Hansen ApS, 2023)

¹ The gradients of lift and drag coefficients provide insight into galloping susceptibility of a cross section. See section 2.3.4.

² Defined based on the Reynolds number dependency test conducted for the static force coefficients.

The experimentally obtained RMS of the amplitude response of the *free vibration* tests for the deck section with and without the wind shielding for two levels of damping is shown in Figure 3.18. The amplitudes and velocities reported here are converted to the full scale as is normal practice in bridge testing. For the deck section without the wind shielding, the maximum displacement (≈ 56 mm) occurs at approximately 17 m/s suggesting a lock-in range between approximately 15.8 m/s and 18 m/s. The addition of the wind shielding (with a porosity level of approximately 80%) had an adverse effect on the results: the maximum displacement amplitude increased to 183 mm (although at a higher wind velocity) and the lock-in region became wider (between 20 m/s and 29 m/s). Increased damping for the case of the deck without the wind shielding, neutralised the vortex-induced vibrations, but for the case with the wind shielding, it only decreased the maximum amplitude by 30% with a virtually unchanged lock-in region. Furthermore, adding the wind shielding increased the Strouhal number, due to the increase in the effective depth. These VIV results are for the tests undertaken under the turbulence intensity in the oncoming flow of 1%. Additional tests were done in the flows with the higher turbulence intensity and the results, along with a discussion, are presented in Appendix A. The data needed for generating spectrograms of the deck displacement and the wake flow velocity was not recorded hence these graphs are not included.

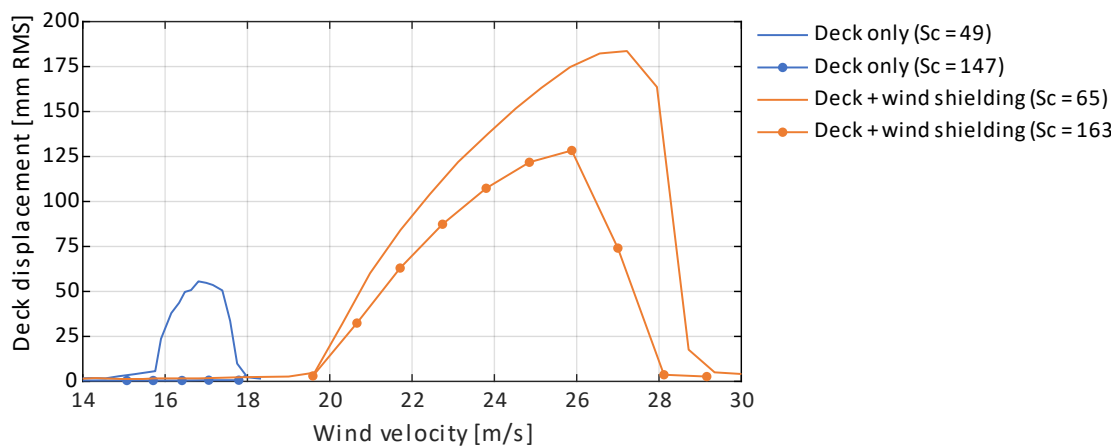


Figure 3.18. Vertical deck response at full scale for the Narrow Water Bridge deck, with and without the wind shielding, as shown in Figure 3.15 (adapted from Svend Ole Hansen ApS, 2023)

3.4 Geometric Considerations

Wind tunnel tests can be conducted at geometric scales ranging between 1:30 (deck sectional tests) and 1:100 (full aeroelastic tests). At these scales, very often it is difficult to construct a model which fully replicates the level of detail present at the full scale. This issue mainly refers to the small geometric features present on bridge decks (e.g., barriers, light poles, and others). As seen in the case of Narrow Water Bridge, wind shielding had a large effect on the VIV response. Figure 3.19 shows the barriers and wind shielding installed on the tested model. The level of detail is achieved by 3D printing the elements.

In the case of the Northern Spire Bridge test, the modelled barrier replicated its height and porosity level, while the exact geometry was neglected (see Figure 3.20). The design code, PD 6688-1-4, adopts a similar approach. In Section 2.1.1 it is shown that the geometry of the leading edge is taken into account only by considering its height and porosity level.



Figure 3.19. Wind shielding and traffic barriers installed on the Narrow Water Bridge model (Svend Ole Hansen ApS, 2023)

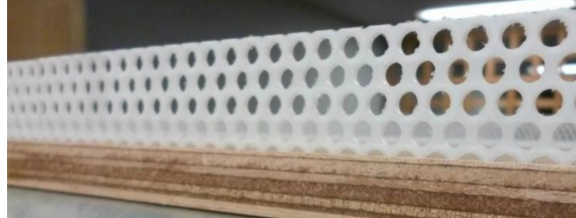


Figure 3.20. The barrier installed on the Northern Spire Bridge model (Flamand & Knapp, 2018)

3.5 Discussion

This chapter comprises different wind tunnel testing results for three cable-stayed bridges. The testing was performed as part of the design process while, here, the results are used for the evaluation of the modelling choices and better understanding of the aerodynamic behaviour of relatively bluff deck sections. All three bridge decks fall within three different bridge deck types as defined by PD 6688-1-4 (see Figure 2.1).

The VIV response of the Northern Spire Bridge was characterised by a smooth deck amplitude response curve. A clear coincidence point was visible, both, on the response curve and the spectrogram, while the lock-out was rather *smeared* across a range of wind velocities. For the Clyde Bridge and the Narrow Water Bridge, the boundaries of the lock-in region were well defined. For the Clyde Bridge, the results at the lock-in were to a certain degree less smooth, suggesting that the model experienced difficulties in reaching stable LCO. In the case of the NSB, a clear dependency between shedding frequency and the wind velocity was visible, which cannot be stated for the Clyde Bridge, suggesting a weaker vortex shedding in the region outside the lock-in. This was noticeable in the spectrograms of the deck displacement oscillations and, in the case of Clyde Bridge, the spectrogram of the flow fluctuations in the deck wake. The response of the NWB deck was highly affected by the introduction of the wind shielding at the edges of the deck. It increased the maximum oscillation amplitude and the size of the lock-in region.

For the NWB, the increased damping had a different effect for cases with and without the wind shielding. It should be noted that adjusting values of damping at the model scale is not a trivial task, especially if the model is developed at a scale for the full aeroelastic test. Firstly, the structural damping cannot be lower than the damping provided by the material of which the model is made. Secondly, artificially increasing the damping can easily alter the modal behaviour of the model, rendering it not suitable for comparison with the full-scale bridge. Both of these points make the damping adjustment a highly involved task. On the other hand, as seen in the case of the Narrow Water Bridge model, having an accurate understanding of the damping level effect on the VIV response is also important. Section 4.4 outlines an approach that can yield these results in a faster manner when combined with CFD. The approach is employed for obtaining results presented in Section 5.3 and Chapter 6.

Chapter 4

Methodology

Chapter 4 gives details of different methodologies and approaches employed in this thesis. It comprises the description of the CFD models and mesh dependency for two studies done. The first part is related to the applicability of CFD in the bridge design process with an emphasis on bluff bridge decks. As the results from the wind tunnel tests conducted on the Northern Spire Bridge (see Section 3.1) were readily available at the research outset, they were used for the evaluation of different modelling choices. The details of this CFD model are given in Section 4.1. The second part of this research deals with the effects of the overhang length on the vortex induced vibration response of the bridge decks. A number of CFD models were developed for this study and their description is given in Section 4.2. The free-to-oscillate and prescribed motion models that were used throughout the thesis, along with the description of the energy balance approach, are given in Section 4.3 and Section 4.4, respectively. All CFD models were developed and executed in Ansys Fluent 2021R1 while the data post-processing and plotting were done in MATLAB.

4.1 Northern Spire Bridge CFD Model

It is worth reiterating that the purpose of this part of the research is to identify practical modelling choices to use CFD as a design tool for bluff bridge decks. The aim is to develop a CFD model that can capture the most important aerodynamic features such as static force coefficients (magnitude and gradients) and the response amplitude under VIV. Furthermore, the model must be simple enough so that its usage is practical for industrial purposes. The approach taken in this part of the research is to compare the results obtained from a commercial RANS code to the wind tunnel data for the Northern Spire Bridge deck described in Section 3.1. This bridge was selected because the results from the wind tunnel tests were readily available at the project outset. The following sections describe the methodology employed for this study while the results are presented in Chapter 5.

4.1.1 Modelling Choices

Figure 4.1 gives an overview of all modelling choices considered in the present study. Three choices for simplifying the geometry were considered: (i) the bare deck (without the girders and barriers), (ii) the deck with the girders and (iii) the full deck. Three options for the domain were explored. While 2D and 3D are self-explanatory, “2.5D” analyses represent a 3D simulation done on a narrow domain with periodic boundary conditions in the spanwise direction. Three turbulence models are covered with each, bringing an extra level of robustness. The simplest one considered was $k - \varepsilon$, this model was then upgraded to SST, and the final turbulence model considered was Transition SST, Fluent’s proprietary model. More details are given in Section 4.1.4. Finally, three different analysis types were considered: steady state (steady fluid flow), transient state (rigid structure, unsteady fluid flow) and full fluid-structure interaction (FSI – vibrating

structure, unsteady fluid flow). The objective was to develop a CFD model sufficiently robust to produce results comparable to the wind tunnel tests but simple enough so that it can be used in industrial practice.

As seen in Figure 4.1, a total of 81 combinations can be made from the suggested choices although not all were simulated. The selected models are listed in the Simulation Matrix (Table 4.1) and the results are presented and discussed in Chapter 5.

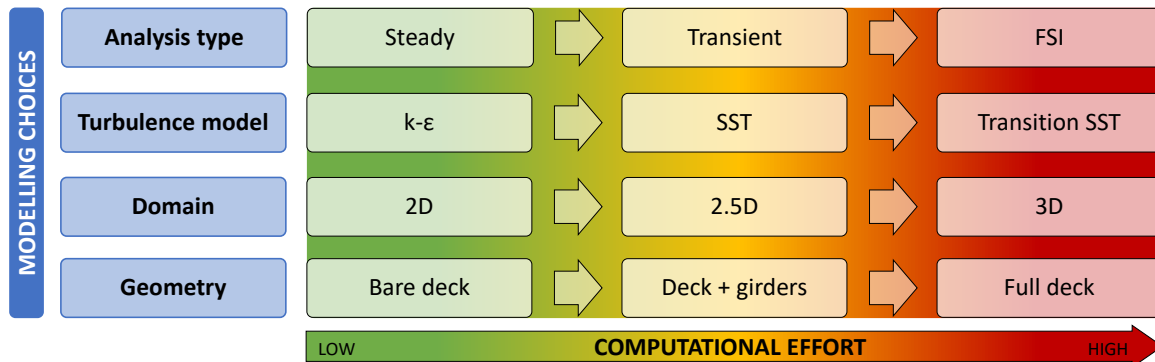


Figure 4.1. Different modelling choices, arranged based on the computational effort needed

Table 4.1. Simulation Matrix

Case no.	Model ID	Description	Model Type	Analysis type	Turbulence model	Turbulence intensity	Domain dimension
1	SM01	Bare deck	Fixed	Steady State	TSST	1%	2D
2	SM02	Bare deck + girders	Fixed	Steady State	TSST	1%	2D
3	SM03	Full deck	Fixed	Steady State	TSST	1%	2D
4	SM11	Full deck - barrier with 2no. openings -	Fixed	Steady State	TSST	1%	2D
5	SM12	Full deck - barrier with 3no. openings -	Fixed	Steady State	TSST	1%	2D
6	SM13	Full deck - barrier with 5no. openings -	Fixed	Steady State	TSST	1%	2D
7	SM14	Full deck - barrier with 6no. openings -	Fixed	Steady State	TSST	1%	2D
8	SM21	Full deck - "2.5D" model -	Fixed	Steady State	SST	1%	3D
9	SM22	Full deck - "3D" model -	Fixed	Steady State	SST	1%	3D
10	SM31 ¹	Full deck	Fixed	Steady State	k-ε	1%	2D
11	SM32	Full deck	Fixed	Steady State	SST	1%	2D
12	SM41 ¹	Full deck	Fixed	Steady State	TSST	10%	2D
13	SM42 ²	Full deck - "3D" model -	Fixed	Steady State	SST	10%	3D
14	TM01 ¹	Full deck - Transient -	Fixed	Transient	SST	1%	2D
15	DM01 ¹	Full deck	FSI	Transient	SST	1%	2D
16	DM02 ¹	Full deck	FSI	Transient	TSST	1%	2D
17	DM03 ¹	Full deck	PM	Transient	SST	1%	2D

Notes: -

¹ The same grid as for model SM03

² The same grid as for model SM22

4.1.2 Meshing Methodology and Grid Sensitivity

In the following sections, a detailed description of the grid used for model SM03 is given. All other models were developed following a similar approach. The model dimensions adopted here correspond to the model dimensions used in the wind tunnel tests (see Figure 3.2).

While it is possible to use a single fully unstructured polyhedral grid, in the current study subdomains are defined and meshed with an unstructured mesh. There are two main reasons for this approach. Firstly, it must facilitate a fully coupled FSI numerical model, albeit with a rigid deck on flexible supports. By defining a subdomain around the bridge deck, only that block would require re-meshing or mesh deformation. Furthermore, the mesh block immediately adjacent to the bridge deck surface can be translated with the deck so moving the re-meshing to a less dense mesh block. Secondly, defining subdomains facilitates control of the localized mesh density, allowing mesh nodes to be concentrated in regions which are fluid-mechanically important (e.g., in the wake, or the vicinity of the barriers). An example of the mesh obtained with this approach is shown in Figure 4.2.

A fully translating body mesh could be considered a better option since it allows for the mesh deformation to be avoided altogether. However, in the case when installation effects (i.e., ground topology) and atmospheric shear layer are to be considered, this approach would not be feasible.

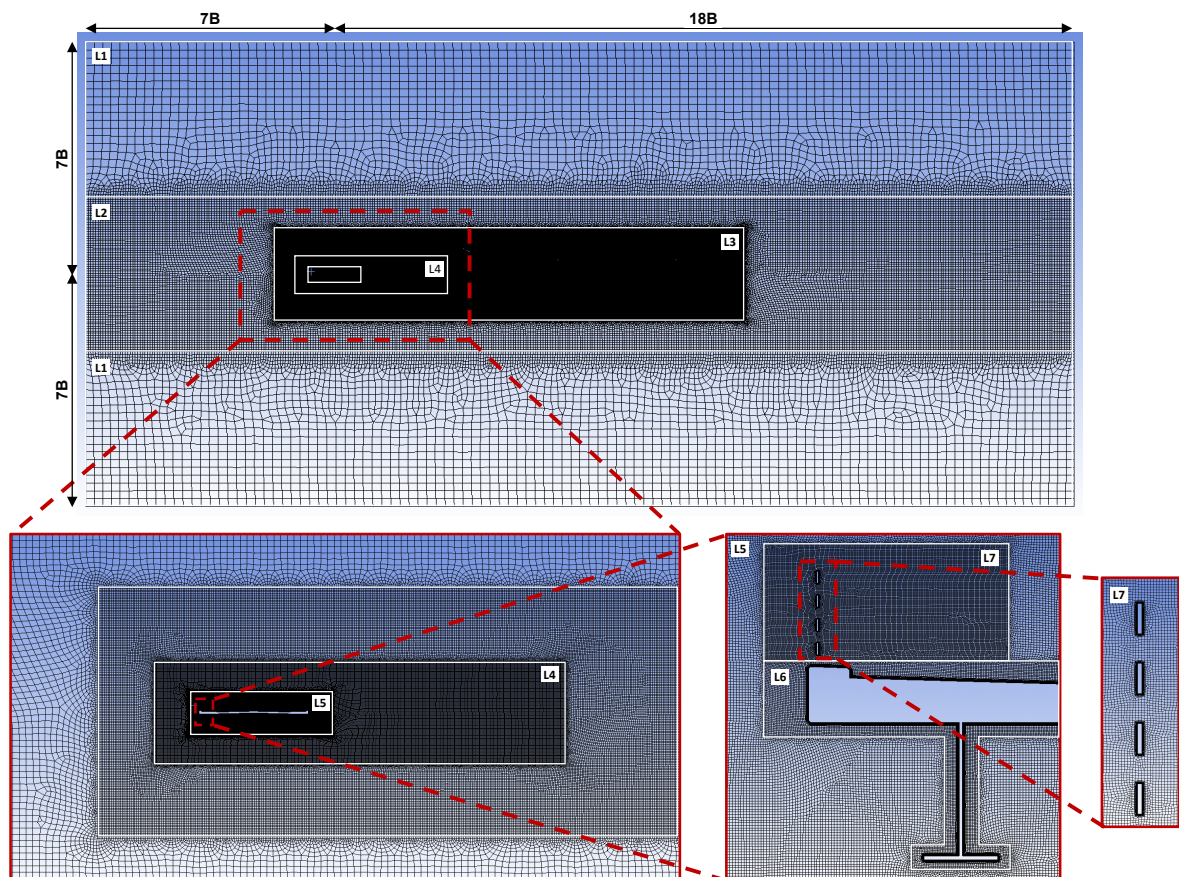


Figure 4.2. Multi-body domain division and grid details (model SM03)

Grid sensitivity was assessed to achieve a solution not dependent on the refinement level of the mesh. A summary of the results for model SM03 is given in Table 4.2. As seen, the change in all static coefficients becomes less than 3 % for the medium mesh refinement level and as such is adopted for all

simulations. It is interesting to note that the moment coefficient appears to be more sensitive to mesh density. This is probably due to the barriers and the fact that the bridge deck width (B) is large compared to the depth (D) which is likely to be true for most bridge decks.

Table 4.2. Grid sensitivity (model SM03)

Mesh refinement level	Number of cells		Drag coefficient		Lift Coefficient		Moment Coefficient	
	#	Ratio	C_D	% Δ	C_L	% Δ	C_M	% Δ
Coarse	580,434	-	0.163	-	-0.150	-	0.054	-
Medium	809,446	1.39	0.159	-3.0 %	-0.134	-10.5 %	0.047	-13.1 %
Fine	1,090,951	1.35	0.158	0.2 %	-0.131	2.2 %	0.048	-0.9 %

Table 4.3 lists the properties of each sub-grid level and the assigned mesh type. During the motion of the system, re-meshing was done only on the *Static-Deforming* sub-grid level while the motion was applied on the *Dynamic-Rigid* sub-grid levels (more details are given in Section 4.3).

Table 4.3 Sub-domain properties of the medium mesh

Mesh level	Cell size [mm]	Type of elements	Sub-grid type
L1	60	Predominantly quadrilateral	Static – Rigid
L2	20	Predominantly quadrilateral	Static – Rigid
L3	6.67	Predominantly quadrilateral	Static – Rigid
L4	2.22	Predominantly quadrilateral	Static – Deforming
L5	0.74	Predominantly quadrilateral	Dynamic – Rigid
L6	0.49	Predominantly quadrilateral	Dynamic – Rigid
L7	0.33	Predominantly quadrilateral	Dynamic – Rigid

4.1.3 Boundary Conditions

Dirichlet boundary conditions were used at all four sides of the domain with constant velocity set at the upper, lower, and upstream sides and constant pressure at the downstream side. The flow velocity for all steady simulations was set at 5 m/s, resulting in $Re \cong 2 \times 10^4$. It is worth noting that at the full scale, the Reynolds number would be of the order of 1×10^6 . The effects the Re number can have on the aerodynamic results are discussed in Section 2.3.3. The angle of attack was controlled by setting two velocity components, in the global X and Y directions as defined in Equation 4-1, respectively. The flow was uniform. All solid boundaries were modelled with no-slip conditions.

$$U_{bx} = U \cos(\alpha); U_{by} = U \sin(\alpha) \quad 4-1$$

where α is the angle of attack, U_{bx} and U_{by} are two velocity components, and U is the freestream velocity. The reason to consider non-zero angles of attack in bridge aerodynamics is twofold. Firstly, depending on the wider topology of the landscape, the bridge deck may experience wind shear and hence a downwash. Secondly, and more importantly, the galloping or flutter stability is determined by the gradients of the aerodynamic coefficients, widely known as the Den Hartog criterion (Den Hartog, 1957).

4.1.4 Turbulence Modelling

For most of the simulations, the Transition SST (TSST) turbulence model was selected with smooth flow conditions at the inlet. The TSST is a proprietary model developed for Fluent. It combines two transport equations present in the classic SST model with additional two transport equations: for intermittency and transition Reynolds number. More information on this turbulence model can be found in the Ansys Fluent User Guide (Ansys, 2021b). As already noted, the Reynolds number at the model scale for these models is of the order of 2×10^4 , hence the TSST turbulence model was explored as it was expected to provide the results of reasonably good accuracy when compared to the wind tunnel tests. Additional analyses were done using SST and $k - \varepsilon$ turbulence models and increased turbulent flow conditions in the upstream region.

4.2 Overhang Length Effects Study

The second part of this research deals with the effects of the overhang length on the VIV response of bridge decks. As shown in Section 2.2.7, a splitter plate placed in the wake of a bluff body can alter its VIV response. It is expected that the deck overhang, present in certain bridge decks, can yield similar effects. To investigate this phenomenon, two sets of deck geometries were developed: (i) deck without the barrier and (ii) deck with the barriers on two ends. Each set comprises five geometries each with a different overhang length. The two limit cases correspond to the decks found on the Clyde Bridge (deck with no overhang) and the Narrow Water Bridge (deck with a relatively large overhang). The models of both bridges have undergone wind tunnel testing as part of their design processes and the results are presented and discussed in Chapter 3. The following sections describe the methodology employed for this study while the results are presented in Chapter 6.

4.2.1 Modelling Choices

As already noted, two sets of deck shapes were developed: (i) without the barrier – denoted with S and (ii) with the barrier – denoted with SB . Each set contains five geometries all with different overhang lengths, starting from $S0/SB0$ – with no overhang (geometry $S0$ shown in Figure 4.3) to $S50/SB50$ – with the largest overhang (geometry $S50$ shown in Figure 4.4). All deck dimensions (deck depth and width, box girder, slab thickness and barrier size) were based on the deck geometries found on the Clyde Bridge (deck shape similar to $S0/SB0$) and the Narrow Water Bridge (deck shape similar to $S50/SB50$) at geometric scale of 1:30. Other geometries include $S10/SB10$, $S25/SB25$ and $S40/SB40$ and are with the overhang lengths between the two limit cases. Relevant dimensions of all geometries at the model scale are summarised in Table 4.4. The barrier detail adopted for geometries $SB0$ - $SB50$ is shown in Figure 4.5.

Table 4.4. Model-scale dimensions of the geometries studied for the overhang length effects

Geometry	B [mm]	D [mm]	S [mm]	B^* [mm]	$2S/B$ [%]	S/D
$S0$			0.00	310.0	-	-
$S10$			22.50	265.0	10	0.32
$S25$	450	70	56.25	197.5	25	0.80
$S40$			90.00	130.0	40	1.29
$S50$			112.50	85.0	50	1.61

For all dynamic models (PM and FSI) the natural oscillation frequency was set at 5 Hz. If it is assumed that at the full scale, a prototype bridge will have the first natural frequency in bending of approximately 1 Hz, the velocity scale will be equal to 6. This is a reasonable value when compared to the ones adopted in the wind tunnel tests described in Chapter 3. The structural damping adopted for all FSI models was 3% logarithmic decrement, again an assumption supported by the experience of practice.

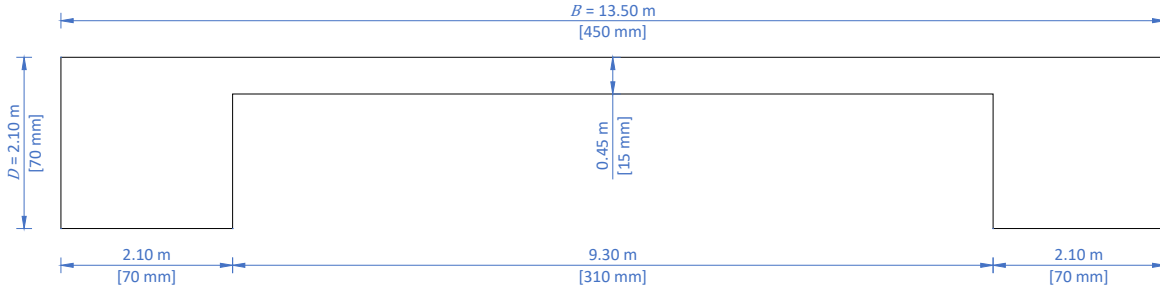


Figure 4.3. Geometry S0 at full scale and [model scale]

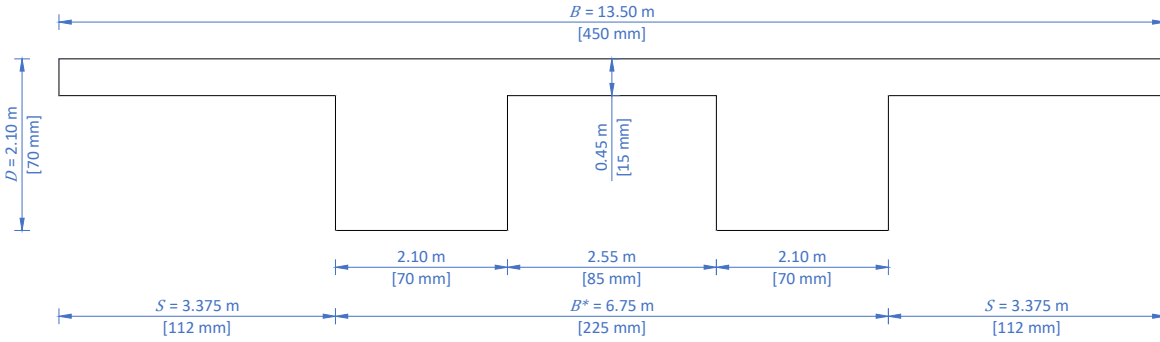


Figure 4.4. Geometry S50 at full scale and [model scale]

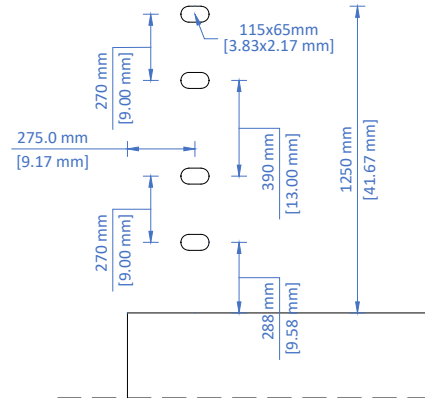


Figure 4.5. Barrier detail for SB0-SB50 geometries; dimensions in full scale and [model scale]

4.2.2 Meshing Methodology and Grid Sensitivity

A meshing technique based on the principles presented in Section 4.1.2 with a number of evolutionary changes was adopted here. Since it is expected that the most of aerodynamically important processes occur in the deck wake (see Section 2.2.7) more refined mesh regions were needed in this part of the domain. Furthermore, the geometry of the barrier is constant (for all SB geometries) hence the region in the vicinity of it was modelled with fewer divisions in the domain.

The domain was divided into five sub-grids and for each, the cell size and cell type were controlled. An example of this approach applied to geometry SB25 is shown in Figure 4.6. The main principles defined

previously were applied here, i.e., the domain was divided so that only a part of the mesh undergoes re-meshing due to the motion. Here, the deforming part of the mesh (sub-grid L4) is meshed using triangular elements, as opposed to quadrilateral elements used previously. It is expected that the re-meshing process is faster for the mesh dominated by the triangular elements, however, the trade-off is in the number of cells. A mesh constructed using triangular elements contains more cells than the one made of quadrilateral elements. Another advantage of this approach is that the part of the grid employed for modelling the wake (sub-grid L3) is generated using mapped quadrilateral mesh, which allows for faster grid generation. Having a geometry which is to a certain level simpler than that of the NSB model (described in Section 4.1.2), allowed for exploring this meshing alternative.

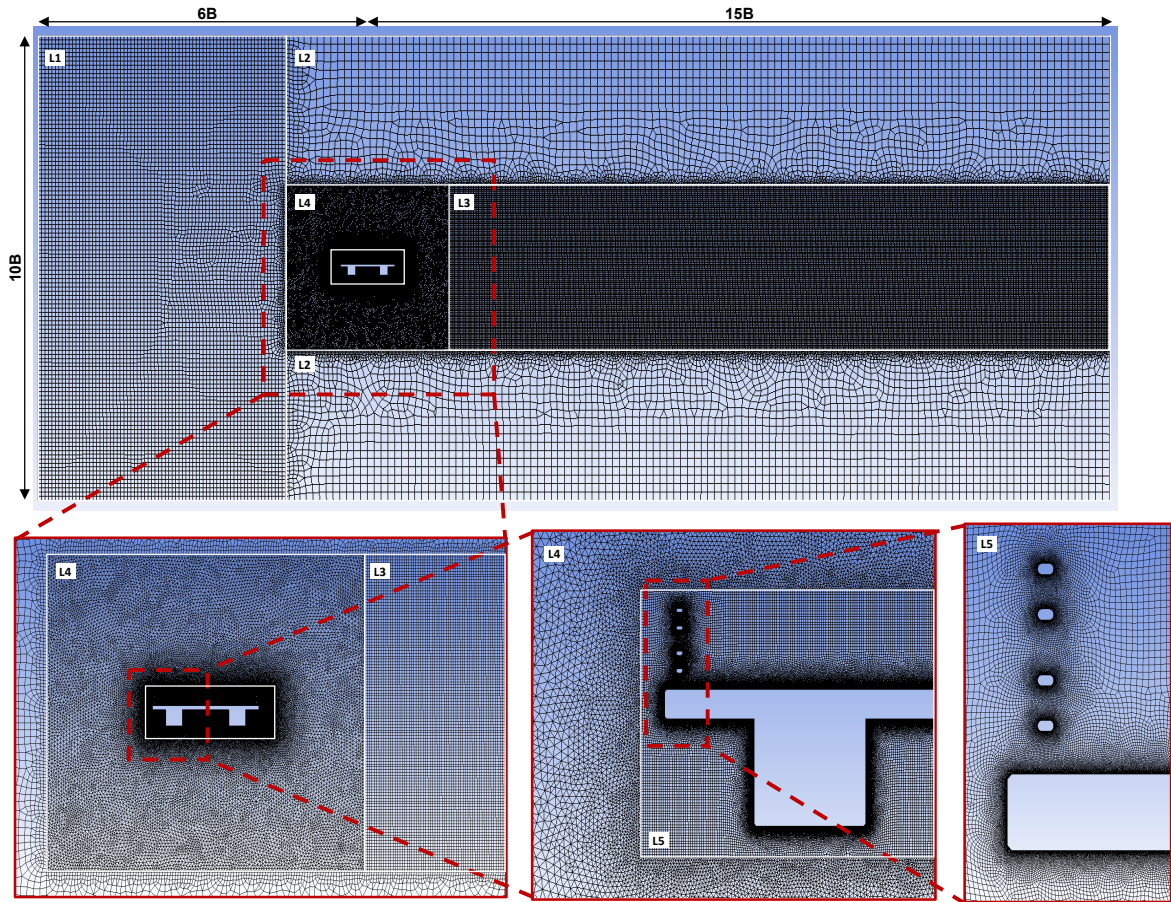


Figure 4.6. Domain division and grid details (geometry SB25)

A mesh sensitivity study was undertaken on geometry SB0 since it can be considered as the one causing the most complex flow features such as large pressure gradients and jetting. This is due to the presence of the barrier on the upwind edge coupled with the largest blockage area (due to the lack of the overhang). The mesh sensitivity was done based on the static coefficients and the results are reported in Table 4.5. It can be seen that all force coefficients exhibit a change smaller than 1% when the medium mesh is further refined. Thus, it was chosen as the appropriate one for the study.

Table 4.6 lists the properties of each sub-grid level and the assigned mesh type. During the motion of the system, re-meshing was done only on the *Static-Deforming* sub-grid level while the motion was applied on the *Dynamic-Rigid* sub-grid levels (more details are given in Section 4.3).

Table 4.5. Grid sensitivity (model SB0)

Mesh refinement level	Number of cells		Drag coefficient		Lift Coefficient		Moment Coefficient	
	#	Ratio	C_D	% Δ	C_L	% Δ	C_M	% Δ
Coarse	350,960	-	0.169	-	-0.212	-	0.034	-
Medium	498,804	1.42	0.169	-2.4%	-0.207	0.1%	0.034	1.6%
Fine	690,293	1.38	0.169	-0.8%	-0.206	0.0%	0.034	0.6%

Table 4.6. Sub-domain properties of the medium mesh

Mesh level	Cell size [mm]	Type of elements	Sub-grid type
L1	28	Predominantly quadrilateral	Static – Rigid
L2	56	Predominantly quadrilateral	Static – Rigid
L3	9.8	Only quadrilateral (mapped)	Static – Rigid
L4	11.2	Only triangular	Static – Deforming
L5	1.4	Predominantly quadrilateral	Dynamic – Rigid

4.2.3 Boundary Conditions

Dirichlet boundary conditions were used at all four sides of the domain with constant velocity set at the upper, lower, and upstream sides and constant pressure at the downstream side. All simulations were done under a purely horizontal flow ($U_{bx} = U$). The values of the flow velocity ranged from 2.2 m/s to 4.5 m/s, resulting in the Reynolds number of approximately 1.6×10^4 , with respect to the deck depth. The flow was uniform. All solid boundaries were modelled with no-slip conditions.

To study the effects of an initial *kick* on the VIV response of the deck, several FSI simulations were executed with the flows initialised using prescribed motion models. The results from these simulations showed (see Section 6.2.3) that the magnitude of the kick controls whether the deck motion will reach the limit cycle oscillations or not.

4.2.4 Turbulence Modelling

All simulations were performed using the SST turbulence model with smooth flow conditions at the inlet. Although the more refined Transition SST turbulence model offers the possibility of more accurate results at lower Reynolds numbers than the SST, due to increased computational requirements¹ the latter one was chosen as the benefit was small. Furthermore, in Chapter 5 it is shown that for higher Re numbers, the difference between the two turbulence models is negligible.

4.3 Fluid-Structure Interaction Model

The fluid-structure interaction was modelled by integrating the single degree of freedom dynamic model (Figure 4.7) into Ansys Fluent. The model is defined by the equation of motion (Equation 4-2).

$$m \ddot{y} + c \dot{y} + k y = F_L \quad 4-2$$

where y is the vertical displacement of the body, and one and two overdots represent the velocity and acceleration of the body, respectively. The system mass, damping coefficient and stiffness are defined with m , c , and k , respectively. The excitation force is denoted as F_L , which in the FSI model is the lift force.

¹ Refer to Section 4.1.5 for the description of the TSST turbulence model.

As the built-in, six-degree of freedom, free body motion feature in Fluent does not allow damping to be directly controlled, the motion was accounted for with an additional lumped parameter model. The lift force was obtained by integrating the pressure over the surfaces of the deck and two barriers. The equation of motion was solved for the vertical velocity of the deck using the *Runge-Kutta 4th Order Method*. The rigid motion was then applied to the deck and barriers, and the *Dynamic-Rigid* sub-grid levels (see Table 4.3 and Table 4.6).

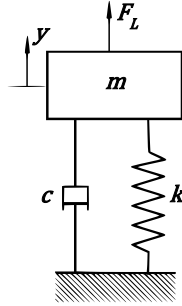


Figure 4.7. Single degree of freedom dynamic model

The dynamic properties (mass, damping and stiffness) were transformed to *per meter* of the deck spanwise length and as such defined within the User Defined Function (UDF). The UDFs were developed in such a way as to support parallel multi-core simulations (see Appendix B for details).

The deck depth D and the system's natural frequency f_n were used for the reduction of all variables in this thesis. Reduced variables are defined in Equation 4-3, where y_r , U_r and t_r are reduced deck displacement, reduced flow velocity and reduced time, respectively.

$$y_r = \frac{y}{D} ; U_r = \frac{U}{f_n D} ; t_r = t f_n \quad 4-3$$

4.4 Prescribed Motion Model and the Energy Balance Approach

Although free-to-oscillate simulations provide valuable information on the maximum displacement amplitude and lock-in region under vortex induced vibrations, the simulations must be performed in series. That is, due to the nature of the lock-in phenomenon, the analyses must be initiated at a velocity lower than the expected coincidence velocity. Because of this, the time required to reach the maximum displacement amplitude can be considerable. Many studies employed prescribed motion simulations to study the VIV of structures submerged in the moving fluid, see for example Jiao & Wu, (2018); Paidoussis et al., (2014); Staubli, (1983). The two main advantages of these simulations are that they can be run simultaneously and that the important properties, such as displacement amplitude and oscillation frequency, are defined. With the results for a range of displacement amplitudes, and flow velocities, the structural system can be defined separately from the simulations. The energy approach is employed to relate the defined structural system to the simulated data.

As the excited body starts to oscillate, the oscillation amplitude increases until it reaches an asymptotic value. When this happens, the oscillating body has entered limit cycle oscillations (LCO) which is characterised by sustained harmonic oscillations without decay or growth i.e., the effective system damping tends to zero. Two things should be noted here: (i) because of the asymptotic nature of logarithmic decay,

the oscillation amplitude never becomes fully constant, it only reaches an asymptotic value and (ii) to reach the LCO from an initial state, the oscillating body can experience either an increase or decrease in the amplitude, depending on where the initial conditions are with respect to the LCO amplitude. Zero effective damping means that the net energy of the dynamic system during one cycle equals zero. That is, during one oscillation cycle, the energy dissipated by the structure and the work done on the body by the fluid forces are equal.

When studying the LCO response of an oscillating body, depending on the structural damping, the time needed for the oscillations to reach the LCO ranges between tens of seconds to minutes. For tests done in wind tunnels, that time is a fraction of what is needed for the overall testing campaign. However, when performing a CFD simulation, with the analysis time of the order of minutes, or even tens of seconds, the required wall-clock time plays a limiting factor in the number of simulations that can be done feasibly. Employing prescribed motion models allows us to avoid running simulations during this transient period because the amplitude is prescribed. Here, the prescribed motion model is used to evaluate the work done on the bridge deck by the vortex-induced forces. The deck motion is controlled by prescribing the velocity at each time-step as defined in Equation 4-4 and the lift force recorded.

$$\dot{y} = 2 \pi Y f \cos(2 \pi t f) \quad 4-4$$

where Y is the prescribed displacement amplitude (varied over a range of values), f is the prescribed oscillation frequency (kept constant), and t is time. The work done by the fluid forces on the deck is then compared to the dissipated energy that originates, from the structural damping of the dynamic system.

The total energy of a damped harmonic oscillator is defined by a sum of the potential and the kinetic energy. Equations 4-5, 4-6, and 4-7 define potential, kinetic, and total energy, respectively.

$$E_p = \frac{1}{2} m Y^2 \omega_0^2 e^{-\gamma t} \cos^2(\omega_d t + \phi) \quad 4-5$$

$$E_K = \frac{1}{8} m Y^2 e^{-\gamma t} [\gamma \cos(\omega_d t + \phi) + 2 \omega_d \sin(\omega_d t + \phi)]^2 \quad 4-6$$

$$E_{TOT} = \frac{1}{2} m Y^2 \omega_0^2 e^{-\gamma t} [1 + \frac{\gamma}{2 \omega_0} \cos(2 \omega_d t - \tan^{-1}(2 \omega_d / \gamma))] \quad 4-7$$

where $\gamma = c/m$ is the damping-mass ratio and ω_d is the natural frequency of the damped system. For lightly damped systems, $\gamma \ll \omega_0$, the cosine in Equation 4-7 can be neglected. This reduces the total energy to Equation 4-8.

$$E_{TOT}(t) \approx \frac{1}{2} m Y^2 \omega_0^2 e^{-\gamma t} \quad 4-8$$

Introducing the quality factor (Q factor) as $Q = \omega_d / \gamma \approx \omega_0 / \gamma$ and evaluating the energy change during one oscillation period, $T = 2\pi\omega_0$, the total energy loss during one oscillation cycle is given by Equation 4-9.

$$\Delta E_S = \frac{1}{2} m Y^2 \omega_0^2 (1 - e^{-\frac{2\pi}{Q}}) \quad 4-9$$

The energy brought into the system by the work done on the bridge deck by the vortex-shedding forces during one oscillation cycle can be obtained by integrating the lift power $F_L(t) \dot{y}(t)$ over one oscillation period T , as given in Equation 4-10.

$$\Delta E_F = \int_0^T F_L(t) \dot{y}(t) dt \quad 4-10$$

where $\dot{y}$ and F_L are the deck velocity and fluid forces exerted on the deck in the vertical direction, respectively.

Finally, the total net energy in one oscillation cycle is given in Equation 4-11, it comprises the energy brought in by the fluid (ΔE_F) and the energy taken away by the structural damper (ΔE_S).

$$\Delta E_{NET} = \Delta E_F - \Delta E_S \quad 4-11$$

For an easier comparison and discussion, the energy is non-dimensionalised as in Equation 4-12.

$$\Delta E^* = \frac{\Delta E}{0.5 U \rho D B Y f} \quad 4-12$$

where U is the freestream wind velocity, ρ is the air density, D and B are the depth and width of the bridge deck and Y and f are the oscillation amplitude and frequency, respectively.

Here it is assumed that at lock-in (which is the region of primary interest), the ratio between the shedding frequency and deck oscillation frequency is unity. Many papers confirm that this is the case for high mass ratio systems, see for example Williamson & Govardhan, (2004). It was also assumed that the fluid-added mass is negligible when compared to the mass of the deck. Furthermore, equations 4-5 to 4-9 assume that the motion of the body is purely sinusoidal and that higher frequency oscillations are negligible. This is a common assumption and is supported by the results presented in Section 5.2.

Chapter 5

CFD as an Early Aerodynamic Design Tool

During the bridge design process, wind tunnel tests are often undertaken as a way of assessing its aerodynamic behaviour. The most common wind tunnel results comprise the static force coefficients, Strouhal number estimate and the free vibration response (see examples in Chapter 3). As discussed in Section 2.5, apart from a couple of exceptions, CFD has not been used as a design tool in bridge engineering but rather for research purposes. Furthermore, the analyses mostly comprised streamlined deck sections. This chapter explores the peculiarities involved in using CFD as a preliminary design tool for bluff bridge decks in an industrial context. For this purpose, a CFD model of the Northern Spire Bridge deck was developed (see Section 4.1) and results from the wind tunnel tests (see Section 3.1) were used for benchmarking different modelling choices. Section 5.1 outlines the findings related to the fixed models while the results from the free-to-oscillate and prescribed motion models are given in Section 5.2 and Section 5.3, respectively. A similar modelling approach was applied to the remaining two case studies and the results are reported in Section 5.4, while Section 5.5 includes an overview of the chapter.

It is noted that the results shown in this chapter are partially presented in Duranovic et al., (2022a, 2022b, 2024).

5.1 Fixed Model

In this section, the results from the simulations on the rigid models (SM01-SM42 in Table 4.1) are presented and discussed. Three sets of simulations were done in steady state conditions and one in transient.

Firstly, the influence of the girder and the barriers on the static coefficients was investigated to assess how different numbers of openings in the barriers affect the aerodynamic profile of the deck while keeping the barrier porosity constant. Secondly, multi-dimension models were considered and the three-dimensionality of the barrier was examined. The results from these, more computationally expensive models, were compared to the results from the 2D models. The third set of simulations included a study on different turbulence models available in Ansys Fluent. These included the $k - \varepsilon$ model, SST model and Transition SST model. The final simulation on the static deck model was done under transient conditions which provided an estimate of the Strouhal number.

5.1.1 Approximation of Geometry as 2D Profile

As an initial simulation, the full geometry shown in Figure 3.2 is simplified by removing the girders and barriers, resulting in the section shown in Figure 5.1 (model SM01). Additional features (i.e., barriers above and girders below) are then added to explore the effect of geometric simplification.



Figure 5.1. Bare deck geometry at model scale [mm] – model SM01

The simulation done with this, much simplified geometry, shortens the meshing time and computational requirements dramatically. It in turn leads to shorter solution times and exhibits fewer convergence issues. Static aerodynamic coefficients are shown and compared to wind tunnel results in Figure 5.2a. It should be noted that the wind tunnel results are for the full deck section (deck, girder, and barriers). Even though there is a large difference in the geometry of the two models, a relatively good agreement between the wind tunnel lift and the CFD simulated lift is obtained, in terms of both the slope around 0° angle of attack (which is important for stability) and the overall magnitude. As expected, the drag coefficient shows a large discrepancy in magnitude when compared to wind tunnel results. There are two reasons for this. Firstly, the drag for both geometries is normalised with the bridge width (B) which is the same for the two cases. The total area projected in the X -direction for the bare deck is 71% smaller than in the case of the deck with the girders. This causes a factor of 3.4. The second reason is that the barrier and girder will generate a proportionately larger wake. Despite the large discrepancy in drag and moment coefficients, this model can be useful for an initial estimation of the lift coefficient behaviour with varying angles of attack.

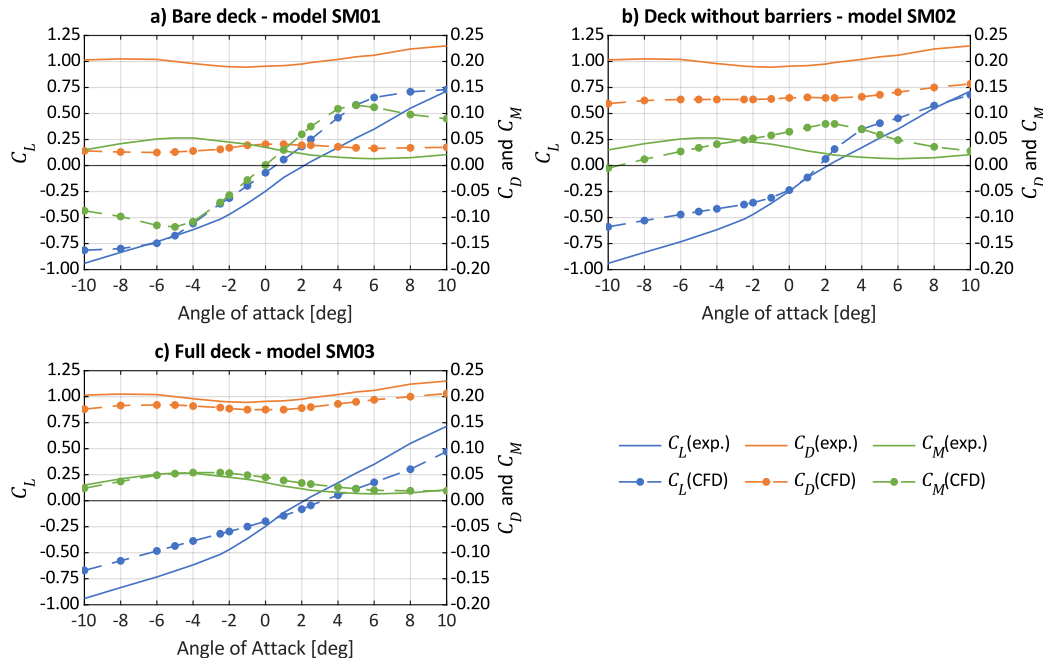


Figure 5.2. Static force coefficients for three deck models compared to the wind tunnel tests (full deck)

The girders are geometrically simple, so including them does not dramatically increase the mesh size (an increase of about 2% in the number of cells). Figure 5.2b shows static coefficients for the deck section without the barriers (model SM02). As with the bare deck analysis, values of drag coefficient are lower than their experimental counterparts, although the discrepancy is much smaller. Considering that most meshing difficulties occur in the vicinity of the barriers this can be regarded as another model worth exploring before moving to the full deck analysis.

Finally, a representation of the barriers was added with four openings (model SM03). The results of this simulation are seen in Figure 5.2c. The drag and moment coefficients now follow the trend of the experimental data well, although with a slightly lower magnitude. The lift, however, is not significantly better

than that obtained for the bare deck and in terms of both gradient and magnitude around $\alpha = 0^\circ$, it is arguably worse.

Including a representation of the barriers is necessary to achieve a reasonable estimate for drag and lift. However, the barriers in the wind tunnel model have a 3D pattern as can be seen in Figure 3.20 in Section 3.4, which is itself an approximation of the full-scale barriers. The wind tunnel barriers are chosen to have an equivalent blockage ratio to the full-scale design. In the 2D approximation, the barrier is modelled as a solid fence with a number of openings.

The size of the openings is varied to assess whether the number of openings in the barrier makes a difference for any of the static coefficients when compared to the no-barrier case. The findings are presented in Figure 5.3 where the change in error with the introduction of the barriers is compared for six different configurations. As expected, the drag coefficient is least susceptible to the varying number of barrier openings since the total area over which the drag force is integrated is kept constant. It is also noted that the introduction of the barriers positively affected the drag coefficient estimate by decreasing the error for all angles of attack. This too suggests that by changing the number of openings, we can only introduce a limited benefit in the accuracy of the drag coefficient.

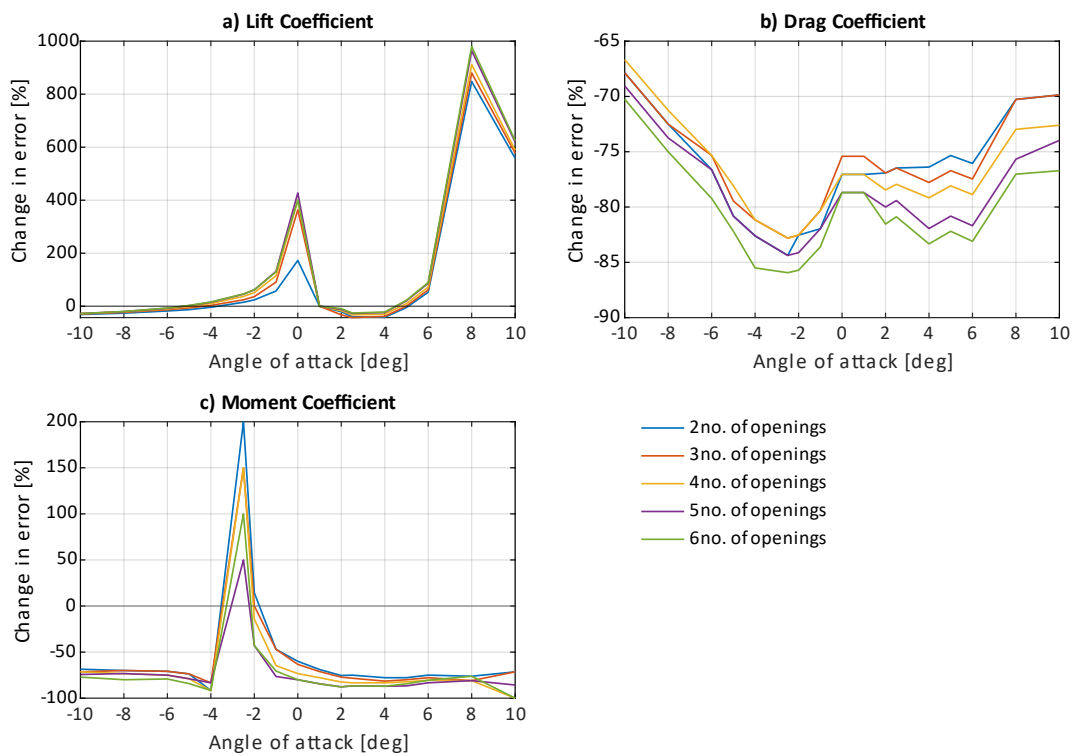


Figure 5.3. Effect of the number of barrier openings as a change in error with respect to the no-barrier case, for a) lift, b) drag and c) moment coefficient (models SM11-SM14)

Varying the barrier configuration has a dramatic effect on the flow adjacent to the top of the bridge deck due to jetting through the lower opening. Thus, the lift and the moment coefficients exhibit significant variations for the five barrier configurations considered. The models with fewer barrier openings produced better lift coefficients when compared to the wind tunnel data. While the opposite holds for the moment coefficients. Thus, there is no clear “best option” when it comes to the number of openings in the barrier. The model with four openings has been chosen as a compromise configuration as it also has the benefit of

being geometrically similar to the wind tunnel model which has between three and four openings over the depth (see Figure 3.20). This brings a valid question of how important it is to accurately model the barrier geometry in wind tunnel tests. Whether a simple matching of the porosity level is sufficient for the tests to yield a behaviour comparable to that of the full-scale bridge, or a more detailed barrier representation is required.

5.1.2 3-Dimensional Models

As already noted, the barrier, both in the wind tunnel model and on the full-scale bridge, has a 3-dimensional geometry. Furthermore, on the underside of the bridge deck, there are cross girders which are streamwise. Although the treatment of turbulence is still based on an isotropic closure, the small-scale features of the barrier and the secondary flows that would be expected in the junction of the cross girders with the deck may affect the far wake and hence the overall aerodynamic behaviour. To assess the impact of the 2D assumption, two 3D domains were considered. The first, which is referred to as the *2.5D model* (see Figure 5.4), represents a narrow strip of the deck with periodic boundary conditions on the two streamwise domain boundaries across the deck width (model SM21). The geometry is still substantially of a 2D character. The strip width is such that it captures the periodic shape of the barrier, but the cross girders are ignored.

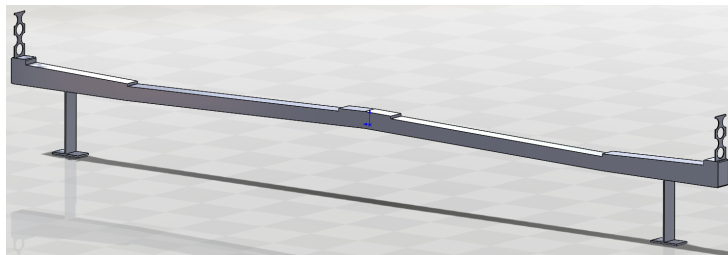


Figure 5.4. 2.5D model (model SM21)

The second model is referred to as the *3D model* (see Figure 5.5) and it is represented by a strip of such width that the half distance between two cross girders is covered (model SM22). Two planes perpendicular to the deck are set as symmetry boundary conditions. With this approach, the domain for the simulation was four times wider than the initially meshed one, see Figure 5.6a. As seen in the figures, the barrier for the 3D models closely mimics the wind tunnel model.

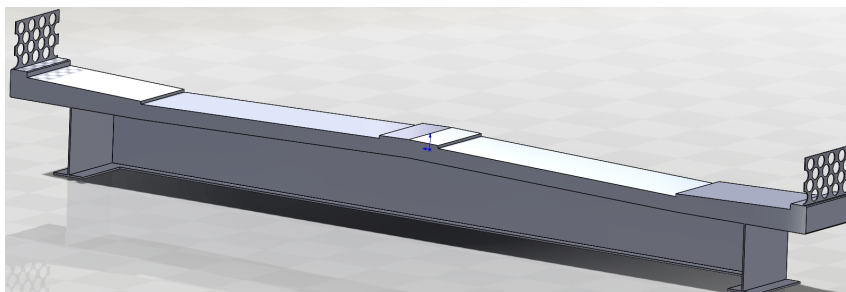


Figure 5.5. 3D model (model SM22)

For both models, a polyhedral mesh is used as it decreases the number of cells required to achieve an acceptable solution, see Figure 5.6b. The mesh sensitivity analysis for this mesh has not been done due to high computational requirements. However, the cell size chosen for these models is comparable to the cell sizes employed in the 2D domains.

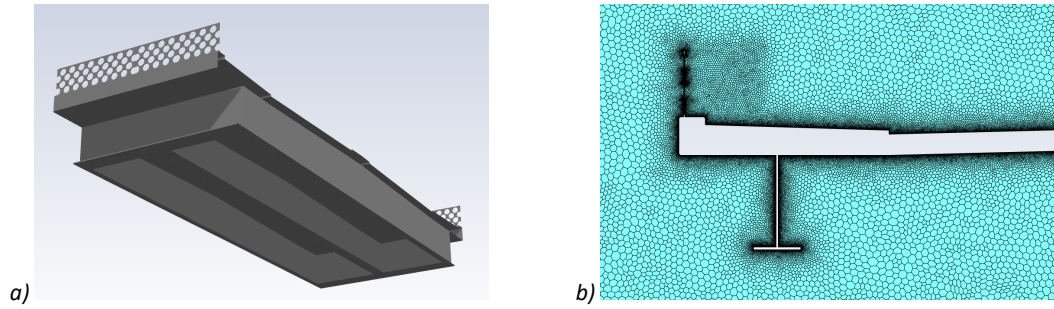


Figure 5.6. (a) The 3D model with applied double symmetry boundary conditions (model SM22); (b) polyhedral mesh available in Fluent – model SM22 and in a similar way developed for model SM21

Simulations presented here employed the SST turbulence model only (as shown in the Simulation Matrix in Table 4.1).

The results of the 2.5D model (model SM21) and the 3D model (model SM22) are shown in Figure 5.7 and Figure 5.8, respectively. Both are compared to the wind tunnel data and the results from the 2D analysis (model SM32).

There is not a substantial difference between multi-dimensional models (2.5D and 3D) and 2D models suggesting that the extra computational expense is not justified; each simulation took approximately 30 CPU hours using 120 CPU cores. However, it should be noted that turbulence is treated with an isotropic model, so the highly 3-dimensional turbulent wake will be poorly captured. A full DDES or LES model may yield improved results (Y. Zhang et al., 2021), but the computational effort will increase dramatically reducing the number of configurations that can be investigated. It is also noted that should a 3D model be of larger spanwise length than the ones covered here, the findings could be different. However, the purpose of this task was to investigate whether a limited extrusion to the third dimension could yield results better than those obtained from the 2D analyses in the context of moderate computational effort.

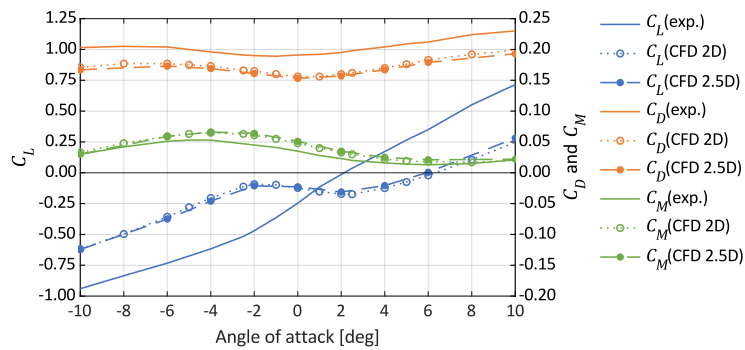


Figure 5.7. Results of the 2.5D model data (SM21) compared to the wind tunnel and 2D data (SM32)

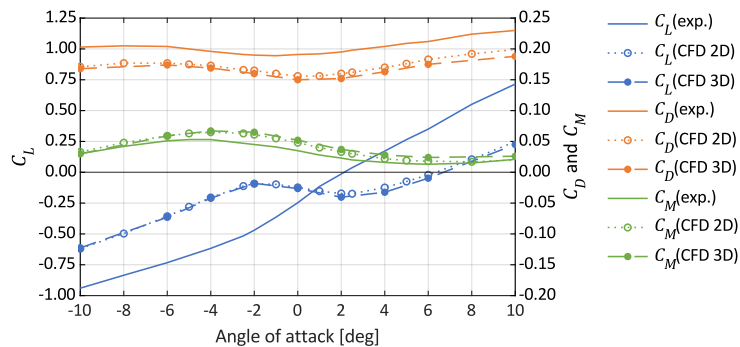


Figure 5.8. Results of the 3D model (SM22) compared to the wind tunnel and 2D data (SM32)

To understand why the increased geometric complexity does not yield improved aerodynamic coefficients, the flow field of the near wake has been visualised. Two shear layers, coming from the top and the bottom of the deck, encapsulate the flow features originating from the barrier and limit their effect to the region just downstream of the deck, while the far wake is unaffected. This is substantiated in Figure 5.9 where the Y -vorticity contour for the plane at 2.6 m from the deck centroid and the Y -velocity streamlines are shown.

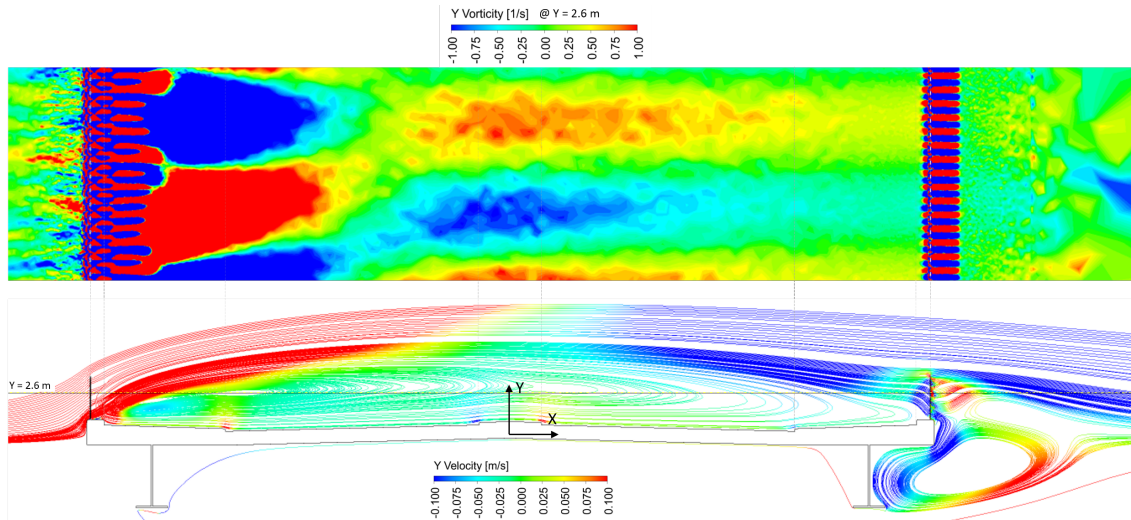


Figure 5.9. 3D model (model SM22); Y -Vorticity contour (top) and Y -velocity streamlines (bottom)

5.1.3 Turbulence Modelling

The simulation approach taken is steady RANS as this is substantially less computationally expensive than LES or DDES. However, it is well known that the choice of the turbulence model in RANS may significantly impact the results. The models considered here are:-

- The $k - \varepsilon$ – model SM31
- SST – model SM32
- Transition SST (TSST) – model SM03

TSST is a proprietary model developed by Fluent, see Ansys, (2021b) for more details. It should be noted that the Reynolds number of the wind tunnel (based on deck depth, D) is in the transitional range at the lower velocities ($\sim 10^4$), but the Reynolds number regime for the full-scale bridge will be fully turbulent ($\sim 10^6$). Low Reynolds wall treatment is employed in all simulations. Figure 5.10 shows static coefficients for a range of angles of attack for different turbulence models employed and compared to the wind tunnel data. As seen, the TSST model yields the closest comparison to the wind tunnel data, especially for the lift coefficient.

While the SST model is developed for fully turbulent flows ($Re > 10^5$), the Transition SST model outperforms it when the flow is in the transition region, between the laminar and fully turbulent conditions. This is documented by Lodefier et al., (2004).

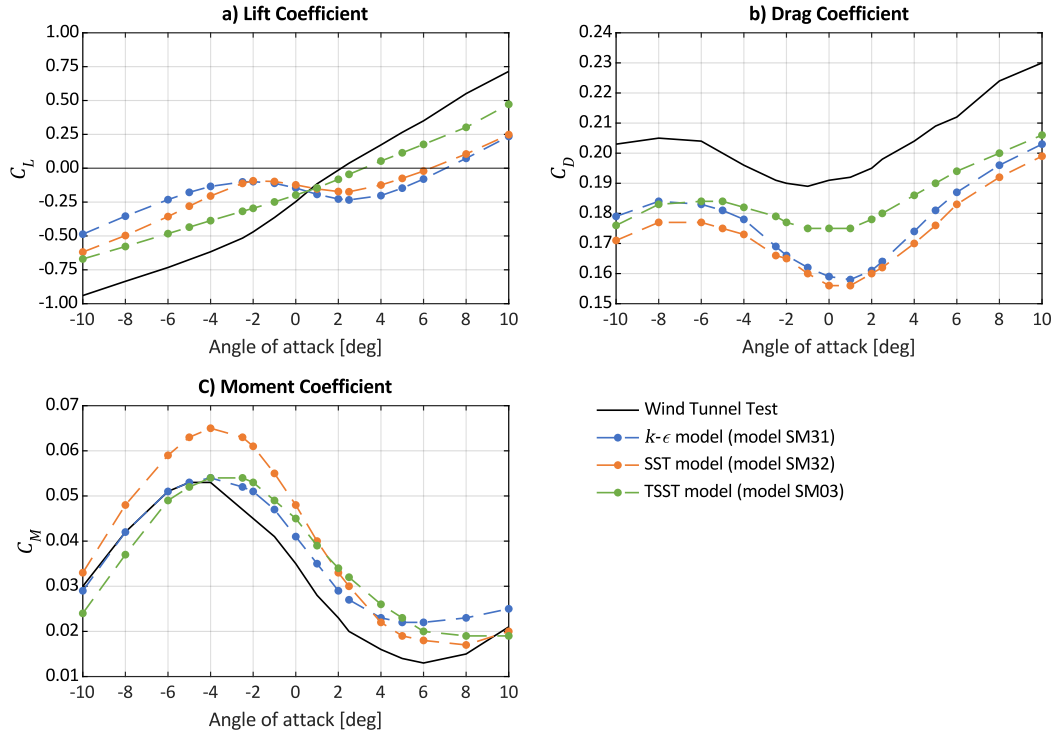


Figure 5.10. Comparison of turbulence models for a) lift, b) drag and c) moment coefficient

Bridges operate in the atmospheric boundary layer, and so the upstream airflow typically exhibits high levels of turbulence. In the wind tunnel, this is modelled with turbulence generators. Figure 5.11 shows the static coefficients obtained with the TSST turbulence model for a turbulence intensity (I_u) of 10% at the inlet, which resulted in the I_u of approx. 4% just upwind from the deck (model SM41). The dissipation of turbulent kinetic energy in the turbulence model tends to reduce the freestream turbulence levels more rapidly than is experienced in practice. In the same figure, static coefficients for the smooth flow ($I_u = 0\%$) at the inlet are shown (model SM03), and a clear difference between the two models is noticeable. Both the magnitude and the slope of the curves show better correspondence with the wind tunnel data for a turbulent inflow.

Similar observations can be made for the simulations of the 3D SST model with increased turbulence intensity (model SM52) shown in Figure 5.12. Here the change in lift slope around 0° angle of attack is even more noticeable and important. Having the slope of the lift coefficient negative around 0° angle of attack would lead one to believe that the deck may be susceptible to instability.

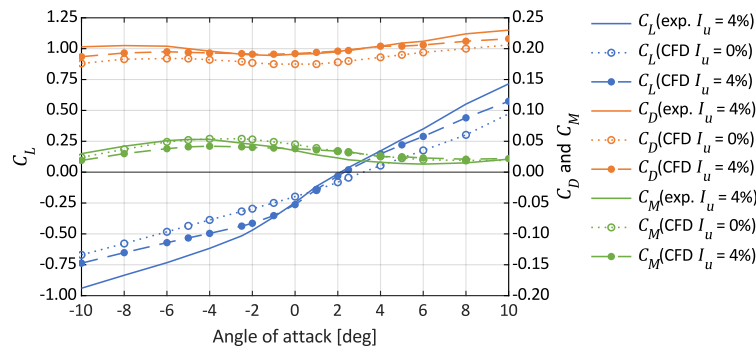


Figure 5.11. Results of the 2D models: with high inflow turbulence intensity (model SM41), and with smooth inflow conditions (model SM03) compared to the wind tunnel data

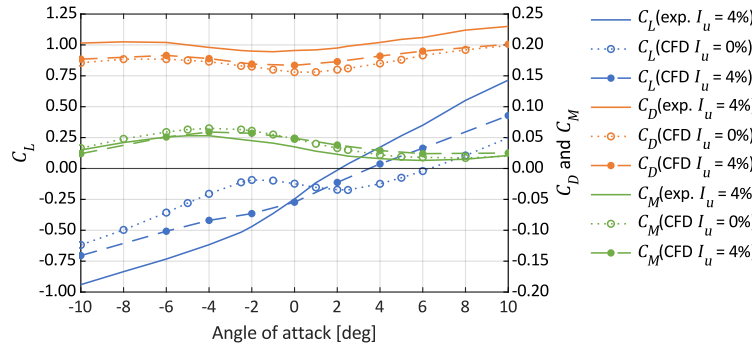


Figure 5.12. Results of the 3D models: with high inflow turbulence intensity (model SM42), and with smooth inflow conditions (model SM22) compared to the wind tunnel data

5.1.4 Transient Analysis

The mesh and geometry of the Fixed Model (model SM03) which was used to obtain the aerodynamic coefficients using steady flow, can now be used to easily yield basic information about the vortex shedding by conducting transient simulations. The initial time-step size was gradually decreased until a constant value of the Strouhal number was reached. Figure 5.13 shows the time history of the static force coefficients obtained from model T01 where the time is reduced with the freestream flow velocity and the deck depth (D). The frequency analysis of the lift coefficient signal is shown in Figure 5.14 with the Strouhal number defined as in Equation 2-9. The transient analysis of the fixed deck model estimates St at 0.065, which compares well with that obtained from the wind tunnel data (0.075).

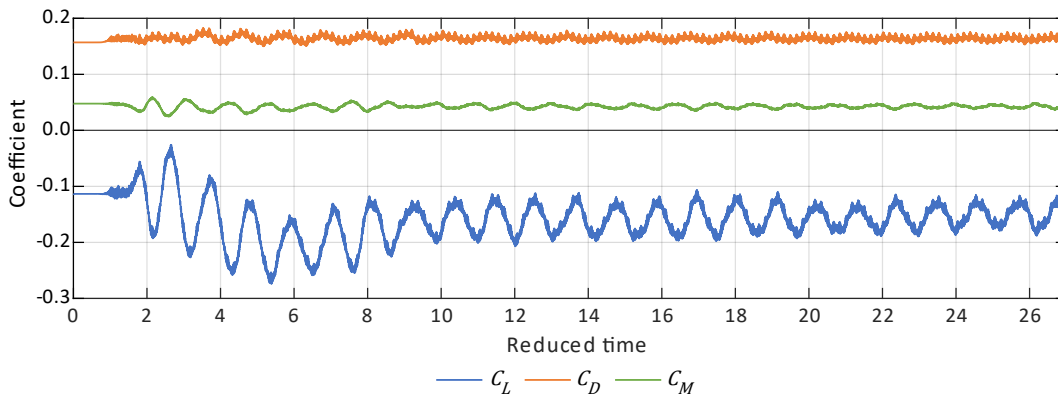


Figure 5.13. Time history of the static coefficients (model TM01)

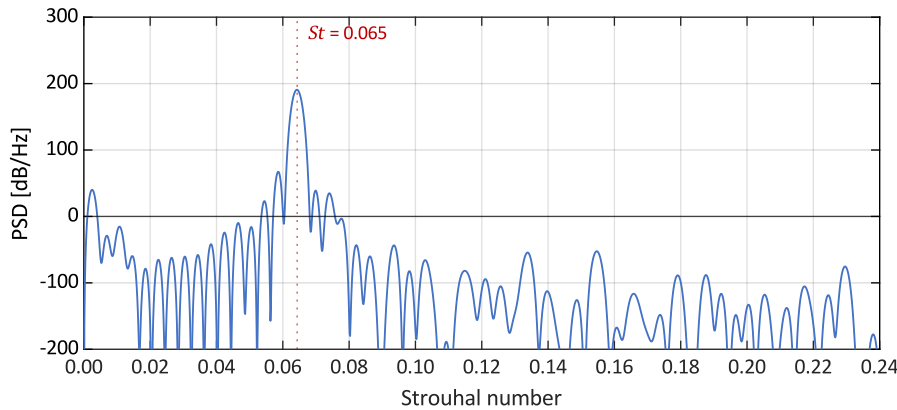


Figure 5.14. Frequency domain analysis of the lift coefficient from the transient simulation (model TM01)

5.2 Free-to-Oscillate Model (FSI)

The mesh and geometry identified with the Fixed Model are now used as the basis for a Free-to-Oscillate model, where the bridge deck is supported on a flexible structure as described in Section 4.3. All the variables are reduced as described in Section 4.3.

Two turbulence models were used for the FSI models: SST (model DM01) and TSST (model DM02). Figure 5.15 shows the deck response obtained from these simulations and compared to the wind tunnel data. The SST model provided a good agreement in the shape of the lock-in region. The Transition SST turbulence model performed considerably better than SST in determining the velocity at which the lock-in occurs. The vibrations start to be noticeable at a reduced velocity of approximately 13.5 which is in good agreement with what is reported in the wind tunnel data. However, with increased flow velocity and for the value of Reynold's number of approximately 13×10^3 , the oscillation amplitude starts to diverge from the wind tunnel values. Clearly, at these large amplitudes, the simulation is either overestimating the excitation or underestimating the dissipation in the flow. This may be due to the 2-dimensional assumption compounded by the simple isotropic turbulence model, as at large amplitudes the spanwise correlation will become more important.

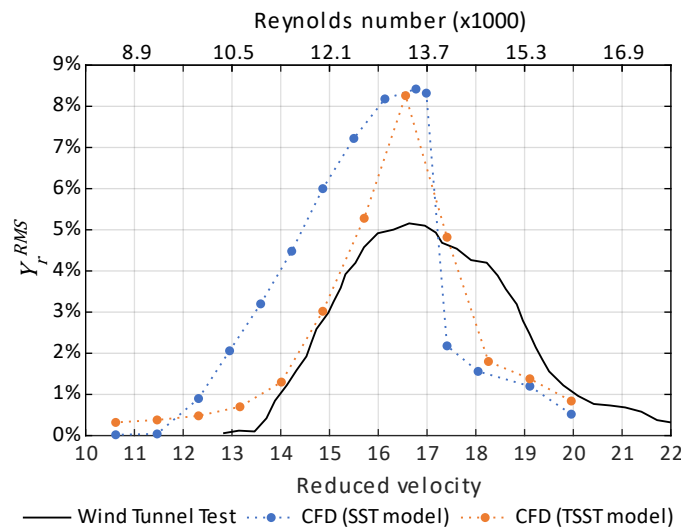


Figure 5.15. Deck response with SST (model DM01) and TSST (model DM02) turbulence models compared to the wind tunnel data

The lock-in region for these two simulations is further examined in Figure 5.16. It shows reduced shedding frequency (f_{vs}/f_n) and the Strouhal law for this deck section. The Strouhal law, as defined in Equation 2-9, is based on $St = 0.065$, as obtained from the transient model (model TM01). A noticeable difference is observed between the two models. The SST model predicts that the shedding frequency locks to the system's natural frequency before the coincidence, while TSST shows behaviour more similar to what is observed in the wind tunnel with the lock-in starting at coincidence between the Strouhal and the natural frequencies. Figure 5.17 shows the spectrogram of the deck displacement obtained in the wind tunnel. For consistency, all the values are reduced as previously described and the spectrogram in Figure 5.17 is shown for the full range of velocities tested in the experiment. It can be seen that the lock-in occurs at the reduced velocity of approximately 13.5 which is more or less the coincidence between the Strouhal frequency and the natural frequency. The lock-in range, as indicated by the suppression of the Strouhal frequency, extends to a

reduced velocity of approximately 20. Note that immediately before coincidence, the vortex shedding frequency is distinct from the natural frequency. The data from the Transition SST (model DM02) in Figure 5.16 shows similar behaviour in both pre- and post-coincidence. The SST results on the other hand show strong pre-coincidence lock-in which is not supported by the wind tunnel data. Having the wind tunnel data better match the results produced using the TSST turbulence model could possibly indicate a strong Re number effect, an issue discussed in Section 2.3.3.

In Figure 5.17, at a reduced velocity of 7.5, there appears to be another small lock-in zone. As the Strouhal frequency at this velocity is exactly half the natural frequency, it is suspected that this is due to the fluctuating drag which will be at twice the shedding frequency and hence will coincide with the natural frequency.

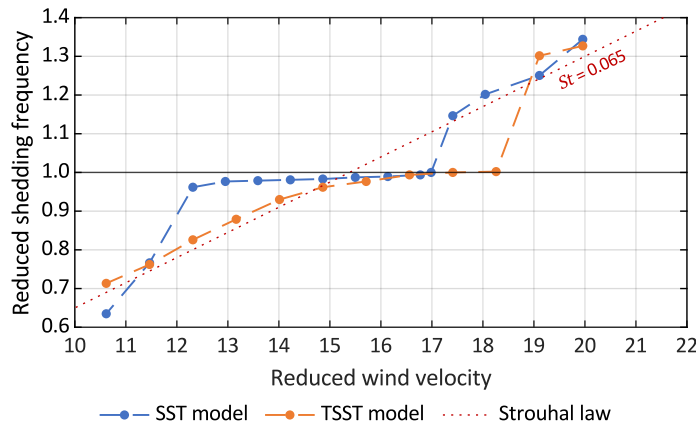


Figure 5.16. Reduced shedding frequency for SST (model DM01) and TSST (model DM02) turbulence models compared to the Strouhal law

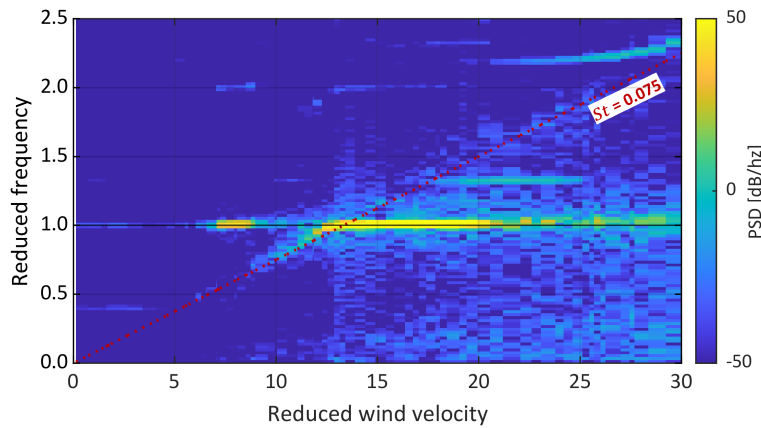


Figure 5.17. Spectrogram of the deck displacement obtained in the wind tunnel – generated from the data acquired from CSTB (reduced values) (Flamand & Knapp, 2018)

The difference in behaviour for the two flow regimes (locked-in and locked-out) is also illustrated in Figure 5.18, depicting the time record of the lift coefficient and the reduced deck displacement. Figure 5.19 shows the frequency domains for these two signals. Figure 5.18a shows a much cleaner response with two signals synchronized. This is confirmed in the frequency domain analysis shown in Figure 5.19a.

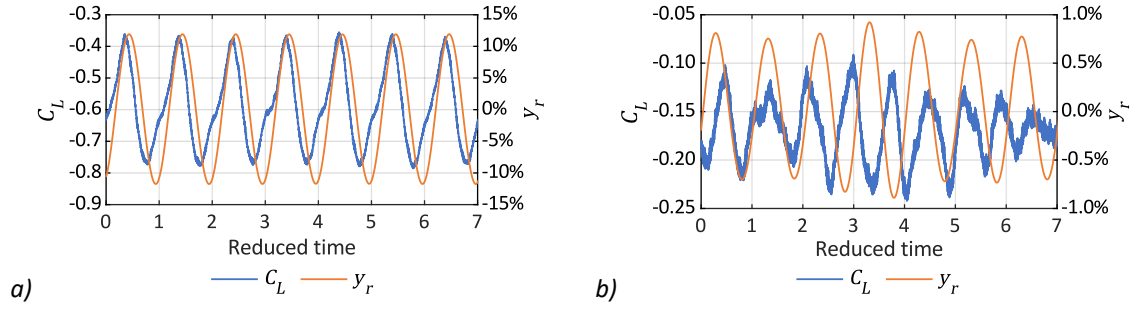


Figure 5.18. Time record of the lift coefficient and deck displacement (DM02) for U_r of a) 17 and b) 20

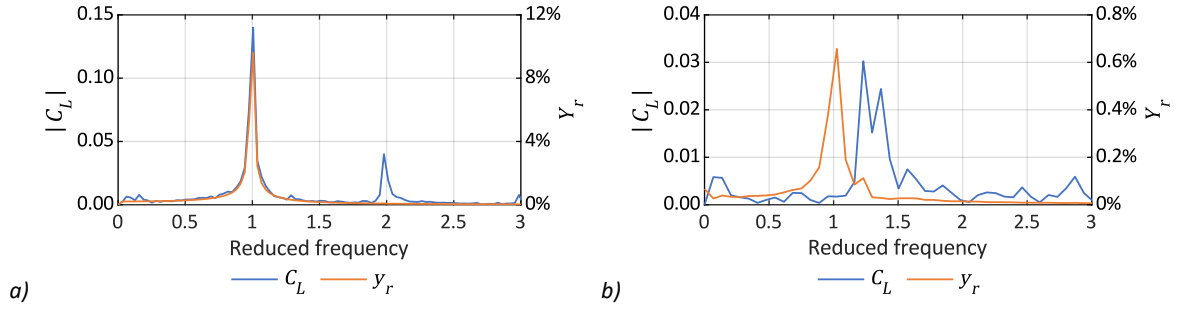


Figure 5.19. Frequency domain of the lift coefficient and deck displacement (DM02) for U_r of a) 17 and b) 20

Figure 5.20 and Figure 5.21 show the vorticity in the flow field at lock-in (reduced velocity of 17) and lock-out (reduced velocity of 20), respectively. In both cases vortex shedding in the wake is evident, but in the lock-in case, it shows spatial periodicity as would be expected. When the flow is in the locked-in regime (Figure 5.20), large alternating vortices form in the near deck wake, within a deck width (B). The vortex street, while very ordered, exhibits what looks like a 2P mode¹ on the lower side, but a 2S mode¹ on the upper vortices. In the case of locked-out flow (Figure 5.21), the vortices are formed much later downstream and the structures never form a clear vortex street as the two frequencies (vortex shedding and structural natural frequency) are distinct and incommensurate.

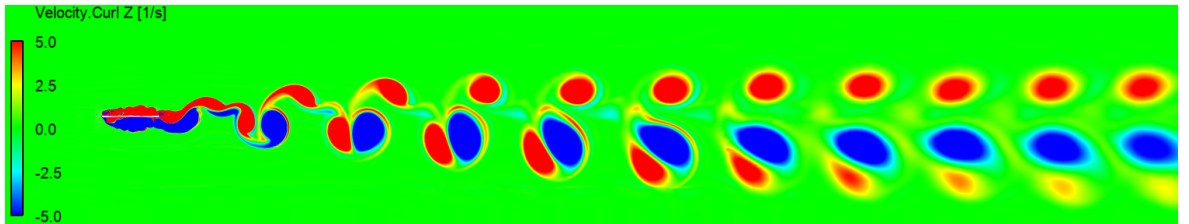


Figure 5.20. Vorticity magnitude in the lock-in region for $U_r = 17$

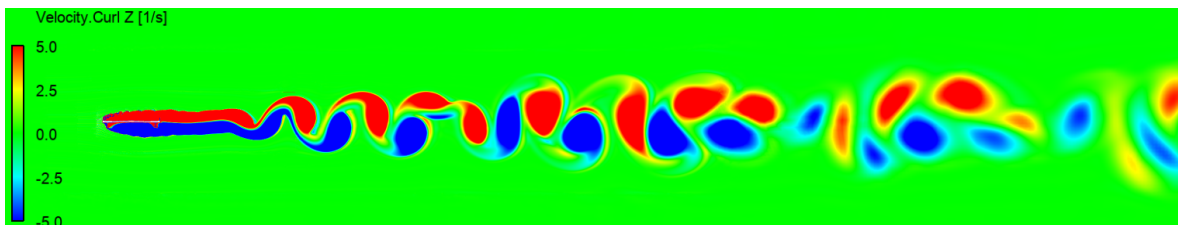


Figure 5.21. Vorticity magnitude at the lock-out for $U_r = 20$

¹ For more details on 2P mode (two pairs of vortices) and 2S mode (two single vortices) see for example Williamson & Govardhan, (2004)

5.3 Prescribed Motion Model (PM)

This section outlines the results obtained using the prescribed motion model coupled with the energy balance approach as described in Section 4.4. All variables are non-dimensionalised as defined in Chapter 4.

Figure 5.22 shows the aerodynamic energy transferred into the system with every oscillation cycle (ΔE_F), for a range of oscillation amplitudes and at the reduced velocity of 16.8. In the same figure, energy taken away by the damping (ΔE_S) is shown with the dashed line. For a given damping level, marked in red is the unstable region, i.e., the region in which limit cycle oscillations can occur. In green is the region for which the energy taken out from the system is greater than the aerodynamic energy brought into the system, and thus the LCO cannot occur. In the same figure, aerodynamic energy calculated from the FSI system (model DM02) for U_r equal to 16.8 is shown. In theory, this point should coincide with the point where *positive* and *negative* energy contributions are equal. The discrepancy can be attributed to the assumptions embedded in Equation 4-9 and the prescribed motion model being defined to capture only the first mode of oscillation.

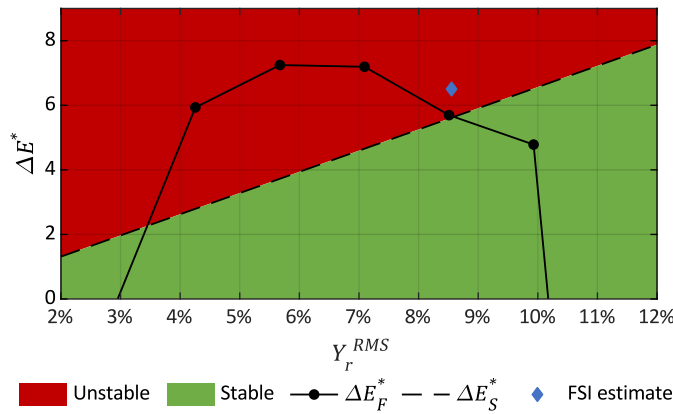


Figure 5.22. Fluid (ΔE_F) and damping (ΔE_S) energy increments per oscillation cycle at $U_r = 16.8$

A different way of presenting the same data is given in Figure 5.23. Here, the *positive* and *negative* energy contributions are subtracted defining the *stable* and *unstable* regions above and below the zero energy line. Again, the FSI estimate of the energy should lie on the zero axis however because of the limitations previously discussed this is not the case. An interesting observation can be made for $Y_r^{RMS} < 3.5\%$, the LCO does not exist for the analysed flow velocity and the given damping level. So, to enter the LCO, the system needs to have an initial displacement, large enough so that the system enters the *unstable* region. With this, the aerodynamic energy per cycle is greater than the dissipated energy, and the oscillation amplitude increases (direction marked with yellow arrows) until the equilibrium between aerodynamic and dissipated energy has been reached. This happens at approximately $Y_r^{RMS} = 8.5\%$, which is the displacement amplitude estimated by the FSI model. This behaviour is further examined in Chapter 6. When the net energy is below zero, the oscillation amplitude decreases either towards zero (direction marked with purple arrow) or towards an equilibrium point (direction marked with blue arrow).

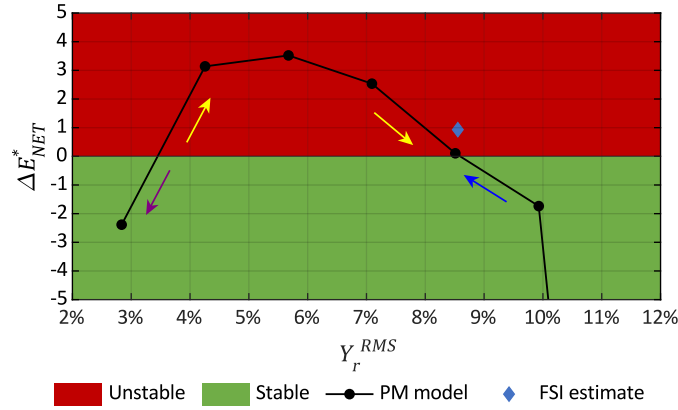


Figure 5.23. System net energy (ΔE_{NET}^*) per oscillation cycle for $U_r = 16.8$; arrows mark the tendency of the system amplitude change depending on the net energy

Here it is shown that for a given flow velocity and the damping level, it is possible to predict, relatively accurately, the maximum oscillation amplitude by using the prescribed motion model (when compared to the FSI model). The main advantage of this approach over the FSI model is that for each data point, the simulations can be run independently and at the same time. This shortens the time needed to generate an estimate for the maximum amplitude. However, when it is unknown at what velocity the maximum amplitude occurs, one is required to run the simulations over a range of velocities and amplitudes generating a 3D representation of the aerodynamic energy that is transferred into the system.

Figure 5.24 shows the data simulated for a range of U_r and Y_r . The aerodynamic energy brought into the system every oscillation cycle is shown in green, while the negative energy, taken out from the system during each oscillation cycle by the damping, is shown in red for a damping ratio of 0.45%. Where these two surfaces intersect, the system energy summed over one oscillation cycle equals zero and the LCO occurs. The same data, shown from the top view, is given in Figure 5.24b. The region where the aerodynamic work is larger than the negative work done by the damper is marked in green. The opposite is true for the region marked in red. Additionally, the results from the FSI model (model DM02) are plotted in black. It is seen that the intersection between the red and green regions, where the *positive* and *negative* energy fluxes are equal and the LCO occurs, matches closely the FSI model predictions of the response amplitude. This approach also predicts the oscillation amplitudes which exist at the LCO for each flow velocity, and they are in good agreement with the FSI model.

The main advantage of the energy approach is that the effect of different structural damping can be easily and rapidly evaluated. For example, in Figure 5.24a, the blue surface represents the energy dissipated with a damping ratio (ζ) of 0.55%. The projection onto the velocity-response amplitude plane can be seen in Figure 5.24c which shows a reduced peak response amplitude and a slightly reduced lock-in range as would be expected for increased damping.

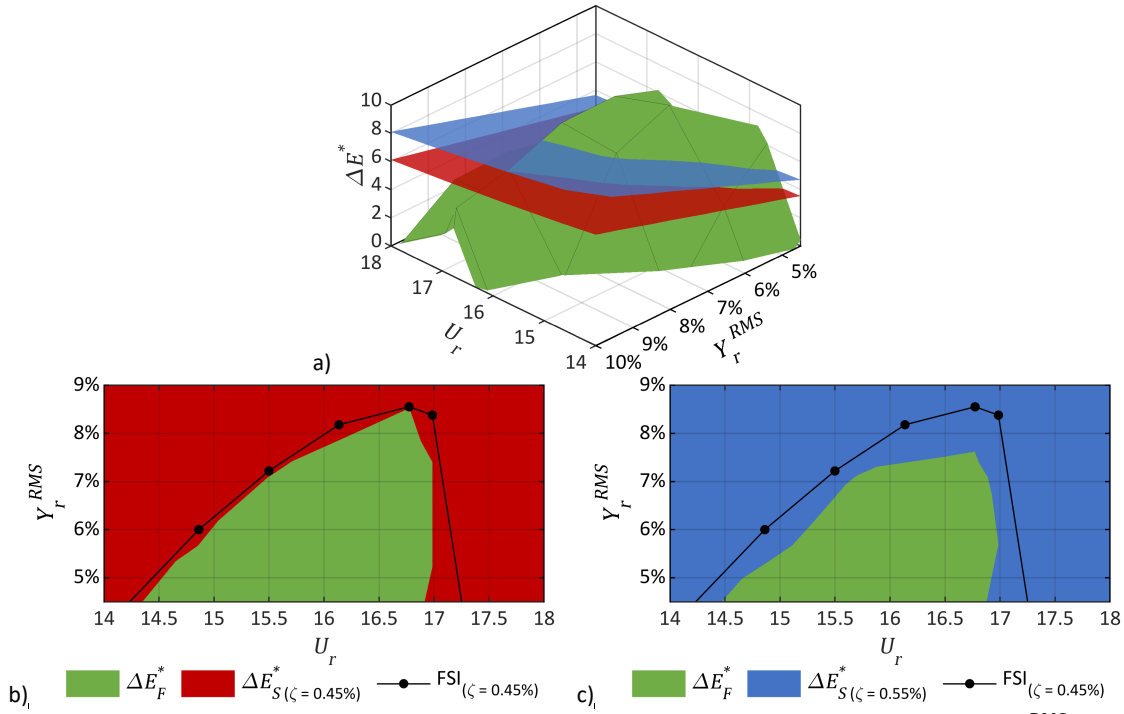


Figure 5.24. 3D plot of energy brought in and energy lost per oscillation cycle for a range of Y_r^{RMS} and U_r compared to the “displacement vs flow velocity” plot obtained from the FSI model (model DM02); a) Isometric view, b) top view $\zeta = 0.45\%$ c) top view $\zeta = 0.55\%$

Figure 5.25 shows the aerodynamic energy brought into the system with every oscillation cycle for Y_r^{RMS} equal to 7% and 11% at $U_r = 16.8$. For the first case, the energy value stabilises after approximately 25 cycles and remains positive, meaning the fluid force is exerting work on the deck. The opposite holds for the second case, with $Y_r^{RMS} = 11\%$, the energy brought into the system is negative, meaning that the deck is exerting work on the surrounding fluid. Having negative aerodynamic work excludes this (U_r, Y_r^{RMS}) condition as one under which LCO can occur, regardless of the structural damping. This is confirmed in Figure 5.22. To have the curve showing the work done by the system damper intersect the curve describing the aerodynamic work, the system damping would have to be negative. For a given flow velocity, it is possible to estimate the maximum LCO amplitude that theoretically exists when the system damping is zero.

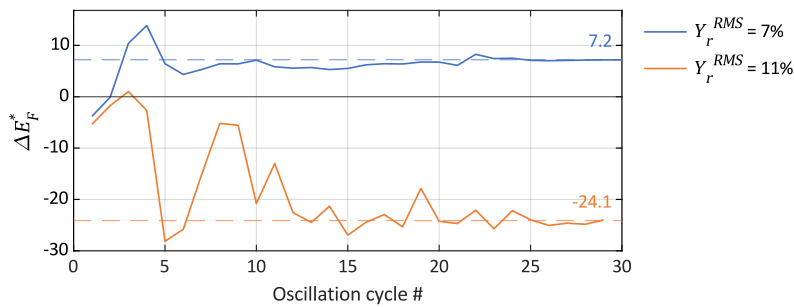


Figure 5.25. Time history of the work done by the fluid force on the oscillating body per oscillation cycle for two amplitudes at $U_r = 16.8$

The results from the prescribed motion analysis, for a given flow velocity, can be used to estimate what is the necessary system damping at which the displacement amplitude is acceptable. The flow velocity at which the maximum displacement occurs can be estimated based on the Strouhal number acquired from the static transient analysis. The transient analysis (model TM01) gives an estimate of $St = 0.065$, which

translates to $U_r = 15.4$ at the coincidence. However, the largest displacement occurred at $U_r = 16.8$. Looking at the results from the FSI model (Figure 5.15), it can be noticed that after the coincidence, the shedding frequency remains locked to the natural frequency of the bridge, until it locks out at approximately $U_r = 17$. This means that, after coincidence, the increase in displacement comes from the increase in aerodynamic energy which is increased due to a rise in the flow velocity. This behaviour continues until the shedding frequency locks-out, which disrupts the inflow of the aerodynamic energy and the oscillations due to vortex shedding disappear. This sudden lock-out is noticeable in the FSI simulations and the prescribed motion models.

The overall goal of these simulations is to show different ways in which CFD can be used as a preliminary design tool to provide early aerodynamic results for bluff bridge decks. At the preliminary stage, the design is not concerned with the maximum displacement amplitude caused by the VIV, but with the overall aerodynamic stability of the bridge deck. If the value of the maximum acceptable displacement amplitude is set, the prescribed motion model can be used to evaluate if and for what wind velocity range that value will be exceeded. The maximum wind velocity up to which that range spans is set as the design wind velocity for the VIV defined by the design code. Figure 5.26 shows the aerodynamic energy per oscillation cycle for a set displacement amplitude over a range of flow velocities. The aerodynamic energy (ΔE_F) for three reduced displacement amplitudes is shown with solid lines. They are indexed 1, 2, and 3, for Y_r^{RMS} values of 6%, 7% and 10%, respectively. The negative work done by the damping (ΔE_S) is shown with dashed lines for each of the three displacement amplitudes. The range of flow velocities for which the LCO can occur decreases with the increasing displacement amplitude. For the highest displacement amplitude shown ($Y_{r,3}^{RMS} = 10\%$), the LCO cannot occur for the analysed range of reduced velocities, because the energy taken out of the system by the damper is always higher than the aerodynamic energy brought into the system by the fluid forces. This gives an indication of how the analysed deck will perform in the wind tunnel tests. As previously mentioned, the prescribed motion models can be run simultaneously for a number of data points so the graph shown in Figure 5.26 can be produced relatively quickly. However, it should be noted that in cases when the damping is rigidly set, and the design needs the maximum displacement amplitude due to VIV at an early stage, the FSI model will provide results faster.

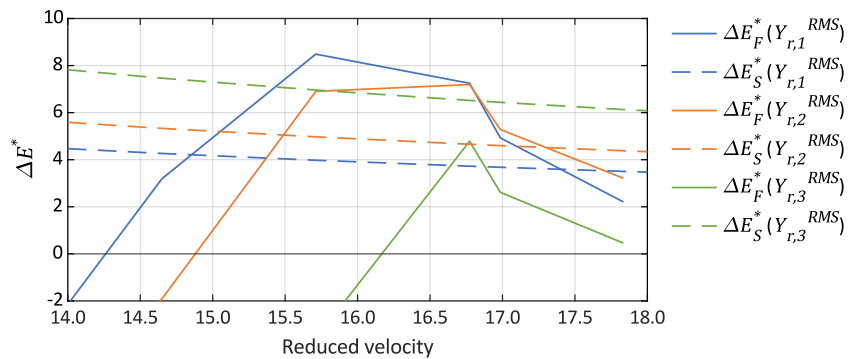


Figure 5.26. Aerodynamic (solid line) and damping (dashed line) energy increments per oscillation cycle for a range of flow velocities for $Y_{r,1}^{RMS} = 6\%$, $Y_{r,2}^{RMS} = 7\%$ and $Y_{r,3}^{RMS} = 10\%$

5.4 Application to Other Case Studies

The principles described in the previous sections were applied to the other two case studies described in Chapter 3 and the results are presented here. The purpose of this exercise was to estimate the aerodynamic behaviour of the decks prior to the commencement of the wind tunnel tests.

The static force coefficients were estimated for the Clyde Bridge deck fitted with the fairing plates. The initial shape of the wind fairings was developed based on the CFD results. The CFD values compared to the wind tunnel results are shown in Figure 5.27 and a good agreement is observed for all three coefficients.

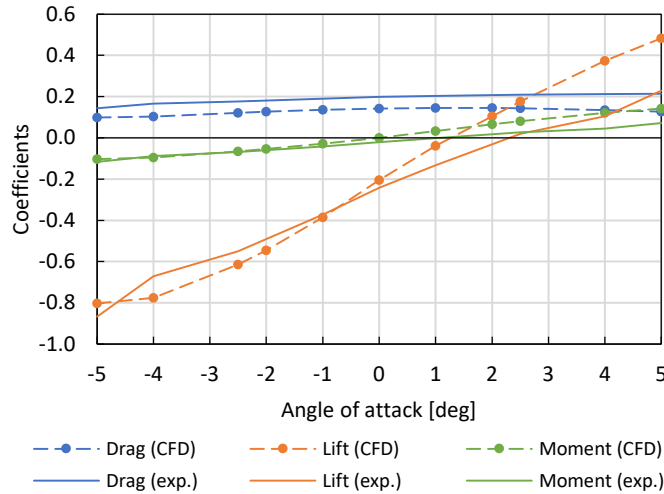


Figure 5.27. Static coefficients for the Clyde Bridge deck with the fairing plates as shown in Figure 3.8 compared to the wind tunnel test results

In the case of the Narrow Water Bridge, a number of points on the free vibration response curve were obtained with relatively good accuracy when compared to the wind tunnel tests (see Figure 5.28). However, the initial simulations experienced issues in yielding meaningful results. The VIV response of the deck was observed only after an initial displacement kick was applied to the deck motion. It was suspected that the large, cantilevered ends of the deck (i.e., overhangs), act as a splitter plate placed in the wake of a bluff body attenuating the VIV response (see Section 2.2.7). This instigated the study on the overhang length effects on the bridge deck VIV response of which the results are presented in Chapter 6.

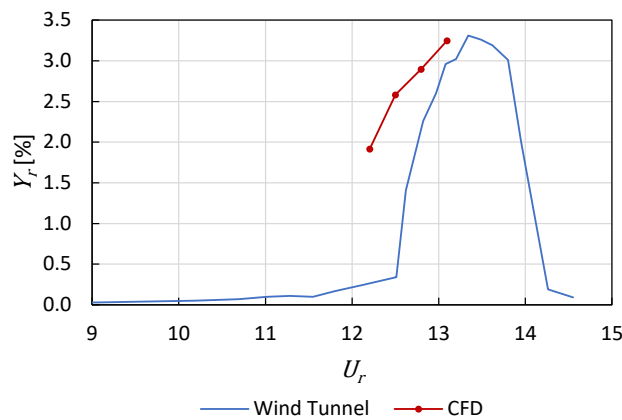


Figure 5.28. VIV response curve for the Narrow Water Bridge deck without the wind shielding ($Sc = 47$) as shown in Figure 3.15 compared to the wind tunnel test results

5.5 Synthesis

In Section 2.5.3, it was shown that most of the CFD analyses done to date were undertaken on streamlined or semi-streamlined bridge decks. It is also shown that only one provided input for the design process, while the others were part of wider research studies. The goal of this research is to explore the applicability of CFD as a tool that could be employed using modest computational power (i.e., a desktop workstation) in everyday bridge design to provide preliminary aerodynamic results for bridge decks whose shape is not aerodynamically optimised (i.e., bluff bridge decks). This will allow for more informed, more efficient, and faster wind tunnel campaigns for these types of decks. A proposed methodology for using CFD in bridge design (at the preliminary stage) is outlined in Table 5.1.

Table 5.1. Proposed modelling approach for the preliminary design stage in bridge engineering

No.	Step	Description
1.	Defining the geometry	The geometry is simplified by removing small-scale features such as curbs and steps. Barriers should be included with the actual porosity level however, the number of holes in the barrier may be varied with little effect on the results.
2.	Building the computational grid and model	The computational domain is divided in such a way that cell size can be conveniently controlled in the areas important for the aerodynamic profile of the deck. If the end goal is producing the FSI results, care should be given to the regions that will undergo re-meshing. Re-meshing should be done away from the regions with small cell sizes. RANS based simulation should be employed with the SST turbulence model. Mesh dependency study and the verification of the y^+ value should be done. It is noted that these should be undertaken for the highest Reynolds number that will be simulated
3.	Performing simulations on the static model	Static force coefficients The results may be compared to those estimated by the design codes to verify for any inconsistencies. Possible issues with the divergent response of the cross-section should be investigated by applying the Den Hartog criterion (Equation 2-11). Should these be identified, the deck shape should be modified, for example by incorporating different leading edge geometry. The process should be repeated until no issues with the divergent response are identified. Strouhal number estimate Transient analysis can provide estimates of the Strouhal number which in turn provides the critical wind velocity for vortex shedding. It should be noted that this value may be different to the one obtained for the vibrating system.
4.	Performing simulations on the dynamic model	Prescribed motion model This analysis will yield useful results should a damping sensitivity analysis be of interest. A range of flow velocities and displacement amplitudes should be defined over which the analysis is to be run. Care should be given to the time needed for the generation of a single data point and an estimate of the total wall clock time needed should be made. Free-to-oscillate model If the damping value is set with little possibility for variation, a free-to-oscillate model should be developed and simulations on it performed for a range of flow velocities. The starting wind velocity should be the critical wind velocity identified from the static model. Should the deck experience low VIV, a model with an initial displacement kick should be tested to verify the possible manifestation of the motion-induced vortices.

The key findings of this study are as follows:-

- Simplified deck geometry can produce useful initial results on the lift coefficient. However, the barriers and girders must be included in the analysis because the flow around the deck is heavily influenced by these elements.
- If the overall porosity of the modelled barriers follows that of the bridge design, the number of openings in the barrier can be varied with little effect on the static force coefficients.

- c) Expanding the 2D model to the third dimension only slightly changes the force coefficients while dramatically increasing the computational power needed. It was shown that two shear layers constrain the flow features originating from small 3D features in the downwind barrier, limiting the effect on the far wake. Thus, when turbulence is modelled and not resolved, 2D analysis can provide sufficiently accurate results. It is noted that should a 3D model be of larger spanwise length than the ones covered in this study, the findings could be different but LES or an anisotropic turbulence model may be necessary.
- d) For the simulations done at the Reynolds number of a similar order of magnitude to the one present in the wind tunnel test ($\sim 10^4$), the Transition SST turbulence model provides the results closest to the wind tunnel tests. With increasing Re number ($>10^5$), the TSST model converges towards the SST model. It should be noted that the TSST model requires increased computational effort due to two additional transport equations when compared to the SST. It was observed that, on average, the TSST models took approximately 15% longer to converge.
- e) Free-to-oscillate Model (FSI) can provide results comparable to the ones obtained from the wind tunnel tests, successfully capturing the lock-in phenomenon and the range of velocities under which VIV occurs. The amplitude of the displacements is larger than what is seen in the wind tunnel tests. It is suspected that the overestimation of the excitation is due to the 2-dimensionality of the simulations which imposes a full spanwise correlation. Better results could be expected from a 3D analysis coupled with LES or an anisotropic turbulence model.
- f) The Prescribed Motion model (PM), coupled with the energy balance approach, can provide results comparable to those obtained with the FSI model. The PM model produces results, per data point, faster than the FSI model. From the simulations done in this study, it was seen that, on average, the time needed to reach a steady solution for a single data point is halved. However, when the wind velocity at which the maximum displacement amplitude occurs is unknown, more data points are needed making the FSI approach faster in yielding results. Nonetheless, the PM model is superior to the FSI model when the sensitivity to structural damping is to be investigated. It was also concluded that for a set displacement amplitude and damping level, the PM model can yield a range of flow velocities for which the VIV can occur.

The results in this chapter demonstrate that CFD can be used for acquiring preliminary aerodynamic results in bridge engineering when used on bluff bridge decks and of comparable shape to the one studied here. Further work should be undertaken to develop and investigate similar approaches that can be applied to a broader range of deck geometries and ultimately use CFD results to avoid wind tunnel testing for cases that are only marginally susceptible to aerodynamic effects.

Chapter 6

Overhang Length and Bridge Deck VIV

This chapter details the results obtained from the study of the overhang length effects on the VIV of the bridge decks. The analysis comprises five geometries without the barrier (Section 6.2) and five geometries with the barriers (Section 6.3) while a synthesis of the results is given in Section 6.4. The description of the models and the methodology employed in this study is given in Section 4.2 and Section 4.4. The results shown in this chapter are partially presented in Duranovic & Meskell, (2024).

6.1 Introduction

A 3D representation of typical results is shown in Figure 6.1. Green and red circular markers represent the positive and negative fluid energy increments obtained from the prescribed motion models for a range of oscillation amplitudes and flow velocities. They define the fluid energy surface (ΔE_F) of the studied deck geometry. A cubic spline function was used for the interpolation of values between the simulated data points and to better capture the intersection between the fluid and damping energy. The function was used only as an interpolation tool, while extrapolation was avoided. For regions where sudden changes in fluid energy values were observed, additional data points were generated. Data points marked as red crosses represent the locked-out cases. The difference between the red (negative) data points and the locked-out points should be emphasised. For the former ones, (U_r, Y_r) conditions are such that if the amplitude (Y_r) was larger, the vortex induced vibrations would occur, i.e., they define the limit of susceptibility to VIV for a given cross-section and Scruton number. For the locked-out cases, the increased amplitude does not guarantee the occurrence of the VIV. The shedding and natural frequencies are out-of-sync which, on average, always yields zero fluid energy. Damping energy surfaces (ΔE_S) for three Scruton numbers are shown in blue, while the $\Delta E_F = 0$ surface is shown in red. As noted in Section 4.4, the data points found below $\Delta E_F = 0$ represent the conditions at which the deck is doing work on the surrounding fluid.

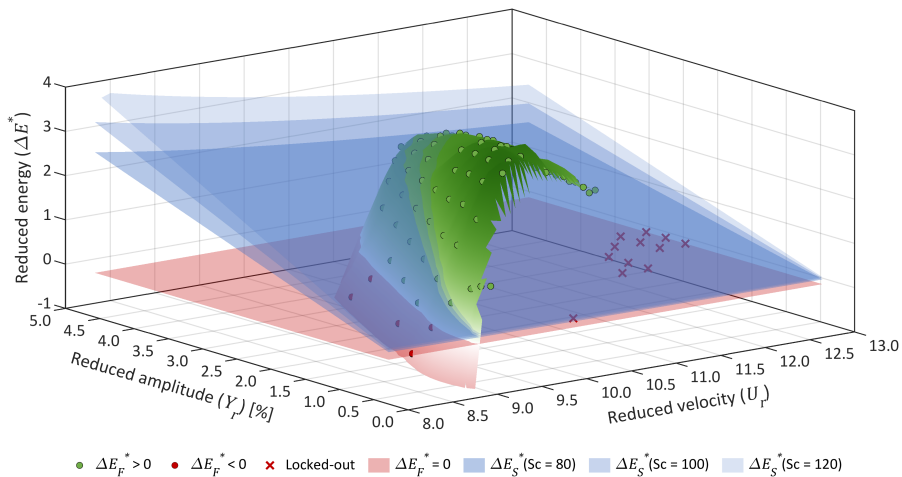


Figure 6.1. Prescribed motion model results (geometry S0) in 3D view; dot and cross markers – data points obtained from the CFD simulations; blue surfaces – damping energy calculated using Eq. 4-9 and Eq. 4-12

For easier discussion, the results in 3D representation are shown in the plan (see e.g., Figure 6.2), where contours represented with dashed, solid, and dot-dashed lines describe the intersection between the fluid energy surface and the damping energy surfaces for each Sc number. The zero fluid energy contour is shown with the red solid line. A typical figure also shows the velocity at coincidence (U_{vs}) as estimated by the fixed model. Furthermore, the reduced amplitude of 1.0% and the reduced velocity of 8.5 are marked in bold for easier referencing across all figures.

6.2 Deck Without the Barrier

The results for deck sections without the barriers, i.e., geometries S0 through S50, are given and discussed in this section. As it will be seen, the results can be divided into two groups based on the shape of the LCO regions characteristic for each, and as such are presented separately. The first group features geometries S0, S10 and S25, while the second one includes the remaining two (S40 and S50).

6.2.1 Geometry S0, S10 and S25

Prescribed motion model results for geometry S0 are shown in Figure 6.2 and they estimate the maximum reduced amplitude at approximately 4.5%, 4.0%, and 3.7% for three Scruton numbers, respectively. The velocity at coincidence estimated from the fixed model is in good agreement with what is obtained from the PM model. The lock-in region spans between reduced velocities of approximately 9 and 12 with higher and lower Scruton numbers shifting the upper boundary by -3% and +10%, respectively. It is worth noting that the lower boundary of the lock-in range remained substantially unaffected by the change in Sc number (the same was observed for the Narrow Water Bridge in experiments, see Figure 3.18). The maximum oscillation amplitude occurred at more or less the same reduced velocity ($U_r \approx 11.5$) for all Sc numbers.

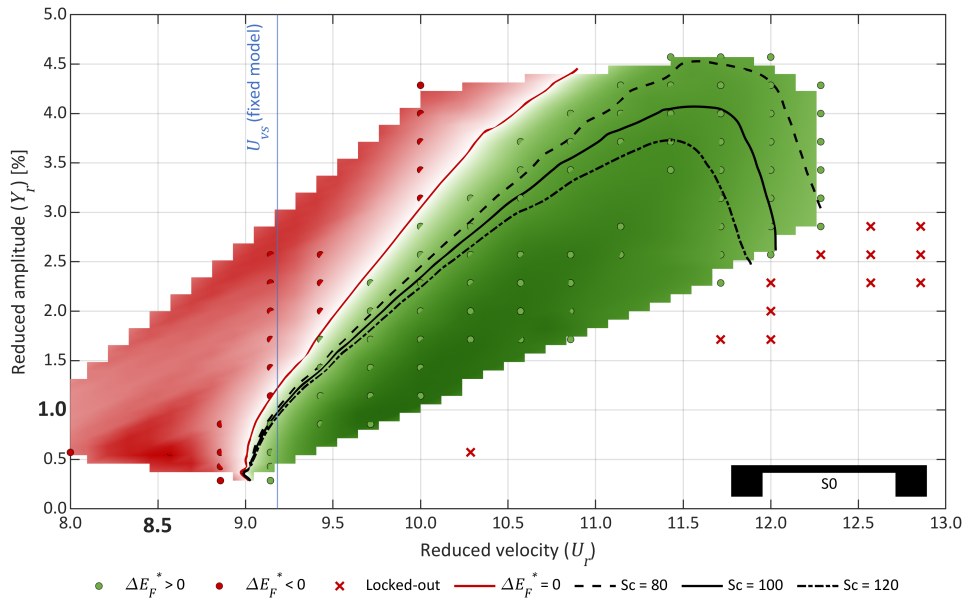


Figure 6.2. Prescribed motion model results for geometry S0

The introduction of a small overhang is assessed with geometry S10 and the results are shown in Figure 6.3. A reduction in maximum amplitude of approximately 18% is noticeable for all three Sc numbers. The width of the lock-in range was reduced by approximately 30% with the onset at the reduced velocity of approximately 8.5.

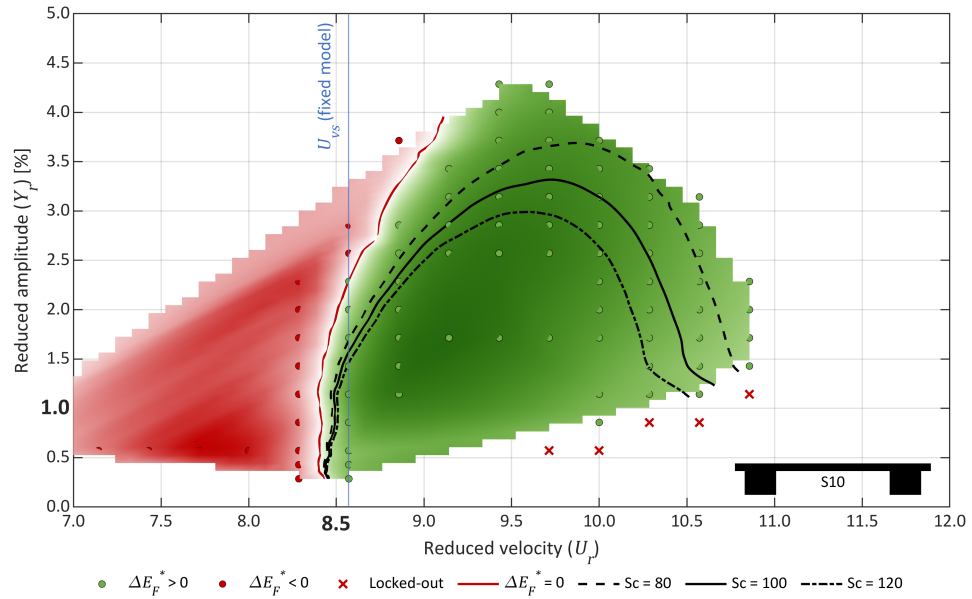


Figure 6.3. Prescribed motion model results for geometry S10

The maximum amplitude is further decreased for geometry S25 with the results shown in Figure 6.4. When compared to those of geometry S0, a reduction in maximum amplitude (approximately 69%, 73% and 74% for three Sc numbers) and the width of the lock-in region (approximately 72%) is observed. The change in Sc number has almost twice as large an effect on the maximum amplitude than what it had for geometries S0 and S10.

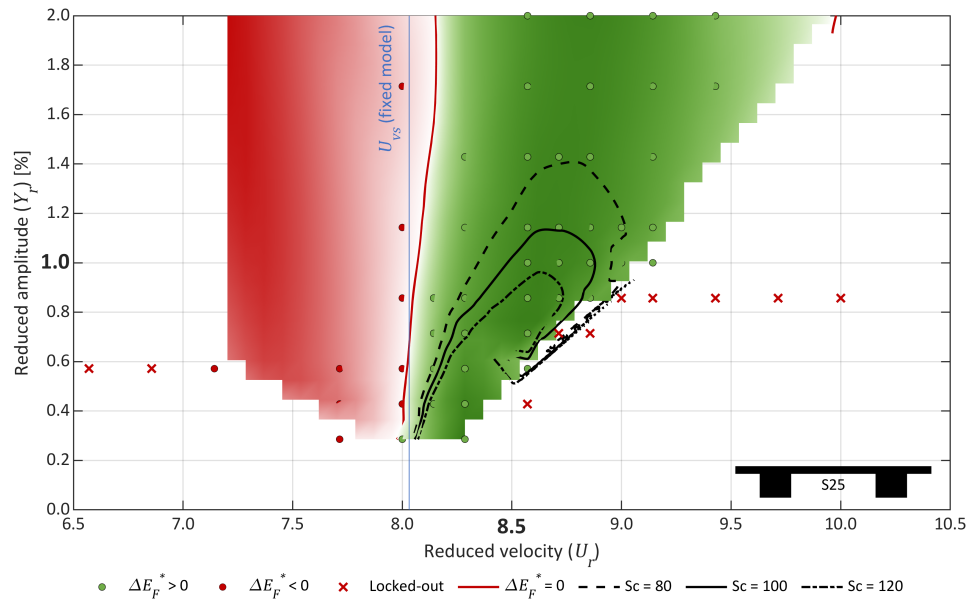


Figure 6.4. Prescribed motion model results for geometry S25

6.2.2 Geometry S40 and S50

Further increase in the overhang length brings a couple of new features to the deck section behaviour. The results for geometries S40 and S50 are presented in Figure 6.5 and Figure 6.6, respectively and are discussed in parallel. Looking at the behaviour exhibited by geometry S50, two separate LCO regions are noticeable. The first one begins at the reduced velocity of approximately 8.0 (a value similar to the one estimated by the fixed model). Similar behaviour is found for the lowest Sc number for S40. This LCO region

is characterised by a rising branch only, which is interrupted by the lock-out of the vortex shedding and oscillation frequencies. For S40, the maximum amplitude characteristic for this region is approximately 75% lower than that of S0 and approximately 76% lower when the overhang length is increased to S50. The same comparison, but for the lock-in region width yields a reduction of approximately 80% and 84%, respectively.

The second LCO region, characterised by an island-like feature, occurs at the reduced velocity which is approximately 1.06 times higher than the coincidence velocity estimated by the fixed model. Furthermore, for each flow velocity within this region, there exist two oscillation amplitudes (upper and lower) that satisfy the LCO stability conditions. The system can reach the upper one as the flow velocity is increased, assuming the system is already in that LCO region. However, reaching the lower stability point may happen only if the system is excited with an amplitude equal to the amplitude for that stability point. Furthermore, even if the system *manages to get* to a point on the lower branch, any perturbation in the flow will cause the system to immediately *slip away* from it. On the energy surface, the perturbation will push the system either *upwards*, increasing the oscillation amplitude until it reaches the upper LCO branch, or *downwards* which will decrease the oscillation amplitude and make the system lose the energy until it reaches the locked-out region. Staubli, (1983) too reported the existence of two LCO regions from the PM model results obtained experimentally for a circular cross-section (see Figure 2.14 in Section 2.2.5), although the author questioned whether the *island* is real or not.

A comparison between geometry S40 and the Narrow Water Bridge deck section without the wind shielding¹ can be made. From Figure 3.18 in Section 3.3, it can be seen that the increased Sc number has a strong effect on the VIV response (it fully mitigated the VIV). In Figure 6.5, an equally significant drop in the maximum displacement amplitude is observed between $Sc = 80$ and $Sc = 120$ for geometry S40.

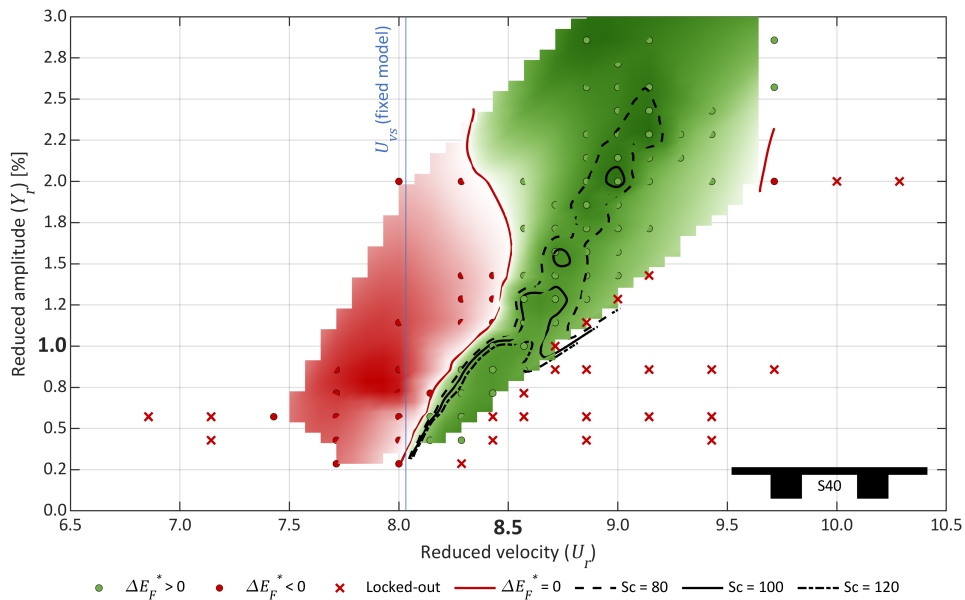


Figure 6.5. Prescribed motion model results for geometry S40

¹ See Figure 3-15. The relative length of the overhang on the NWB deck section is between that of S40 and S50 (although closer to S40).

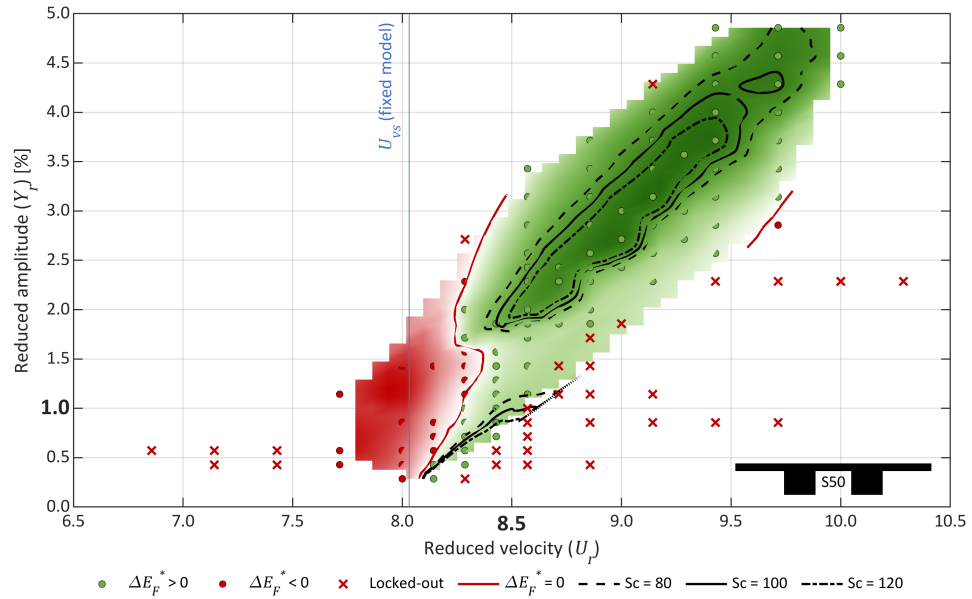


Figure 6.6. Prescribed motion model results for geometry S50

The behaviour within two LCO regions is further examined by visualising the flow around the deck. The vorticity fields for two simulation points picked from the energy surface for geometry S50 (Figure 6.6) are visualised in Figure 6.7 at the phase angle $\frac{\pi}{2}$, Figure 6.8 at the phase angle π , and Figure 6.9 at the phase angle $\frac{3\pi}{2}$. The first condition is located in the bottom LCO region and is for $(U_r, Y_r) = (8.1, 0.4\%)$, while the second one is within the upper LCO region (i.e., the *island*) and is for $(U_r, Y_r) = (9.1, 3.4\%)$. It can be observed that for the condition in the upper LCO the bottom vortices originate from the leading edge of the upstream box girder, and the vortices on the top side start forming from the deck leading edge. Unlike for the lower LCO region, here the vortices seem to go around the overhang thus avoiding being affected by it. The vortices for the lower LCO region form in the wake of the deck. Another observation can be made regarding the vortex street. For the upper LCO region, it is at an angle, suggesting a stronger downwash (which is related to a stronger lift force).

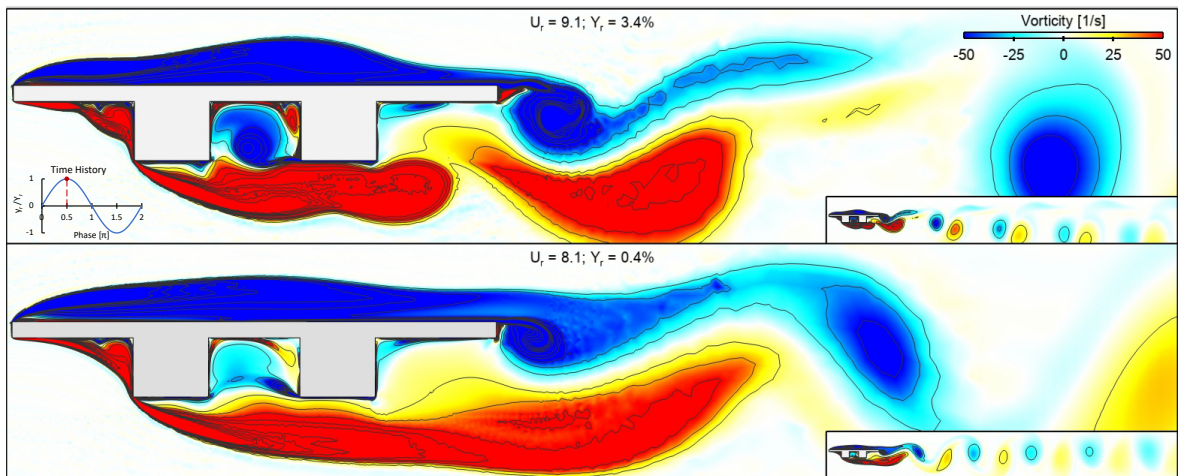


Figure 6.7. Vorticity field around geometry S50 at the phase angle $\frac{\pi}{2}$ for (U_r, Y_r) equal to $(8.1, 0.4\%)$ in bottom and $(9.1, 3.4\%)$ in top figure

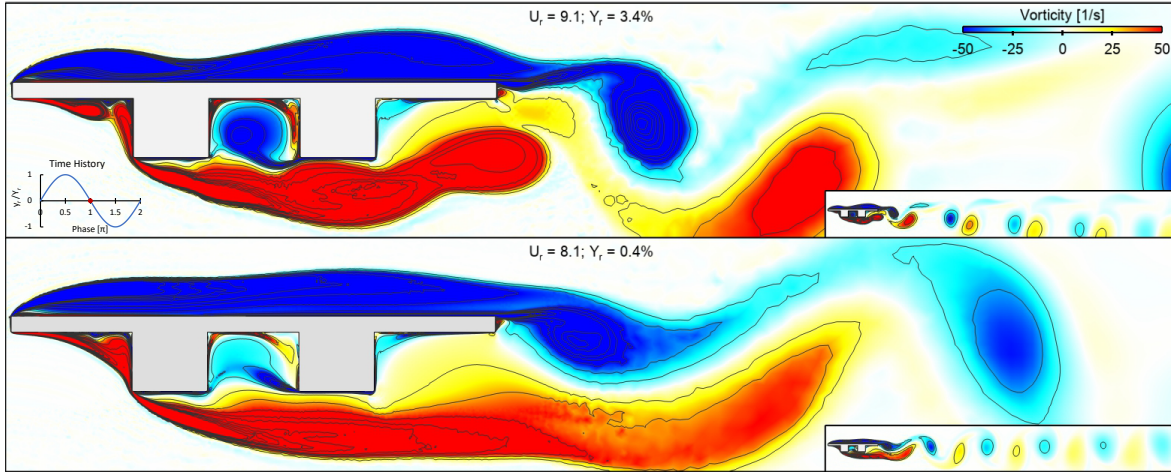


Figure 6.8. Vorticity field around geometry S50 at the phase angle π for (U_r, Y_r) equal to (8.1, 0.4%) in bottom and (9.1, 3.4%) in top figure

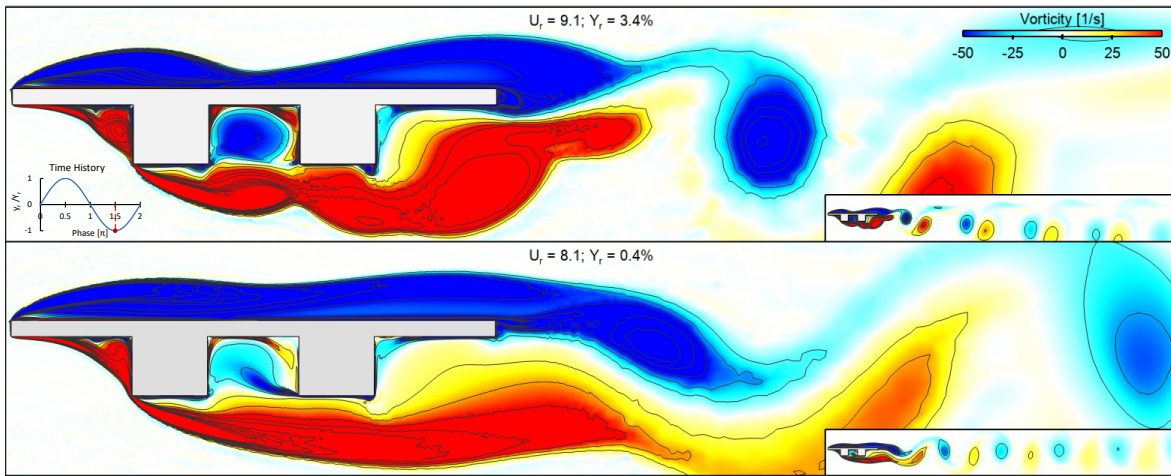


Figure 6.9. Vorticity field around geometry S50 at the phase angle $\frac{3\pi}{2}$ for (U_r, Y_r) equal to (8.1, 0.4%) in bottom and (9.1, 3.4%) in top figure

6.2.3 Discussion

This section includes further discussion on two LCO regions noticeable for geometries S40 and S50. Since the lower LCO region begins at U_{vs} and the vortices are formed in the wake of the flow (suggesting wake instability), it can be assumed that the lower LCO region corresponds to the Karman vortex (KV) shedding phenomenon. This assumption is further strengthened by the diminishing effects that the overhang has on the intensity of the shed vortices, which are similar to the effects produced by a splitter plate placed in the wake of a bluff body (see Section 2.2.7).

When it comes to the upper LCO region, as already noted, the flow velocity at which the region becomes noticeable is different than U_{vs} . It occurs for oscillation amplitudes larger than the ones occurring due to the KV shedding. Also, this type of shedding mechanism is only observed for geometries that exhibit low KV shedding (S40 and S50), implying that the KV shedding might have mitigating effects on it. These observations suggest that the vortices formed within the upper LCO region could be related to motion-induced vortices (see Section 2.2.3 and Section 2.2.7). Matsumoto et al., (1993) state that, for a rectangular section, MIV occurs at the reduced velocity $U_{r,MIV} = \frac{B}{0.6D}$ where B and D are the section width

and depth, respectively. Furthermore, B_{eff} was defined for non-rectangular sections as $B_{eff} = U_{r,MIV} 0.6D$ when $U_{r,MIV}$ is obtained from the experimental tests. From Figure 6.5 and Figure 6.6, $U_{r,MIV}$ can be estimated at approximately 8.50 for S40 and between 8.37 and 8.50 (depending on the Sc number) for S50, which yields B_{eff} equal to $5.1D$ and $5.0D - 5.1D$ for these two geometries, respectively. From Table 4.4 it is seen that for both geometries $B = 6.4D$ while $B^* = 3.9D$ for S40 and $B^* = 3.2D$ for S50. Matsumoto et al., (1993) justified $B \neq B_{eff}$ (for the HB section – see Figure 2.23) by the fact that the separation points are not at the deck leading edge but further downstream. Similar justification can be given for geometries S40 and S50 with the top and bottom separation points being displaced by the overhang length. It can be noted that, although the overhang length for S50 is larger by 25%, B_{eff} remained more or less constant.

Whether the oscillating deck *manages* to reach the upper LCO region or not, depends on the motion amplitude corresponding to the lowest flow velocity in that LCO region. As seen in Figure 6.6, for the geometry S50 deck to oscillate in the upper LCO region the reduced motion amplitude must be at least 1.8%. To investigate what are the conditions needed for the deck to lock-on to the top branch of the upper LCO region, three PM-FSI simulations on geometry S50 were performed. A PM-FSI simulation consists of the initial stage where the deck motion is controlled by the PM model, and the main stage where the deck motion is controlled by the FSI model. Essentially, the PM model is used to set the initial conditions for the FSI model simulation. For all three simulations: the flow velocity was set at $U_r = 9.14$ and the Scruton number (once in the FSI regime) was equal to 100. Figure 6.10 shows the time history of three PM-FSI simulations:-

- i) the PM model was executed with $Y_r = 3.4\%$ and was interrupted at $0.25T$ with T being one oscillation period (shown in blue),
- ii) the PM model was executed with $Y_r = 3.4\%$ and was interrupted at $10T$ (shown in orange),
- iii) finally, the PM model was executed with a larger displacement amplitude, $Y_r = 4.6\%$ and was interrupted at $0.25T$ (shown in green).

Furthermore, for $U_r = 9.14$ and $Sc = 100$ the PM model estimates the LCO to occur at approximately $Y_r = 3.18\%$ (see Figure 6.6 – the upper branch). This LCO amplitude is marked with the black dashed line in Figure 6.10.

By comparing the first two simulations, it can be noticed that for the one with the longer initialisation period ($10T$) the deck motion eventually converged to the estimated LCO amplitude. While in the first simulation, the deck motion declined (due to the structural damping) and eventually completely faded out. This is because the wake did not have time to become *fully populated* with the shed vorticity associated with the large deck motion, which would then cause MIV to occur. The third simulation shows that if the kick is large enough, even with a short initialisation time ($0.25T$), the deck motion reaches the LCO. This is because, by the time the deck motion amplitude would have declined below a *critical* value, the motion-induced vortices were already formed and were able to excite the deck. Here, the minimum oscillation amplitude needed for the MIV to occur is referred to as the *critical amplitude*.

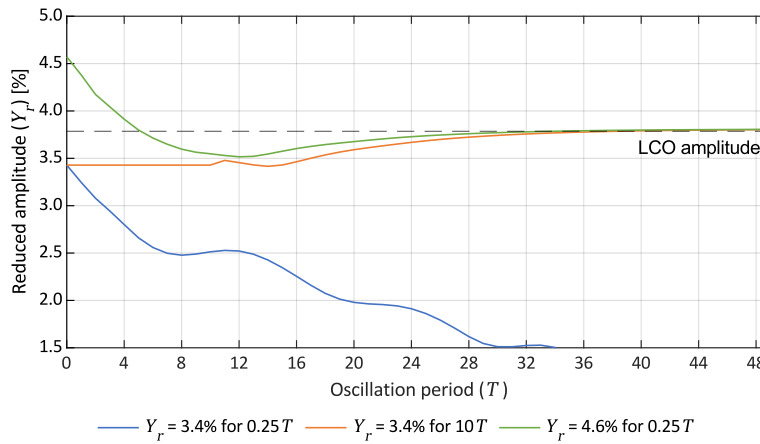


Figure 6.10. Time history for the three PM-FSI simulations for $U_r = 9.14$ and $Sc = 100$; in blue – $Y_r = 3.4\%$ sustained for $0.25T$; in orange – $Y_r = 3.4\%$ sustained for $10T$; in green – $Y_r = 3.4\%$ sustained for $0.25T$;

The aim of this exercise was not to quantify this *critical amplitude* but rather to understand whether a kick can instigate the motion-induced vibrations. The results suggest that unless the kick amplitude is sustained for a number of oscillation periods, the kick amplitude needs to be larger than the estimated LCO amplitude for the deck motion to reach it. This is to allow enough time for the wake to become populated by the flow that is disturbed by the deck motion. A separate study should be done to investigate the magnitude of the kick needed for the deck to reach an LCO amplitude. Other than the kick magnitude, this study should also include the level of the structural damping. This is because the damping controls the number of oscillations needed for the deck amplitude to decline from the value of the initial kick to below the critical one (not large enough for the MIV to occur). Furthermore, consideration should be given to whether the magnitude of the kick needed to instigate the MIV is possible to occur on a full-scale bridge, and if yes, what is the probability of it occurring during the *required* wind conditions. This probability should be comparable to the probability of the same kick occurring in the wind tunnel setting. Note that, due to scaling, the force needed to excite the deck model is relatively small, hence it can be argued that the likelihood of the kick occurring is larger than for the full-scale bridge. This in turn means that for the deck shapes susceptible to MIV, a great deal of attention should be paid so the tests do not yield behaviour unrealistic for the real bridge.

6.3 Deck With the Barrier

This section contains the results for deck geometries with the barriers placed on both edges (geometries SB0-SB50). The barrier detail is shown in Figure 4.5, while the geometry of the deck is the same as the geometry of the deck without the barrier (Figure 4.3 and Figure 4.4). Here, the results are divided into three groups based on the shape of the LCO regions they yielded. Firstly, the results for geometry SB0 are shown, then in the second group the results for geometries SB10 and SB25 are included and finally, the third group comprises the remaining two geometries (SB40 and SB50).

6.3.1 Geometry SB0

Figure 6.11 shows the results from the prescribed motion model simulations for geometry SB0. When compared to the results for the section without the barrier (geometry S0 in Figure 6.2) a substantial difference can be seen, the most notable of which is that at low oscillation amplitudes ($Y_r = 0.5\%$) the lock-in

is not observed. This does not necessarily mean that the lock-in does not exist, but that the region is rather narrow and was not captured with the current data resolution.

The maximum reduced amplitude observed for this geometry is 3.5%, 3% and 2.8% for three Scruton numbers, respectively. The effects of varying Sc number on the maximum oscillation amplitude are of similar magnitude as for geometry S0.

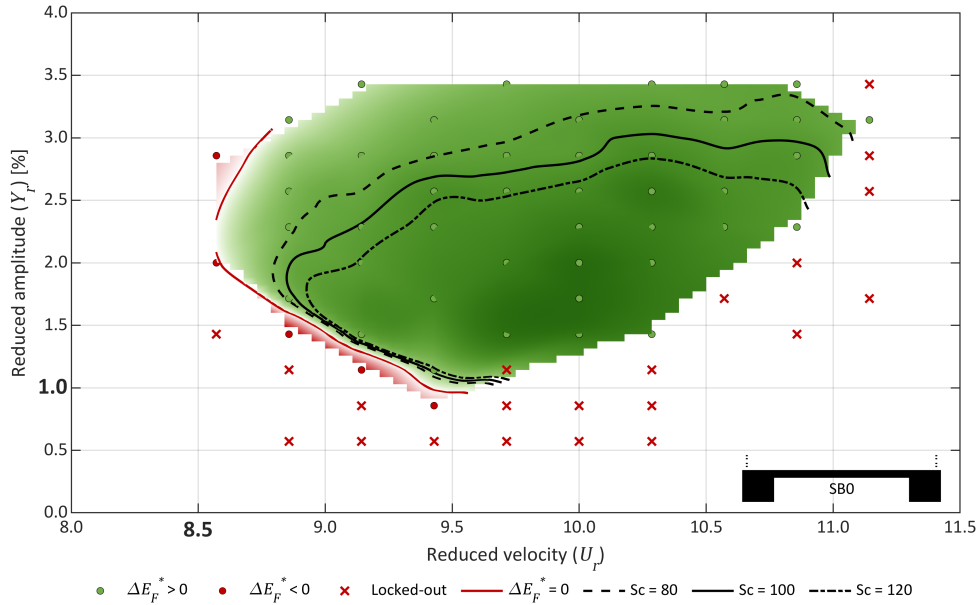


Figure 6.11. Prescribed motion model results for geometry SB0

It should be noted that for these simulations (i.e., data points for geometry SB0), the computational time needed for a flow field to converge to a state that can be considered a stable periodic, was larger by a factor of two (on average) than what was experienced for other simulations (including other SB geometries). Similar unstable behaviour is noticed in the VIV response for the Clyde Bridge deck¹ obtained in the wind tunnel (see Figure 3.11 in Section 3.2). This can be attributed to the interaction between the large blockage area and the barrier on the upwind deck edge.

6.3.2 Geometry SB10 and SB25

Unlike the results for the deck section without the overhang, the results for geometries SB10 and SB25 (Figure 6.12 and Figure 6.13) feature LCO regions of similar shape to those seen for their counterparts without the barrier (S10 in Figure 6.3 and S25 in Figure 6.4). However, when compared to results for geometry SB0, the maximum displacement amplitude has increased by 36% for SB10 and 52%, 40% and 29% for three Scruton numbers for SB25. Furthermore, the locked-in region for the lower amplitudes for both geometries is captured, although it is narrower than for the sections without the barrier (S10 and S25). The lock-in width at the reduced amplitude of 0.5% for S10 is wider than that for SB10 by at least a factor of two.

A notable difference when results for geometries SB25 and S25 are compared is that the VIV susceptibility region ($\Delta E_F^* > 0$) at higher flow velocities is defined by $\Delta E_F^* = 0$ contour line instead of the locked-out conditions. Similar behaviour is observed for geometry SB50 (and possibly for SB40 although there are no data points either confirming or discarding this).

¹ Clyde Bridge deck geometry is comparable to geometry SB0.

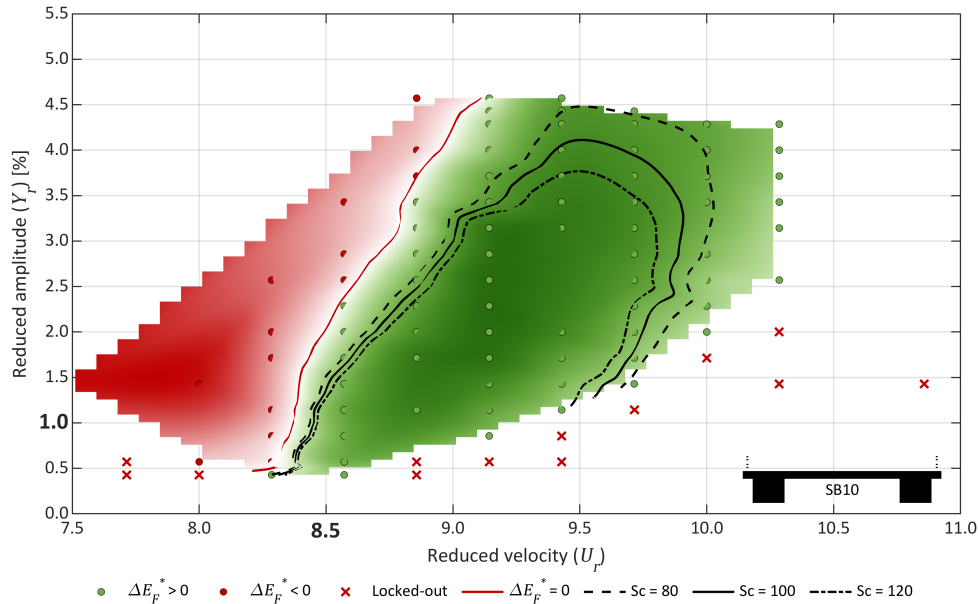


Figure 6.12. Prescribed motion model results for geometry SB10

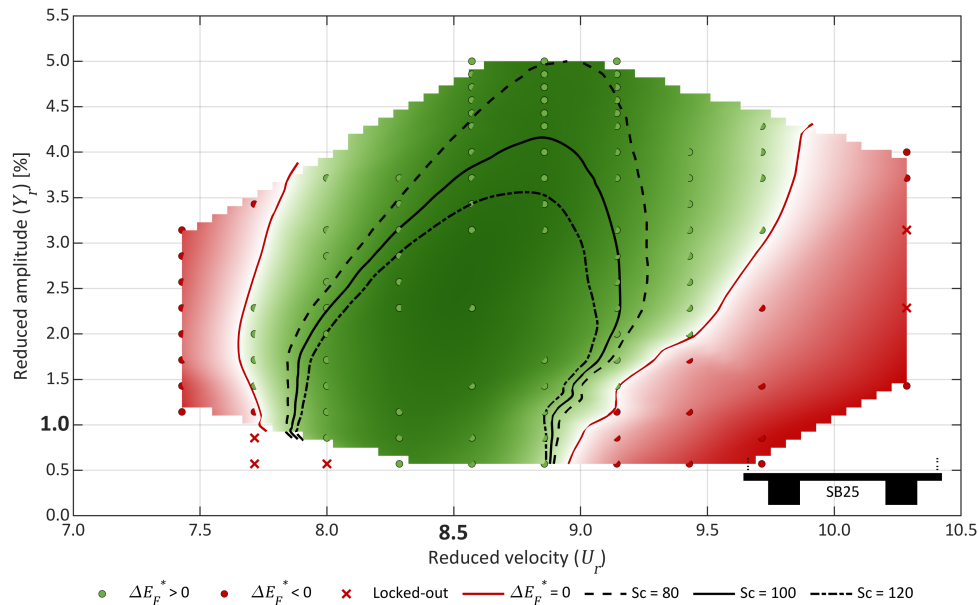


Figure 6.13. Prescribed motion model results for geometry SB25

6.3.3 Geometry SB40 and SB50

Figure 6.14 and Figure 6.15 show results for geometries SB40 and SB50, and when compared to geometry SB0, the maximum amplitude has increased by approximately 22% and 42%, respectively. The width of the lock-in region, however, has decreased by approximately 25% for geometry SB40 and 44% for geometry SB50. Similar to geometry SB0, the lock-in velocity at lower amplitudes for geometry SB40 is not captured while for geometry SB50 it is, but only at the reduced velocity of 8.6. This reinforces the statement made in Section 6.3.1 that the lack of the lock-in velocity in the data (at lower amplitudes) is most likely due to the data resolution.

Again, a comparison between geometry SB40 and the Narrow Water Bridge deck section can be done, but now with attention to the VIV response of the NWB deck with the wind shielding (Figure 3.18). Two points can be made: (i) the increase in Scruton number has a lower effect on the maximum displacement

amplitude than what it had for the section without the barrier/wind shielding, (ii) by adding the barrier (or wind shielding in the case of the NWB) the maximum response amplitude has increased by a factor of 3.5. Both of these observations are seen for geometry S40/SB40 and the Narrow Water Bridge deck.

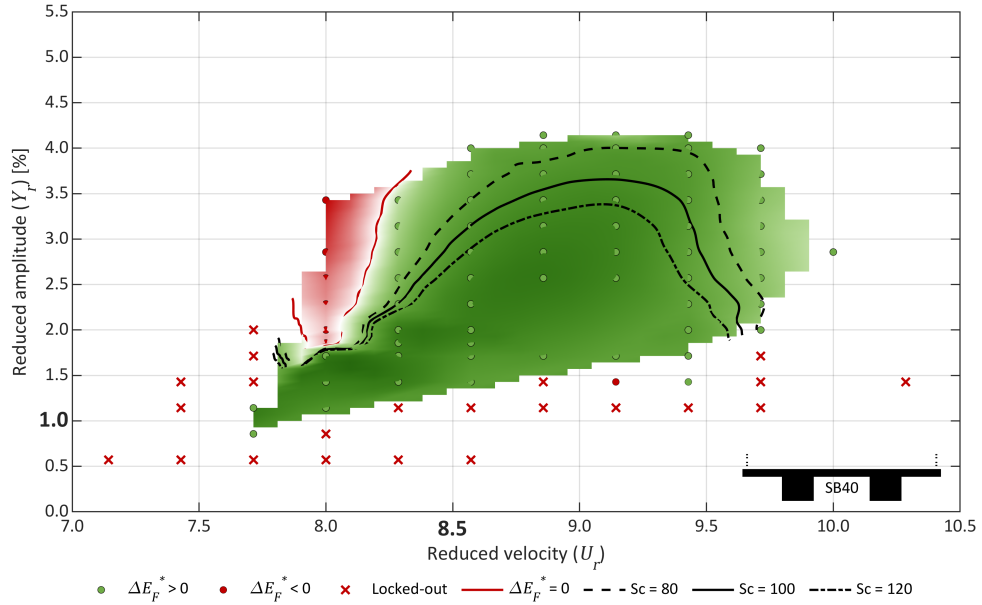


Figure 6.14. Prescribed motion model results for geometry SB40

The results for geometry SB50 are characterised by the island-like feature which was present in the results for geometries S40 and S50. Here, the upper LCO region is wider than for geometry S50 and the changing Scruton number has a somewhat smaller effect on the maximum oscillation amplitude. However, the lower LCO region (related to Karman vortex shedding) is of smaller size, although the data points for geometry SB50 are obtained at a coarser resolution than what was done for geometry S50.

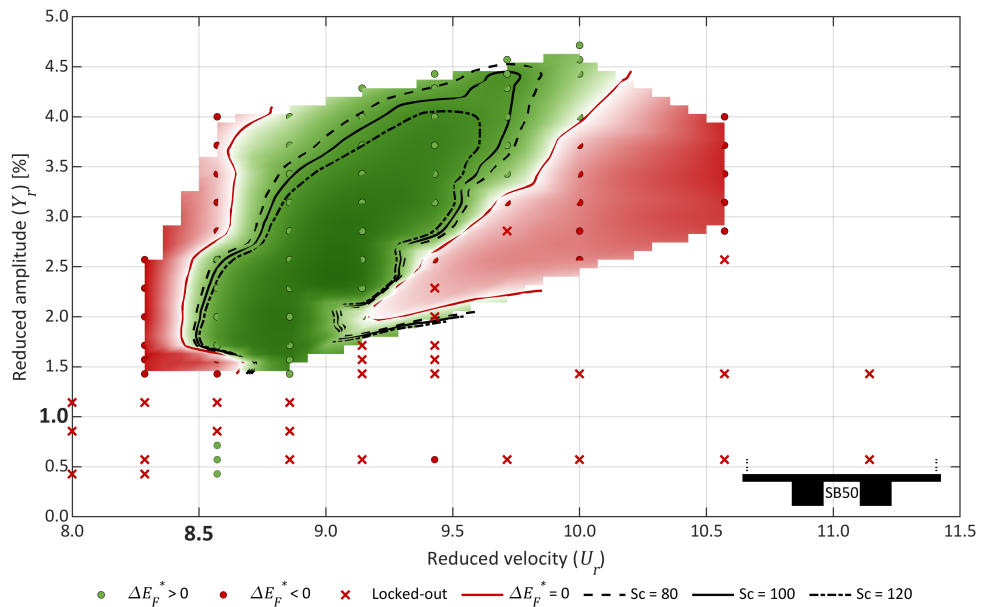


Figure 6.15. Prescribed motion model results for geometry SB50

6.3.4 Discussion

By adding the barrier, the effect of the overhang length on the VIV response has decreased in the simulation. Similar behaviour can be observed in results from wind tunnel tests (with and without the barriers). The existence of the island-like LCO region was noticed for geometry SB50 (comparable to what was found for geometry S50). It is suspected that this region is related to the motion-induced vortices. Furthermore, the results for geometry SB0 also show the existence of the upper LCO region, which was not the case for geometry S0. All SB geometries (except SB10 and SB25) experience the lack of clear lock-in flow velocity for lower oscillation amplitudes.

The aim of this study was to evaluate the effect of the overhang length on the VIV response of the deck with the barrier. A separate study on the barrier geometry should be undertaken to investigate possible effects. This may include the barrier height, porosity level and its vicinity to the deck edge.

6.4 Synthesis

The maximum amplitude and the width of the lock-in region for five geometries without the barrier (S0-S50) are given in Figure 6.16. All results are normalised with the results for geometry S0. The dashed lines mark the results related to the upper LCO region (related to the motion-induced vibrations), while the solid lines mark the results related to Karman vortex shedding (lower LCO region). As can be seen, with the increased overhang length the maximum displacement amplitude decreases with the largest drop occurring between geometries S10 and S25. With the introduction of geometry S40, the upper LCO region is observed in the results and the maximum amplitude experiences an increase compared to S10 and S25 but still a decrease when compared to geometry S0. Finally for geometry S50, the maximum oscillation amplitude is larger than for geometry S0 by a factor of approximately 1.1. Similar behaviour is observed for the lock-in region width, with the exception of being the largest for geometry S0.

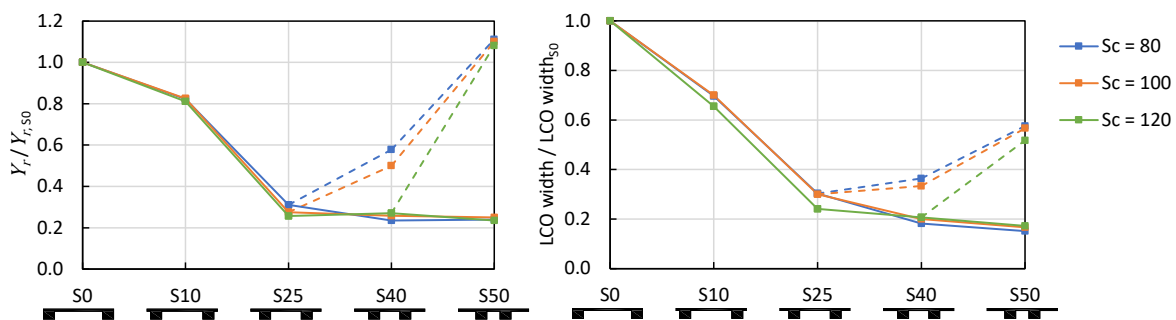


Figure 6.16. Maximum displacement amplitude (left) and lock-in region width (right) for the deck geometry without the barrier (S0-S50) normalised with values for geometry S0; solid line – lower LCO region; dashed line – upper LCO region

In Section 6.2.3 it was shown how different initial conditions govern whether the motion-induced vortices will occur or not. A separate study on the magnitude of the initial kick was also suggested. Although the MIV were observed in the CFD analysis done here, they were performed at the model scale and they occurred only after an initial kick was introduced. A clear example of this was seen in the case of the Narrow Water Bridge deck VIV response, modelled in CFD and briefly discussed in Section 5.4. The difference between the model scale analysis done in CFD and in a wind tunnel is that for the former the initial conditions can be

rigorously controlled, while in the wind tunnel environment, a small perturbation in the on-coming flow or facility vibrations can provide the kick amplitude needed for the MIV to occur. These vibrations could then be interpreted as the VIV instability of the tested section which would not have existed without the initial perturbation. Furthermore, it should be noted that this study employed turbulence models for dealing with turbulence effects. This means that turbulent buffeting is not captured which might exist in a wind tunnel.

When it comes to the results for the deck section with the barrier (SB0-SB50) shown in Figure 6.17, initially the length of the overhang has the opposite effect, i.e., the maximum amplitude increases with the introduction of the overhang (by a factor of approximately 1.4). With a further increase in the overhang length, the maximum amplitude stabilises and varies to a lesser extent but remains above the one recorded for geometry SB0. The width of the lock-in region decreases monotonously with increasing overhang length (with the exception of the change between SB25 and SB40). For these simulations, the difference between the upper and lower LCO regions is not clearly noticeable and hence was not differentiated in the graphs.

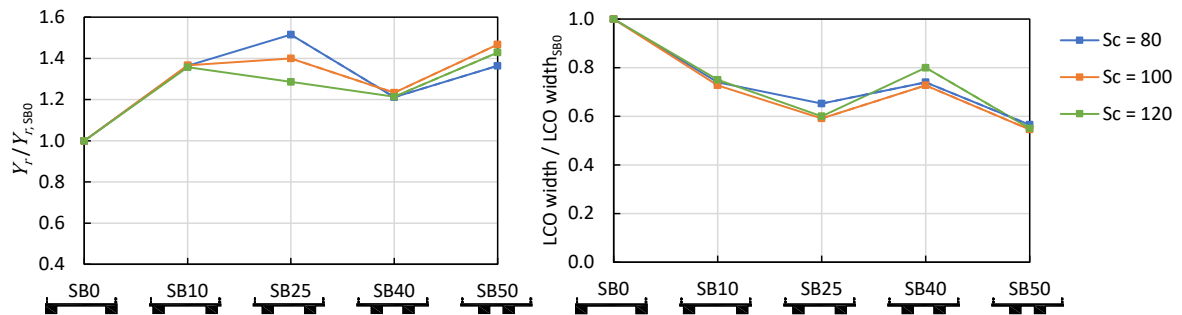


Figure 6.17. Maximum displacement amplitude (left) and lock-in region width (right) for deck geometry with the barrier (SB0-SB50) normalised with values for geometry SB0

Chapter 7

Conclusions

7.1 Summary

The first objective of this research was to investigate the feasibility of using CFD as an early design tool in bridge engineering with a focus on bluff bridge decks. As seen in Section 2.5, the presence of CFD in bridge engineering is limited and it is mostly confined to research endeavours of aerodynamically streamlined bridge decks. Here, peculiarities involved in CFD modelling and obtaining useful results for the purpose of the preliminary design of a bluff bridge deck were investigated. The methodology employed for this is presented in Chapter 4, while the detailed results are given in Chapter 5. The results showed that the barriers and girders are important for the analyses, which is in line with the previous research in this field. However, if the overall porosity of the barrier remains constant, the number of openings can be altered with little effect on the static coefficients. The possibility of extruding the 2D model to the third dimension was explored. This increased the computational power required for the simulations to run by an order of magnitude, which was not followed by an equal increase in the quality of the results. It was concluded that unless LES or an anisotropic turbulence model is employed, the benefit of 3D analyses is negligible. While most of the simulations in this section employed the Transition SST turbulence model, additional analyses using SST and $k - \varepsilon$ models were also undertaken (see Section 4.1.4 for details). By comparing the results from these three turbulence models it was shown that when the analysis is done at the Reynolds number similar to that present in the wind tunnel tests (in bridge engineering), the TSST model can outperform the other two. However, at higher Reynolds numbers the results converge towards the results obtained using the SST model. It is also noted that the TSST model has two transport equations more than the SST, hence increasing the computational time. The next point in this part of the research dealt with the VIV response of the bridge deck. Two types of models were developed, free-to-oscillate or the FSI model and the prescribed motion model. Firstly, it was shown that the FSI model can be employed in estimating the deck VIV response. The results yielded the maximum displacement amplitude and the VIV onset velocity, both of which compared reasonably well to the wind tunnel data. The overestimation in the displacement amplitude was attributed to the imposed full spanwise correlation in the CFD model. The FSI model was developed with a set of dynamic properties (mass, stiffness, and damping), meaning that, while the results were valuable if the maximum VIV amplitude is needed, the simulations would have to be repeated should the dynamic properties of the model require adjusting. As an alternative to the FSI model, the PM model coupled with the energy approach (see Section 4.4) offers the possibility to perform the CFD simulations independently of the model's dynamic properties. Once the CFD results are obtained any combination of mass-damping parameters can be applied yielding the system's VIV response (amplitude against flow velocity). It was found that the results obtained in this way were equivalent to the results delivered by the FSI model. Furthermore, the PM model produced data points faster than the

FSI model, by a factor of 2. However, the former approach requires a larger number of data points, hence when the sensitivity to structural damping is not of interest, the FSI model will yield results faster.

Finally, a methodology was proposed which may be followed should CFD be used at the preliminary design stage. First, a simplified geometry of the deck section to be tested is defined. This geometry should include any barriers which should be modelled with the porosity present in the full-scale model. Then the computational mesh is created which should be divided in such a way as to allow for easy cell size control and possible re-meshing. A mesh dependency study should be undertaken and the y^+ value verified. This is followed by the simulations on the static model. These results will give an estimate of the Strouhal number and the static force coefficients for a range of angles of attack which then may be used for evaluation of the deck's susceptibility to divergent response (Den Hartog criterion). Should any issues be identified, the geometry should be altered and previous steps repeated. Finally, simulations on the dynamic model may be undertaken which will identify any possible issues with the vortex-induced vibrations. The process described here is summarised in Figure 7.1.

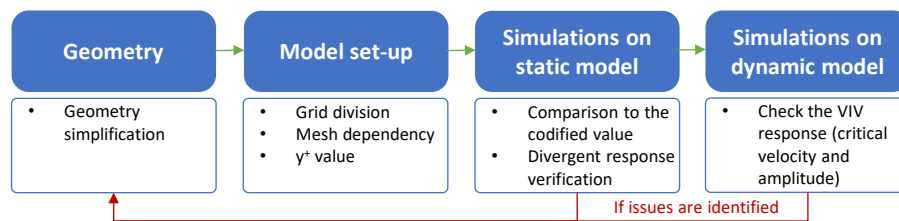


Figure 7.1. Methodology for CFD application in bridge engineering at the preliminary design stage

The prescribed motion model approach, introduced and tested in the first part of the thesis, was then used for the second objective and results were presented in Chapter 6. Here, the effects of the overhang on the VIV response of the bluff bridge decks are investigated. In Section 2.2 it was shown that the vortex-shedding process can be affected by placing a horizontal plate in the wake of a bluff body. It is suspected that a similar effect can be caused by the bridge overhang. For the purpose of this study, two sets of geometries were developed, with and without the barriers. Each set comprised five geometries with varying overhang lengths (see Section 4.2 for details). In the case of the bridge deck without the barrier, it was found that with increasing overhang length the maximum displacement amplitude dramatically decreases until a second LCO region begins to occur. Due to its characteristics, it is suspected that this LCO region is governed by motion-induced vortices. Furthermore, it was shown that unless an initial displacement amplitude is introduced, the system cannot reach this LCO state. At the end of Section 6.2, the results from FSI-PM simulations showed how initial kicks of different sizes yield different maximum VIV amplitudes. The possibility of the MIV occurring on a full-scale bridge is discussed and attention was drawn to the increased likelihood of its occurrence at the model scale. Other than the maximum amplitude, the width of the lock-in region also experienced a decrease with increasing overhang length. All results were presented for three levels of Scruton number. The change in Sc number affected the VIV response more or less equally for all geometries with slightly larger effects present for geometries with longer overhangs. When it comes to geometries with the barrier, the results suggested that with the introduction of the overhang, the maximum displacement amplitude increases. However, with the further increase in its length, the VIV response

experienced a reduction, albeit it was still larger than for the initial geometry. The width of the lock-in region decreased with the overhang introduction and with subsequent increases in its length. Unlike for the cases without the barrier, the upper region was not clearly visible although some similar characteristics were observed.

7.2 Future Work

The first part of this work showed a large potential for CFD to be used as a tool at the preliminary design stage of bridge decks. The analyses can provide static force coefficients but also a reasonable estimate of the dynamic response. Furthermore, the prescribed motion model coupled with the energy approach displayed the possibility for further refinement. The goal of this would be to minimise the number of data points needed for the results to yield useful information for the preliminary design stage. In this research, the interpolation between the data points was utilised at a very low level, however further work can analyse different fitting functions and quantify their suitability for a given deck shape. A step further can be made with the introduction of the data extrapolation which was avoided in this study. Finally, extensive testing and/or simulation campaigns for a number of different deck shapes can be undertaken. This will generate a large amount of data comprising the energy surfaces for each analysed deck shape. This data could then be fed to an AI algorithm which would *learn* the dependency between deck shapes and the energy surfaces. Ultimately, engineers would be in a position to obtain the energy surface for a deck shape simply by providing its shape to the algorithm.

As noted in Chapter 6, the influence of the barrier on the VIV response for different overhang lengths has been studied here only to a limited extent. Further research should include the height, porosity, and the angle of the barrier but also the spacing between the barriers relative to the overall bridge width. While the structural bridge width is defined by the specific needs the bridge is to cater for (number of traffic lanes, footpaths, cycleways, and others), the length of the overhangs may be influenced by the aesthetics requirements too. As seen in the case of the Clyde Bridge, the total width of the bridge was changed effortlessly with the introduction of the non-structural elements, in this case, fairing plates. Similarly, it is not uncommon that the leading and/or trailing edges are fitted with horizontal elements in order to hide the visual impact of the full bridge depth. Furthermore, it is noted that in this study, the depth of the overhangs was taken as constant along the bridge width, while in practice this does not have to be the case. The distance between the barriers relative to the overall bridge width, and depth, may affect the VIV response by altering the spatial characteristics of the vortices and hence affecting the shedding frequency. Another geometric characteristic of the barriers worth exploring is their angle relative to the deck. Inclined barriers will disperse the vortices further away from the deck's top surface, which will again cause a change in the shedding frequency. By altering the shedding frequency, the VIV response is affected by a shift in the onset velocity. Finally, the ultimate goal of this research should be the introduction of the overhang length in the design code as a deck shape parameter which would inform on the level of the deck VIV response.

The results in Chapter 6 suggested that some deck shapes may experience motion induced vibrations if an initial kick is introduced, and that this response can be mistaken for the Karmen vortex shedding phenomenon. It was argued that, due to the scaling, the likelihood of MIV occurring in the wind

tunnel testing is larger than for the full-scale bridge prototypes. As a consequence, this *opens the door* to wind tunnel testing yielding an overestimated VIV response. Furthermore, the design codes are almost always based on wind tunnel tests at model scale which may make them systematically conservative. The existence of MIV behaviour should be further investigated either in the wind tunnel tests at larger scales or by undertaking full-scale CFD simulations. It is noted that, while a great deal of attention was paid to CFD modelling in this research, the turbulence was modelled, and not resolved. This means that the turbulence buffeting was neglected, which would normally exist in a wind tunnel setting. Should future research show that, in fact for certain deck shapes the MIV may manifest itself in the wind tunnel tests and that the main reason for this is a kick in the displacement, the probability of its occurrence at full scale during the *correct* wind conditions must not be neglected.

Bibliography

- AASHTO. (2020). *LRFD Bridge Design Specifications* (9th ed.). The American Association of State Highway and Transportation Officials.
- Akhter, M. Z., & Mays, R. C. (2019). Prescribed motion flow dynamics. *Journal of Mechanical Science and Technology*, 33(1), 289–297. <https://doi.org/10.1007/s12206-018-1228-z>
- Andersson, B., Andersson, R., Håkansson, L., Mortensen, M., Sudiyo, R., & Van Wachem, B. (2011). Computational fluid dynamics for engineers. In *Computational Fluid Dynamics for Engineers* (Vol. 9781107018). <https://doi.org/10.1017/CBO9781139093590>
- Ansys. (2021a). *ANSYS Fluent UDF Manual, Release 2021 R1*.
- Ansys. (2021b). *ANSYS Fluent User's Guide, Release 2021 R1*.
- AS 5100-2:2017. (n.d.). *Bridge Design - Part 2: Design Loads*. Standards Australia, Austroads.
- ASCE. (1961). Wind Forces on Structures. *Transactions of the American Society of Civil Engineers*, 126(2), 1124–1198. <https://doi.org/10.1061/TACEAT.0008256>
- ASCE. (1992). *Guidelines for the Design of Cable-Stayed Bridges*. American Society of Civil Engineers.
- Battista, R. C., & Pfeil, M. S. (2000). Reduction of vortex-induced oscillations of Rio-Niteroi bridge by dynamic control devices. *Journal of Wind Engineering and Industrial Aerodynamics*, 84(3), 273–288. [https://doi.org/10.1016/S0167-6105\(99\)00108-7](https://doi.org/10.1016/S0167-6105(99)00108-7)
- Benicke, O., & Butz, C. (2015). Volgograd-bridge: Efficiency of passive and adaptive tuned mass dampers. *IABSE Conference, Geneva 2015: Structural Engineering: Providing Solutions to Global Challenges - Report*, 1428–1435. <https://doi.org/10.2749/222137815818358826>
- Blevins, R. D. (2001). *Flow-Induced Vibration*. Krieger Publishing Company, Malabar.
- Bruno, L., & Khriis, S. (2003). The validity of 2D numerical simulations of vortical structures around a bridge deck. *Mathematical and Computer Modelling*, 37(7–8), 795–828. [https://doi.org/10.1016/S0895-7177\(03\)00087-6](https://doi.org/10.1016/S0895-7177(03)00087-6)
- Bruno, L., & Mancini, G. (2002). Importance of deck details in bridge aerodynamics. *Structural Engineering International: Journal of the International Association for Bridge and Structural Engineering (IABSE)*, 12(4), 289–294. <https://doi.org/10.2749/101686602777965234>
- Brusiani, F., De Miranda, S., Patruno, L., Ubertini, F., & Vaona, P. (2013). On the evaluation of bridge deck flutter derivatives using RANS turbulence models. *Journal of Wind Engineering and Industrial Aerodynamics*, 119, 39–47. <https://doi.org/10.1016/j.jweia.2013.05.002>
- Carusone, A., & Flamand, O. (2022). *Aerodynamic stability of Clyde Bridge*. Report to ROD (available on request from the author)
- Chen, S.-S. (1985). *Flow-Induced Vibration of Circular Cylindrical Structures*. <https://doi.org/10.2172/6331788>
- Daniels, S. J., Castro, I. P., & Xie, Z. T. (2016). Numerical analysis of freestream turbulence effects on the vortex-induced vibrations of a rectangular cylinder. *Journal of Wind Engineering and Industrial Aerodynamics*, 153, 13–25. <https://doi.org/10.1016/j.jweia.2016.03.007>

- Davenport, A. G. (2002). Past, present and future of wind engineering. *Journal of Wind Engineering and Industrial Aerodynamics*, 90(12–15), 1371–1380. [https://doi.org/10.1016/S0167-6105\(02\)00383-5](https://doi.org/10.1016/S0167-6105(02)00383-5)
- Den Hartog, J. P. (1957). Mechanical Vibrations. Fourth Edition. J. P. Den Hartog. McGraw-Hill, New York, 1956. 67s. 6d. *The Journal of the Royal Aeronautical Society*, 61(554), 139–139. <https://doi.org/10.1017/S0368393100131049>
- Deniz, S., & Staubli, T. (1997). Oscillating rectangular and octagonal profiles: Interaction of leading- and trailing-edge vortex formation. *Journal of Fluids and Structures*, 11(1), 3–31. <https://doi.org/10.1006/jfls.1996.0065>
- Duranovic, M., Dempsey, T., & Meskell, C. (2022a). A Practical Approach to Using CFD as an Early Design Tool for Estimating Aerodynamic Force Coefficients of Bridge Decks. *Flow Induced Vibration, Paris-Saclay 2022*.
- Duranovic, M., Dempsey, T., & Meskell, C. (2022b). Modelling Vortex Induced Vibrations in a Model of the Northern Spire Bridge. *Flow Induced Vibration, Paris-Saclay 2022*.
- Duranovic, M., Dempsey, T., & Meskell, C. (2024). CFD as an early aerodynamic design tool for estimating aerodynamic force coefficients and vortex-induced vibrations of bluff bridge decks. *Proceedings of the Institution of Civil Engineers: Bridge Engineering*. (under review)
- Duranovic, M., & Meskell, C. (2024). Effects of the overhang length on the vortex induced vibration response of bridge decks. *Proceedings of the Institution of Civil Engineers: Bridge Engineering*. (under review)
- EN 1991-1-4. (2005). *Eurocode 1: Actions on Structures. Part 1-4: General Actions - Wind Actions*. European Committee for Standardisation (CEN).
- Feng, C. C. (1968). *The measurement of vortex induced effects in flow past stationary and oscillating circular and D-section cylinders* [MSc Thesis, University of British Columbia]. <https://doi.org/https://dx.doi.org/10.14288/1.0104049>
- Flamand, O., & Knapp, G. (2018). *New Wear Bridge - Wind Study & Wind Tunnel Tests*. Report to ROD (available on request from the author)
- Flint & Neil, & TRL. (1986). *Partial Safety Factors for Bridge Aerodynamics Rules and Requirements for Wind Tunnel Testing*. Transport and Road Research Laboratory.
- Fujino, Y., Kimura, K., & Tanaka, H. (2012). *Wind Resistant Design of Bridges in Japan*. Springer Japan. <https://doi.org/10.1007/978-4-431-54046-5>
- Fujino, Y., & Siringoringo, D. (2013). Vibration mechanisms and controls of long-span bridges: A review. *Structural Engineering International: Journal of the International Association for Bridge and Structural Engineering (IABSE)*, 23(3), 248–268. <https://doi.org/10.2749/101686613X13439149156886>
- Ge, Y. J., & Xiang, H. F. (2008). Computational models and methods for aerodynamic flutter of long-span bridges. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(10–11), 1912–1924. <https://doi.org/10.1016/j.jweia.2008.02.017>
- Ge, Y., Zhao, L., & Cao, J. (2022). Case study of vortex-induced vibration and mitigation mechanism for a long-span suspension bridge. *Journal of Wind Engineering and Industrial Aerodynamics*, 220(December 2021), 104866. <https://doi.org/10.1016/j.jweia.2021.104866>

- Gerhart, P. M., Gerhart, A. L., & Hochstein, J. I. (2016). *Munson, Young and Okiishi's Fundamentals of Fluid Mechanics* (8th ed.). Wiley, Hoboken.
- Goering, A., & Ramponi, R. (2019). Wind Analysis of Long-Span Bridges Using Computational Fluid Dynamics. *Structures Congress 2019: Bridges, Nonbuilding and Special Structures, and Nonstructural Components - Selected Papers from the Structures Congress 2019*, 210–220. <https://doi.org/10.1061/9780784482230.021>
- Goswami, I., Scanlan, R. H., & Jones, N. P. (1993). Vortex-Induced Vibration of Circular Cylinders. I: Experimental Data. *Journal of Engineering Mechanics*, 119(11), 2270–2287. [https://doi.org/10.1061/\(ASCE\)0733-9399\(1993\)119:11\(2270\)](https://doi.org/10.1061/(ASCE)0733-9399(1993)119:11(2270))
- Harper, R. F. (1904). *The code of Hammurabi, King of Babylon, about 2250 B.C: autographed text, transliteration, translation, glossary, index of subjects, lists of proper names, signs, numerals, corrections and erasures, with map, frontispiece and photograph of text*. University of Chicago Press : Callaghan & Co.
- Hirt, C. W., Amsden, A. A., & Cook, J. L. (1974). An arbitrary Lagrangian-Eulerian computing method for all flow speeds. *Journal of Computational Physics*, 14(3), 227–253. [https://doi.org/10.1016/0021-9991\(74\)90051-5](https://doi.org/10.1016/0021-9991(74)90051-5)
- IS 875-3:2015. (n.d.). *Design Loads (Other than Earthquake) for Buildings and Structures - Code of Practice*. Bureau of Indian Standards.
- Japan Road Association. (2007). *Wind resistant design manual for highway bridges (in Japanese)*. Japan Road Association.
- Jeong, W., Liu, S., Jakobsen, J. B., & Ong, M. C. (2019). Unsteady RANS simulations of flow around a twin-box bridge girder cross section. *Energies*, 12(14). <https://doi.org/10.3390/en12142670>
- Jiao, H., & Wu, G. X. (2018). Free vibration predicted using forced oscillation in the lock-in region. *Physics of Fluids*, 30(11). <https://doi.org/10.1063/1.5056203>
- Kawatani, M., Kim, H., Uejima, H., & Kobayashi, H. (1993). Effects of turbulent flows on vortex-induced oscillation of bridge girders with basic sections. *Journal of Wind Engineering and Industrial Aerodynamics*, 49(1–3), 477–486. [https://doi.org/10.1016/0167-6105\(93\)90042-M](https://doi.org/10.1016/0167-6105(93)90042-M)
- Larose, G. L., & D'Auteuil, A. (2006). On the Reynolds number sensitivity of the aerodynamics of bluff bodies with sharp edges. *Journal of Wind Engineering and Industrial Aerodynamics*, 94(5), 365–376. <https://doi.org/10.1016/j.jweia.2006.01.011>
- Larsen, A., Esdahl, S., Andersen, J. E., & Vejrum, T. (2000). Storebaelt suspension bridge - vortex shedding excitation and mitigation by guide vanes. *Journal of Wind Engineering and Industrial Aerodynamics*, 88(2–3), 283–296. [https://doi.org/10.1016/S0167-6105\(00\)00054-4](https://doi.org/10.1016/S0167-6105(00)00054-4)
- Larsen, A., Savage, M., Lafrenière, A., Hui, M. C. H., & Larsen, S. V. (2008). Investigation of vortex response of a twin box bridge section at high and low Reynolds numbers. *Journal of Wind Engineering and Industrial Aerodynamics*, 96(6–7), 934–944. <https://doi.org/10.1016/j.jweia.2007.06.020>

- Larsen, A., & Wall, A. (2012). Shaping of bridge box girders to avoid vortex shedding response. *Journal of Wind Engineering and Industrial Aerodynamics*, 104–106, 159–165. <https://doi.org/10.1016/j.jweia.2012.04.018>
- 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*, 1(2), 131–137. [https://doi.org/10.1016/0094-4548\(74\)90150-7](https://doi.org/10.1016/0094-4548(74)90150-7)
- Li, H., Laima, S., Zhang, Q., Li, N., & Liu, Z. (2014). Field monitoring and validation of vortex-induced vibrations of a long-span suspension bridge. *Journal of Wind Engineering and Industrial Aerodynamics*, 124, 54–67. <https://doi.org/10.1016/j.jweia.2013.11.006>
- Li, Y. Le, Chen, X. Y., Yu, C. J., Togbenou, K., Wang, B., & Zhu, L. D. (2018). Effects of wind fairing angle on aerodynamic characteristics and dynamic responses of a streamlined trapezoidal box girder. *Journal of Wind Engineering and Industrial Aerodynamics*, 177(April), 69–78. <https://doi.org/10.1016/j.jweia.2018.04.006>
- Li, M., Sun, Y., Lei, Y., & Li, M. (2020). Experimental studies on the vortex-induced forces of a 4:1 rectangular cylinder and two typical bridge girders. *Advances in Structural Engineering*, 23(3), 399–410. <https://doi.org/10.1177/1369433219873985>
- Li, Y., Hu, P., Xu, X., & Qiu, J. (2017). Wind characteristics at bridge site in a deep-cutting gorge by wind tunnel test. *Journal of Wind Engineering and Industrial Aerodynamics*, 160(November 2016), 30–46. <https://doi.org/10.1016/j.jweia.2016.11.002>
- Lodefier, K., Merci, B., de Langhe, C., & Dick, E. (2004). Transition modelling with the k- Ω turbulence model and an intermittency transport equation. *Journal of Thermal Science*, 13(3), 220–225. <https://doi.org/10.1007/s11630-004-0035-2>
- Mannini, C., Šoda, A., Voß, R., & Schewe, G. (2010). Unsteady RANS simulations of flow around a bridge section. *Journal of Wind Engineering and Industrial Aerodynamics*, 98(12), 742–753. <https://doi.org/10.1016/j.jweia.2010.06.010>
- Matsuda, K., Cooper, K. R., Tanaka, H., Tokushige, M., & Iwasaki, T. (2001). An investigation of Reynolds number effects on the steady and unsteady aerodynamic forces on a 1:10 scale bridge deck section model. *Journal of Wind Engineering and Industrial Aerodynamics*, 89(7–8), 619–632. [https://doi.org/10.1016/S0167-6105\(01\)00062-9](https://doi.org/10.1016/S0167-6105(01)00062-9)
- Matsumoto, M., Shiraishi, N., Shirato, H., Stoyanoff, S., & Yagi, T. (1993). Mechanism of, and turbulence effect on vortex-induced oscillations for bridge box girders. *Journal of Wind Engineering and Industrial Aerodynamics*, 49(1–3), 467–476. [https://doi.org/10.1016/0167-6105\(93\)90041-L](https://doi.org/10.1016/0167-6105(93)90041-L)
- Menter, F. R. (1994). Two-equation eddy-viscosity turbulence models for engineering applications. *AIAA Journal*, 32(8), 1598–1605. <https://doi.org/10.2514/3.12149>
- Miyata, T. (2003). Historical view of long-span bridge aerodynamics. *Journal of Wind Engineering and Industrial Aerodynamics*, 91(12–15), 1393–1410. <https://doi.org/10.1016/j.jweia.2003.09.033>

- Murakami, S., & Mochida, A. (1995). On turbulent vortex shedding flow past 2D square cylinder predicted by CFD. *Journal of Wind Engineering and Industrial Aerodynamics*, 54–55(C), 191–211. [https://doi.org/10.1016/0167-6105\(94\)00043-D](https://doi.org/10.1016/0167-6105(94)00043-D)
- NA to IS EN 1991-1-4. (2013). *Irish National Annex to Eurocode 1: Actions on structures - Part 1-4: General Actions - Wind Actions*. National Standards Authority of Ireland.
- Nakamura, Y., Ohya, Y., & Tsuruta, H. (1991). Experiments on vortex shedding from flat plates with square leading and trailing edges. *Journal of Fluid Mechanics*, 222, 437–447. <https://doi.org/10.1017/S0022112091001167>
- Namiranian, F. (1982). *Fluctuating Lift On Cylinders of Rectangular Cross Section in Smooth and Turbulent Flows* (Vol. 46, Issue 2) [University of Washington, USA]. <http://eprints.uanl.mx/5481/1/1020149995.PDF>
- NASA MODIS Rapid Response System. (2010). *Vortices off Tristan da Cunha, Jul 14 2010 1100(UTC)*. <http://rapidfire.sci.gsfc.nasa.gov/cgi-bin/imagery/single.cgi?image=TristandaCunha.A2010195.1100.250m.jpg>
- Nguyen, D. T., Hargreaves, D. M., & Owen, J. S. (2018). Vortex-induced vibration of a 5:1 rectangular cylinder: A comparison of wind tunnel sectional model tests and computational simulations. *Journal of Wind Engineering and Industrial Aerodynamics*, 175(February), 1–16. <https://doi.org/10.1016/j.jweia.2018.01.029>
- Nieto, F., Hernández, S., & Jurado, J. Á. (2009). Aerodynamic study of the preliminary design of a 425 m cable-stayed bridge deck using CFD. *5th European and African Conference on Wind Engineering, EACWE 5, Proceedings, January 2015*.
- Nieto, F., Owen, J. S., Hargreaves, D. M., & Hernández, S. (2015). Bridge deck flutter derivatives: Efficient numerical evaluation exploiting their interdependence. *Journal of Wind Engineering and Industrial Aerodynamics*, 136, 138–150. <https://doi.org/10.1016/j.jweia.2014.11.006>
- Païdoussis, M. P., Price, S. J., & de Langre, E. (2010). Fluid-structure interactions: Cross-flow-induced instabilities. In *Fluid-Structure Interactions: Cross-Flow-Induced Instabilities* (Vol. 9780521119). Cambridge University Press. <https://doi.org/10.1017/CBO9780511760792>
- Paidoussis, Price, S. J., & De Langre, E. (2014). *Fluid-Structure Interactions*.
- PD 6688-1-4. (2015). *Background information to the National Annex to BS EN 1991-1-4 and additional guidance*. The British Standards Institution.
- Placzek, A., Sigrist, J.-F., & Hamdouni, A. (2009). Numerical simulation of an oscillating cylinder in a cross-flow at low Reynolds number: Forced and free oscillations. *Computers & Fluids*, 38(1), 80–100. <https://doi.org/10.1016/j.compfluid.2008.01.007>
- Pritchard, P. J., & Mitchell, J. W. (2015). *Fox and McDonald's Introduction to Fluid Mechanics* (9th ed.). John Wiley & Sons Inc., New York.
- Ralph, A., Sang-Won, K., You-Chan, H., & Ho-Kyung, K. (2016). Effect of Turbulence Length Scale on Vortex Induced Vibration of Twin Deck Bridge Section. *Advances in Civil, Environmental, and Materials Research*. <https://hdl.handle.net/10371/124339>

- Roberts, S. (2013). *Wind wizard : Alan G. Davenport and the art of wind engineering*. Princeton University Press, Princeton and Oxford.
- Rodi, W. (1997). Comparison of LES and RANS calculations of the flow around bluff bodies. *Journal of Wind Engineering and Industrial Aerodynamics*, 69–71, 55–75. [https://doi.org/10.1016/S0167-6105\(97\)00147-5](https://doi.org/10.1016/S0167-6105(97)00147-5)
- Sahu, T. R., Furquan, M., Jaiswal, Y., & Mittal, S. (2019). Flow-induced vibration of a circular cylinder with rigid splitter plate. *Journal of Fluids and Structures*, 89, 244–256. <https://doi.org/10.1016/j.jfluidstructs.2019.03.015>
- Scanlan, R. H. (1975). *Recent methods in the application of test results to the wind design of long, suspended-span bridges*. Federal Highway Administration.
- Schewe, G. (2013). Reynolds-number-effects in flow around a rectangular cylinder with aspect ratio 1:5. *Journal of Fluids and Structures*, 39, 15–26. <https://doi.org/10.1016/j.jfluidstructs.2013.02.013>
- Schewe, G., & Larsen, A. (1998). Reynolds number effects in the flow around a bluff bridge deck cross section. *Journal of Wind Engineering and Industrial Aerodynamics*, 74–76, 829–838. [https://doi.org/10.1016/S0167-6105\(98\)00075-0](https://doi.org/10.1016/S0167-6105(98)00075-0)
- Shiraishi, N., & Matsumoto, M. (1983). On classification of vortex-induced oscillation and its application for bridge structures. *Journal of Wind Engineering and Industrial Aerodynamics*, 14(1–3), 419–430. [https://doi.org/10.1016/0167-6105\(83\)90043-0](https://doi.org/10.1016/0167-6105(83)90043-0)
- Simiu, E., & Yeo, D. (2019). *Wind Effects on Structures Modern Structural Design for Wind* (4th ed.). Wiley, Blackwell.
- Spalart, P., & Allmaras, S. (1992). A one-equation turbulence model for aerodynamic flows. *30th Aerospace Sciences Meeting and Exhibit*, 23. <https://doi.org/10.2514/6.1992-439>
- Stathopoulos, T., & Surry, D. (1983). Scale effects in wind tunnel testing of low buildings. *Journal of Wind Engineering and Industrial Aerodynamics*, 13(1–3), 313–326. [https://doi.org/10.1016/0167-6105\(83\)90152-6](https://doi.org/10.1016/0167-6105(83)90152-6)
- Staubli, T. (1983). Calculation of the vibration of an elastically mounted cylinder using experimental data from forced oscillation. *Journal of Fluids Engineering, Transactions of the ASME*, 105(2), 225–229. <https://doi.org/10.1115/1.3240968>
- Sun, Y., Li, M., Li, M., & Liao, H. (2019). Spanwise correlation of vortex-induced forces on typical bluff bodies. *Journal of Wind Engineering and Industrial Aerodynamics*, 189(April), 186–197. <https://doi.org/10.1016/j.jweia.2019.03.020>
- Svend Ole Hansen ApS. (2023). *Narrow Water Bridge - Wind Study & Wind Tunnel Tests*. Report to ROD (available on request from the author)
- Theodorsen, T. (1935). General Theory of Aerodynamic Instability and the Mechanism of Flutter. *Naca Report No 496*, 58(12), 413–433.
- THINK Fluid Dynamix. (2017). *Time dependent CFD Simulation: Vortex Shedding*. <https://www.youtube.com/watch?app=desktop&v=uqUUbe4GT60>
- TRL. (2002). *Background to the development of BD 49/01: Design rules for aerodynamic effects on bridges*.

- Vairo, G. (2003). A numerical model for wind loads simulation on long-span bridges. *Simulation Modelling Practice and Theory*, 11(5–6), 315–351. [https://doi.org/10.1016/S1569-190X\(03\)00053-4](https://doi.org/10.1016/S1569-190X(03)00053-4)
- Versteeg, H. K., & Malalasekera, W. (2007). *An Introduction to Computational Fluid Dynamics* (2nd ed.). Prentice Hall.
- Vickery, B. J. (1966). Fluctuating lift and drag on a long cylinder of square cross-section in a smooth and in a turbulent stream. *Journal of Fluid Mechanics*, 25(3), 481–494. <https://doi.org/10.1017/S002211206600020X>
- Wilcox, D. C. (1988). Reassessment of the scale-determining equation for advanced turbulence models. *AIAA Journal*, 26(11), 1299–1310. <https://doi.org/10.2514/3.10041>
- Williamson, C. H. K., & Govardhan, R. (2004). Vortex-induced vibrations. *Annual Review of Fluid Mechanics*, 36(1982), 413–455. <https://doi.org/10.1146/annurev.fluid.36.050802.122128>
- Yin, R. T., Zhu, B., Yang, Z. Y., Yang, Z. Y., Guo, C., & Cui, S. (2022). Experimental study of aerodynamic force coefficients for a bridge deck sitting above water waves. *Journal of Wind Engineering and Industrial Aerodynamics*, 226(May), 105030. <https://doi.org/10.1016/j.jweia.2022.105030>
- Zdravkovich, M. M. (1997). *Flow Around Circular Cylinders*. Oxford University Press.
- Zhan, Q. L., Zhou, Z. Y., Yang, T., & Ge, Y. J. (2014). Vortex Induced Vibration Analysis of Twin Box Girders by Means of Computational Fluid Dynamics. *Applied Mechanics and Materials*, 638–640, 1012–1017. <https://doi.org/10.4028/www.scientific.net/AMM.638-640.1012>
- Zhang, H., Xin, D., & Ou, J. (2019). Wake control using spanwise-varying vortex generators on bridge decks: A computational study. *Journal of Wind Engineering and Industrial Aerodynamics*, 184(November 2018), 185–197. <https://doi.org/10.1016/j.jweia.2018.11.031>
- Zhang, T., Sun, Y., Li, M., & Yang, X. (2020). Experimental and numerical studies on the vortex-induced vibration of two-box edge girder for cable-stayed bridges. *Journal of Wind Engineering and Industrial Aerodynamics*, 206(February), 104336. <https://doi.org/10.1016/j.jweia.2020.104336>
- Zhang, Y., Cardiff, P., Cahill, F., & Keenahan, J. (2021). Assessing the Capability of Computational Fluid Dynamics Models in Replicating Wind Tunnel Test Results for the Rose Fitzgerald Kennedy Bridge. *CivilEng*, 2(4), 1065–1090. <https://doi.org/10.3390/civileng2040057>
- Zhang, Y., Sweeney, C., Cardiff, P., Cahill, F., & Keenahan, J. (2023). Quantifying the impact of bridge geometry and surrounding terrain: Wind effects on bridges. *Proceedings of the Institution of Civil Engineers: Bridge Engineering*, 1–18. <https://doi.org/10.1680/jbren.23.00005>
- Zhou, Z., Mao, W., & Ding, Q. (2019). Experimental and numerical studies on flutter stability of a closed box girder accounting for ground effects. *Journal of Fluids and Structures*, 84, 1–17. <https://doi.org/10.1016/j.jfluidstructs.2018.09.009>
- Zhu, Q., Chen, B. Y., Zhu, L. D., & Xu, Y. L. (2017). Investigation on characteristics and span-wise correlation of vortex-induced forces on a twin-box deck using newly-developed wind-tunnel test technique. *Journal of Wind Engineering and Industrial Aerodynamics*, 164(April 2016), 69–81. <https://doi.org/10.1016/j.jweia.2017.02.009>

Appendix A

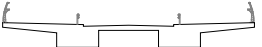
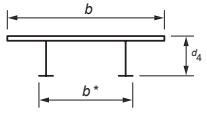
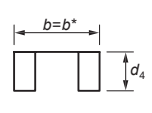
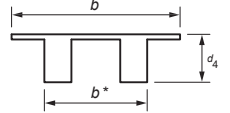
Effects of Turbulence Intensity on the VIV of Bridges

This appendix contains further results from wind tunnel testing campaigns described in Chapter 3. During their execution, it was not envisaged that the results would be used for research purposes thus the output of the tests is limited to what is required by the design and certification procedures. The results for each bridge model are presented in their respective Factual Reports which are not publicly available. The data presented here are not the final conclusions and recommendations of the testing campaigns.

A deck model of the Northern Spire Bridge (see Figure 3.2) and the Clyde Bridge¹ (see Figure 3.8) were fabricated and tested by Centre Scientifique et Technique du Bâtiment in their Boundary Layer wind tunnel facility in Nantes, France. The third deck model, for the purpose of designing the Narrow Water Bridge (see Figure 3.15), was developed and tested by Svend Ole Hansen ApS in Copenhagen, Denmark.

The geometric and dynamic properties of all three deck models are discussed in Chapter 3 and are summarised here in Table A - 1. All the results shown here are non-dimensionalised using deck depth d_4 and first bending frequency f_n .

Table A - 1. Summary deck sections tested in three wind tunnel testing campaigns

Property		Northern Spire Bridge		Clyde Bridge		Narrow Water Bridge	
Deck shape							
Deck type (PD 6688-1-4)		 Bridge type 2		 Bridge type 1A		 Bridge type 1	
Scale		1:1	1:50	1:1	1:30	1:1	1:35
Deck width (m)	b	24.4	0.488	12.3	0.41	16.8	0.480
Girder/Box distance (m)	b^*	21.6	0.432	12.3	0.41	9.6	0.274
Deck depth (m)	d_4	2.5	0.050	2.4	0.08	1.68	0.048
Parapet height (m)	h	1.25	0.025	1.25	0.05	1.4	0.040
Mass (kg/m)	m	29,325	11.73	3,933	4.37	9,395	7.67
Bending frequency (Hz)	f_n	0.41	4.71	1.51	7.08	0.75	3.61
Damping (log-dec)	δ_s	2.8%	2.8%	3.0%	3.0%	3.1%	3.1%
Scruton number	Sc	209	209	33	33	168	168

Northern Spire Bridge

Three sets of results for the Northern Spire Bridge deck model are shown in Figure A - 1. Each of the tests is done with different turbulence intensities upwind from the deck: smooth flow ($I_u = 0.8\%$), low turbulent ($I_u = 4\%$) and highly turbulent flow ($I_u = 9\%$).

Looking at the smooth flow response, a clear lock-in region is observed between the reduced velocities of approximately 11 and 20. The spectrogram shows the Strouhal Law for this deck section along

¹ The VIV results presented in Appendix A are for a section with the wind fairings thus are different from the ones shown in Section 3.2.

with the lock-in of the shedding frequency to the structural frequency in the region where the limit cycle oscillations occur. By increasing the turbulence intensity, the vortex shedding strength has decreased. This is reflected in the displacement amplitude (almost halved when compared to the smooth flow) and on the spectrogram, where a clear Strouhal Law line is not visible anymore. When the turbulence is further increased the vibrations related to the vortex shedding are not observed for any of the tested velocities. However, the buffeting response, characterised by the jagged displacement amplitude curve for the higher flow velocities, is clearly visible for the tests under turbulent flows.

It is worth noting that these tests were done on a model comprising the parapet with a solidity ratio of 50%. Similar tests were undertaken on a model with the solid parapet (100% solidity ratio) and the findings were the same, i.e., the VIV displacements were suppressed with an increased turbulence intensity in the oncoming flow.

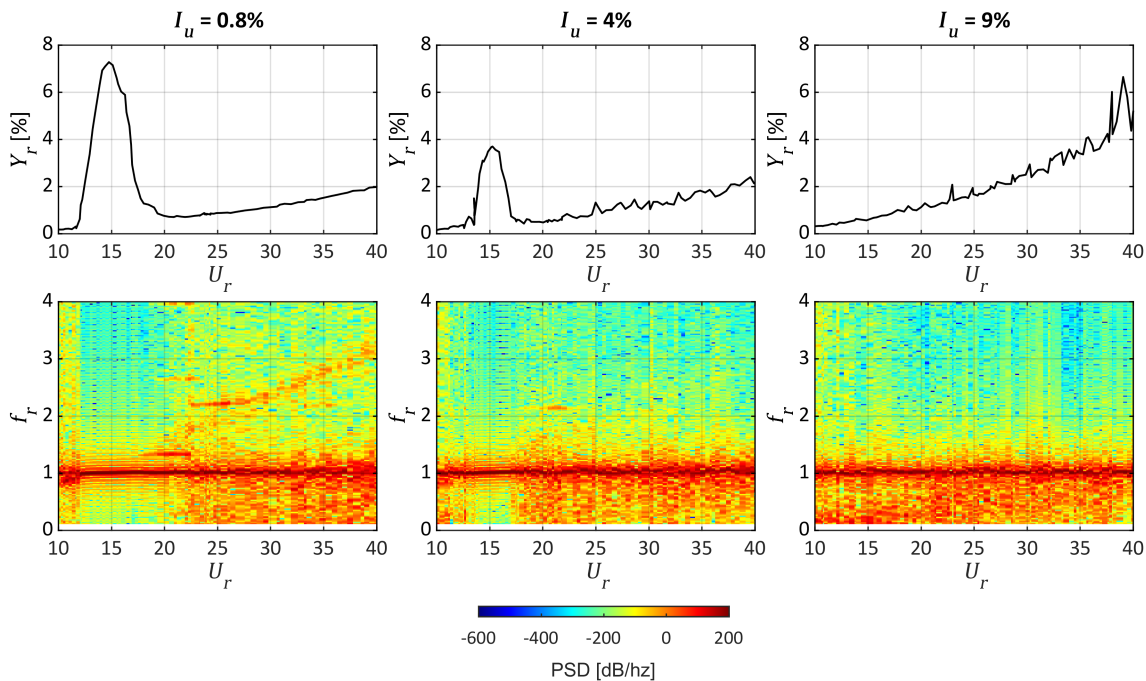


Figure A - 1. Northern Spire Bridge deck model response for three levels of turbulence intensity; VIV response in the top and spectrogram of the deck displacement in the bottom (adapted from Flamand & Knapp, 2018)

Clyde Bridge

Two sets of results for the Clyde Bridge deck model are shown in Figure A - 2. The tests are done in low turbulent flow ($I_u = 3.4\%$) and highly turbulent flow ($I_u = 9\%$). The results presented here are for the deck section with fascia plates installed on upwind and downwind deck edges hence are different from the results presented in Section 3.2.

As seen here, the introduction of the turbulence intensity increased the oscillation amplitude between the two cases by almost a factor of two. In neither of the cases, the Strouhal law curve can be observed. Similar findings were reported by Matsumoto et al., (1993) in their tests undertaken on a hexagonal box girder. As discussed at the end of Section 2.2.3, Karman vortex shedding is disrupted by the increased turbulence in the flow. The KV shedding attenuates motion-induced vortices, hence by disrupting the KV shedding, the effects of the MIV are increased. The MIV onset reduced velocity ($U_{r,MIV} = \frac{B}{0.6D}$) and is different

from the KV shedding reduced velocity ($U_{r,vs} = \frac{1}{St}$), where B and D are section width and depth, respectively and St is the Strouhal number. A difference in the oscillation onset velocity is noticeable for the two turbulence levels. Furthermore, when applied to the Clyde Bridge deck section, the equation estimating $U_{r,MIV}$ yielded a value of 8.55 which is in good agreement with the results for highly turbulent flow.

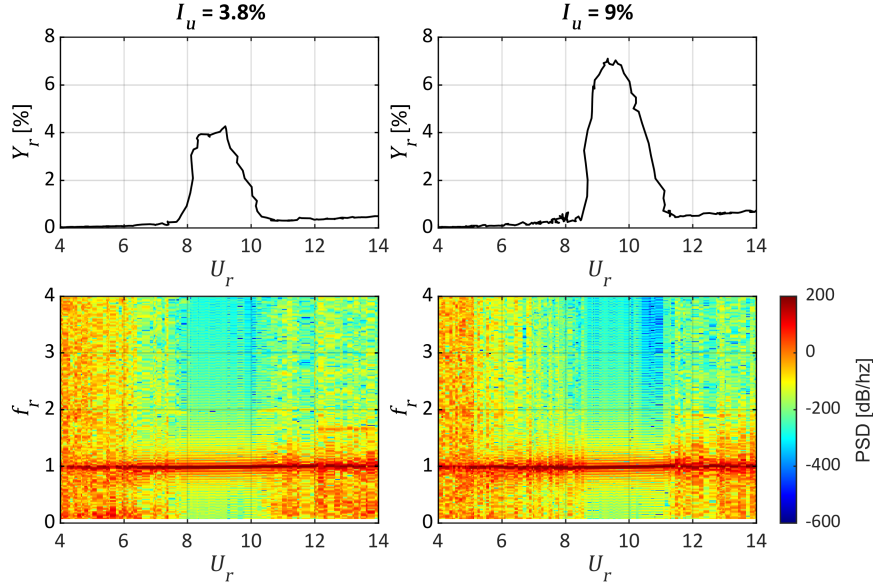


Figure A - 2. Clyde Bridge deck model response for two levels of turbulence intensity; VIV response in the top and spectrogram of the deck displacement in the bottom (adapted from Carusone & Flamand, 2022)

Narrow Water Bridge

Similar to the Clyde Bridge, two sets of results for the Narrow Water Bridge deck model are shown in Figure A - 3. The tests are done in smooth flow ($I_u = 1\%$) and highly turbulent flow ($I_u = 15\%$). As in the case of the Northern Spire Bridge, high turbulence intensity completely mitigates the oscillations.

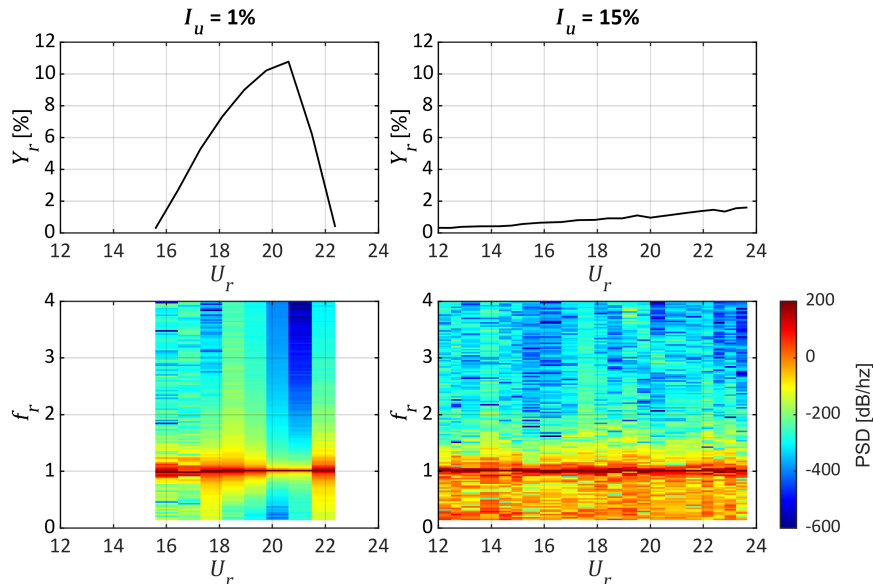


Figure A - 3. Narrow Water Bridge deck model response for two levels of turbulence intensity with the wind shielding; VIV response in the top and spectrogram of the deck displacement in the bottom (adapted from Svend Ole Hansen ApS, 2023)

Appendix B

UDF Functions

This appendix describes the UDF functions developed for the prescribed motion and FSI models used throughout the research. Firstly, a notation of several commands used throughout the script which are needed for the parallelisation of the UDF functions is given. More detailed information can be found in Fluent's UDF Manual (Ansys, 2021a). If correctly used, the scripts that include these commands should work both on parallel and serial simulations. When simulations are run in parallel, a distinction between the *host node* and *compute nodes* should be made. The host node is used to pass the commands to compute nodes that contain the mesh data and run the simulation.

```
#if !RP_HOST
/* commands written here are executed on either serial or compute nodes */
#endif

#if !RP_NODE
/* commands written here are executed on either serial or host nodes */
#endif

#if !RP_NODE
/* commands written here are executed only on compute nodes */
#endif

host_to_node_type_num /* to transfer the data from the host to the compute nodes */
node_to_host_type_num /* to transfer the data from the compute nodes to the host*/
```

At the start of the script, a number of variables are initiated and defined. The first three lines in the code are loading the so-called *header files* which define some of the basic operations such as input, output, and others. The system's dynamic properties are defined in lines 4,5 and 6. Variable `t1_cutoff` defines the time (in seconds of the analysis time) when the prescribed motion regime starts, while variable `t_cutoff` defines the duration of the prescribed motion model regime in terms of the number of oscillation periods. It is noted that the total analysis time is defined within Fluent.

```
1  #include "udf.h"      /* main UDF header file, defines all UDF functions */
2  #include "mem.h"      /* defines memory operations */
3  #include "stdio.h"    /* defines standard input output operations */
4  #define m 10.0042     /* mass [kg/m] */
5  #define k 9873.717    /* stiffness [N/m/m] */
6  #define psi 0.4775    /* damping ratio [%] */
7  #define amp 2.1       /* amplitude for initial kick, if no kick is needed set to 0.0 [mm]*/
8  #define zone_ID_num 1 /* number of zones collecting forces (deck, barriers)*/
9  #define zone_ID_deck 34 /* zone number of the deck*/
10 #define c ((psi/100)*2*sqrt(m*k)) /* damping */
11 #define freq (sqrt(k/m)/2/M_PI) /* natural frequency */
12 #define t_cutoff (1/freq*10) /* time when the kick shuts off [s] */
13 #define t1_cutoff 5 /* first time period (5s) for which FSI is active*/
14 real force_y, force_vy;
15 static real vy=0; /* centroid velocity */
16 static real vy_prev=0; /* centroid velocity - previous step*/
17 static real vy_cg=0; /* centroid velocity - applied to model*/
18 static real y=0; /* centroid location */
19 static real X.CG[3];
20 static real t_time;
21 static int zone_ID_all[zone_ID_num] = {zone_ID_deck}; /* all zones that are collecting
22 FILE *fout, *flastRes;
```

Between lines 23 and 28 shown below, two simple functions are defined which are used in the Runge-Kutta 4th Order Method. Function `func1` defines $\frac{dy}{dt}$ term i.e., the velocity, and `func2` defines $\frac{d^2y}{dt^2}$ i.e., the acceleration. In general, the definition of these two functions could be avoided and the resulting terms could be directly written within the R-K method, however by defining them the code is more compact and easier to follow.

```

23 real func1(real vy) { /* dydt */
24     return(vy);
25 }
26 real func2(real y, real vy, real f) { /* ddydt */
27     return(f - k*y - c*vy)/m;
28 }

```

The following parts of the script are not essential for its unhindered running. However, if the simulation is to be continued after being interrupted and Fluent closed, the following parts are needed. `DEFINE_EXECUTE_ON_LOADING` is a UDF function that is automatically executed upon opening the simulation files in Fluent. Here, two text files are used for the collection of the main data such as centroid velocity, simulation time, centroid location and others. `UDF_results.txt` contains the data for each simulated time step while `LAST_RES.txt` includes only the data from the last one and is used when the simulation needs resuming. If the simulation is being initiated, lines 33 to 43 create the two text files while the code between lines 44 and 57 reads in the data should the simulation is resumed. The remaining lines display some of the input information in Fluent's command window, such as the dynamic properties of the system.

```

29 DEFINE_EXECUTE_ON_LOADING(on_load,libname)
30 {
31     #if !RP_NODE
32     Message("Initial start \n");
33     if (!(fout=fopen("UDF_results.txt","r")))
34     {
35         Message("UDF_results.txt not available.\n");
36         fout = fopen("UDF_results.txt", "w");
37         fclose(fout);
38         Message("UDF_results.txt created.\n");
39     }
40     if (!(flastRes=fopen("LAST_RES.txt","r")))
41     {
42         Message("LAST_RES.txt not available.\n");
43     }
44     else
45     {
46         Message("LAST_RES.txt available.\n");
47         flastRes=fopen("LAST_RES.txt","r");
48         fscanf(flastRes, "%lf %lf %lf %lf %lf %*lf %lf",
49 &t_time,&vy,&y,&vy_cg,&X.CG[1],&force_y);
50         fclose(flastRes);
51         vy_prev = vy;
52         Message("RESUMED VALUE - time: %.15e\n",t_time);
53         Message("RESUMED VALUE - vy: %.15e\n",vy);
54         Message("RESUMED VALUE - y: %.15e\n",y);
55         Message("RESUMED VALUE - vy_cg: %.15e\n",vy_cg);
56         Message("RESUMED VALUE - X.CG[1]: %.15e\n",X.CG[1]);
57         Message("RESUMED VALUE - force_y: %.15e\n",force_y);
58     }
59     Message("stiffness: %g N/m/m\n",k);
60     Message("mass: %g kg/m\n",m);
61     Message("damping coeff: %g\n",c);
62     Message("frequency: %g Hz\n",freq);
63     Message("initial kick: %g mm\n",amp);
64     Message("kick cutoff: %g s\n",t_cutoff);
65     #endif
66 }

```

The next part of the script contains the core UDF function `DEFINE_EXECUTE_AT_END` which is executed at the end of each time step. Firstly, the script calculates the lift force by integrating pressure in the y-direction over all surfaces (deck and barriers). This is contained between lines 73 and 107 and is done on every computation node. It is noted that in lines 104 and 105, the lift force is summed across all computational nodes and then in line 108, along with the centroid location, transferred to the host node.

After the lift force is calculated, the velocity of the dynamic part of the mesh at the time step end is calculated. This is done on the host node by either solving the equation of motion (for the FSI regime) or calculating the centroid velocity from the sinusoidal equation (for the PM regime). The equation of motion is solved using Runge-Kutta 4th Order Method and is contained between lines 125 and 134, while the sinusoidal equation for velocity is written in line 142. At the end of this function, a series of lines is written for outputting the results to two text files (between lines 147 and 152) and displaying some of it in Fluent's output window (between lines 153 and 157).

```

66 DEFINE_EXECUTE_AT_END(exec_end)
67 {
68     int n;
69     for (n=0; n<zone_ID_num; n++)
70     {
71         host_to_node_int_1(zone_ID_all[n]);
72     }
73     #if !RP_HOST
74         int zoneID;
75         Domain *domain;
76         Thread *thread;
77         Dynamic_Thread *dt;
78         face_t f;
79         domain=Get_Domain(1);
80         force_y = 0.0;
81         force_vy = 0.0;
82         for (n=0; n<zone_ID_num; n++)
83         {
84             zoneID = zone_ID_all[n];
85             real NV_VEC(A);
86             thread = Lookup_Thread(domain,zoneID);
87             /*Force calculation*/
88             begin_f_loop(f,thread)
89             {
90                 if (PRINCIPAL_FACE_P(f,thread))
91                 {
92                     force_vy = F_STORAGE_R_N3V(f,thread,SV_WALL_SHEAR)[1]*-1;
93                     F_AREA(A,f,thread);
94                     /*Force calculation with a depth of 1m*/
95                     force_y += F_P(f,thread)*A[1] + force_vy;
96                 }
97             }
98             end_f_loop(f,thread)
99         }
100         thread = Lookup_Thread(domain,zone_ID_deck);
101         dt=THREAD_DT(thread);
102         X_CG[1] = DT_CG(dt)[1];
103         #if RP_NODE
104             force_y = PRF_GRSUM1(force_y);
105             force_vy = PRF_GRSUM1(force_vy);
106         #endif
107     #endif
108     node_to_host_real_2(X_CG[1],force_y); /* move centroid location and lift force to host*/
109     #if !RP_NODE
110         real t_time=CURRENT_TIME;
111         real dtime=CURRENT_TIMESTEP;
112         if (t_time<t1_cutoff || t_time>=(t1_cutoff+t_cutoff)) /* First FSI is active, then PM
for the kick, then again FSI */
113         {
114             if (t_time<t1_cutoff)
115             {
116                 Message("*** 1st FSI active ***\n");

```



```

117     }
118     else
119     {
120         Message("*** 2nd FSI active ***\n");
121     }
122     /* RK4 - start */
123     real k1_y1,k2_y1,k3_y1,k4_y1;
124     real k1_y2,k2_y2,k3_y2,k4_y2;
125     k1_y1 = dttime * func1(vy);
126     k1_y2 = dttime * func2(y,vy,force_y);
127     k2_y1 = dttime * func1(vy + k1_y1*0.5);
128     k2_y2 = dttime * func2(y + k1_y1*0.5, vy + k1_y2*0.5, force_y);
129     k3_y1 = dttime * func1(vy + k2_y1*0.5);
130     k3_y2 = dttime * func2(y + k2_y1*0.5, vy + k2_y2*0.5, force_y);
131     k4_y1 = dttime * func1(vy + k3_y1);
132     k4_y2 = dttime * func2(y + k3_y1, vy + k3_y2, force_y);
133     y = y + (k1_y1 + 2*k2_y1 + 2*k3_y1 + k4_y1)/6; /* displacement */
134     vy = vy + (k1_y2 + 2*k2_y2 + 2*k3_y2 + k4_y2)/6; /* velocity */
135     vy_cg = (vy + vy_prev)/2;
136     vy_prev = vy;
137     k1_y1=k1_y2=k2_y1=k2_y2=k3_y1=k3_y2=k4_y1=k4_y2=0; /* reset coefficients */
138     /* RK4 - end */
139 }
140 else
141 {
142     Message("*** PM active ***\n");
143     vy_cg = amp*freq*M_PI*2*cos(t_time*freq*M_PI*2)/1000; /* velocity */
144     vy = vy_cg;
145     vy_prev = vy_cg;
146     y = X.CG[1]; /* displacement */
147 }
148 fout = fopen("UDF_results.txt", "a");
149 fprintf(fout, "%.15e %.15e %.15e %.15e %.15e %.15e \n",
t_time,vy,y,vy_cg,X.CG[1],X.CG[1]+vy_cg*dttime,force_y); /* save to output file */
150 fclose(fout);
151 flastRes=fopen("LAST_RES.txt","w");
152 fprintf(flastRes, "%.15e %.15e %.15e %.15e %.15e %.15e \n",
t_time,vy,y,vy_cg,X.CG[1],X.CG[1]+vy_cg*dttime,force_y); /* overwrite the last time step */
153 fclose(flastRes);
154 Message("time: %g\n",t_time);
155 Message("y: %g\n",y);
156 Message("X.CG[1]: %g\n",X.CG[1]);
157 Message("vy: %g\n",vy);
158 Message("force y: %g\n",force_y);
159 #endif
160 }

```

The last UDF function defined is `DEFINE CG_MOTION`. It controls the motion of the dynamic parts of the mesh. This is done by applying the velocity at the start of each time step to sub-grids labelled *Dynamic-Rigid* in Table 4.3 and Table 4.6.

```

160 DEFINE	CG_MOTION(motion,dt,vel,omega,time,dttime)
161 {
162     host_to_node_real_1(vy_cg);
163     #if !RP_HOST
164         NV_S(vel, =, 0.0);
165         NV_S(omega, =, 0.0);
166         vel[1] = vy_cg;
167     #endif
168 }

```

The research was partially funded by the Irish Research Council under grant number EBPPG/2020/96. I would like to acknowledge the Irish Centre for High-End Computing for the provision of computational facilities and support. I would like to thank everyone at Centre Scientifique et Technique du Bâtiment and Svend Ole Hansen ApS for sharing the data from the wind tunnel analyses and promptly answering any questions related to it.
