Modeling and simulation of bridge-section buffeting response in turbulent flow

Tore A. Helgedagsrud
Department of Structural Engineering, Norwegian University of Science and Technology (NTNU),
Richard Birkelands v 1a, Trondheim NO-7491, Norway
tore.a.helgedagsrud@ntnu.no

Yuri Bazilevs*
School of Engineering, Brown University,
184 Hope Street, Providence, RI 02912, USA
yuri_bazilevs@brown.edu

Kjell M. Mathisen
Department of Structural Engineering, Norwegian University of Science and Technology (NTNU),
Richard Birkelands v 1a, Trondheim NO-7491, Norway
kjell.mathisen@ntnu.no

Jinhui Yan
Department of Civil and Environmental Engineering, University of Illinois at Urbana-Champaign,
Newmark 2106, 205 N. Mathews, Urbana, IL 61801, USA
yjh@illinois.edu

Ole A. Øiseth
Department of Structural Engineering, Norwegian University of Science and Technology (NTNU),
Richard Birkelands v 1a, Trondheim NO-7491, Norway
ole.oiseth@ntnu.no

Received (Day Month Year)
Revised (Day Month Year)
Communicated by (xxxxxxxxxx)

*Corresponding author.
Buffeting analysis plays an important role in the wind-resistant design of long-span bridges. While computational methods have been widely used in the study of self-excited forces on bridge sections, there is very little work on applying advanced simulation to buffeting analysis. In an effort to address this shortcoming, we developed a framework for the buffeting simulation of bridge sections subjected to turbulent flows. We carry out simulations of a rectangular bridge section with aspect ratio 10 and compute its aerodynamic admittance functions. The simulations show good agreement with airfoil theory and experimental observations. It was found that inflow turbulence plays an important role in obtaining accurate wind loads on the bridge sections. The proposed methodology is envisioned to have practical impact in wind engineering of structures in the future.

Keywords: Turbulence; Buffeting; Bridge aerodynamics; RBVMS; Multi-domain modeling; Weak boundary conditions; Isogeometric Analysis.

AMS Subject Classification: 22E46, 53C35, 57S20

1. Introduction

Buffeting analysis plays an important role in the wind-resistant design of long-span bridges and other line-like structures, particularly in serviceability and fatigue limit states.

As for most wind engineering aspects of bridges, buffeting theory originates from the developments in the aerospace engineering and sciences. The earliest formulation was reported in,[1] in which the evolution of the aerodynamic lift on an airfoil was defined for a uniform vertical gust. In[2] the work was extended to define a function that relates a sinusoidal variation of the vertical wind velocity to a fluctuating lifting force, i.e., the Sears function. A spectral form of the Sears function represents the earliest version of the aerodynamic admittance function, and is still commonly regarded as the reference solution in buffeting analysis.

Unlike airfoils, bridge decks and bluff bodies with detached flows do not have an analytical basis for buffeting, and, therefore, rely on empirical relations. Such a framework was first presented in,[3] where a frequency-domain analysis of buffeting and the concept of aerodynamic admittance were applied to bridges. This methodology was further developed in,[4,5,6,7] and, in addition to time-domain methods,[8] presents a standard approach in bridge design today.

Aerodynamic admittances were traditionally obtained from wind tunnel experiments, and several test strategies report successful results, see, e.g.,[9,10,11,12,13,14,15] However, as the majority of these methods measure forces over a wider segment than the flow integral length scale, span-wise coherence of the turbulence and buffeting forces must be considered. As discussed in,[16,15] this introduces some uncertainty in the estimation of admittance functions. Whereas methods based on pressure strips do not have these problems, they do instead suffer from inaccuracies of the pressure integration.[17]

In contrast to experimental approaches, high-fidelity Computational Fluid Dynamics (CFD) modeling does not suffer from these shortcomings, which makes it a good candidate for buffeting analysis. CFD methods have been widely used in the study of self-excited forces on bridge sections, see e.g.,[18,19,20,21,22,23,24,25] and references therein. For buffeting analysis, however, the only application of CFD, to the best of the authors’ knowledge, may be found in.[26,27] Although synthetic inlet turbulence was considered in the simulations, the simulations themselves were carried out in 2D, which only gives a limited representation
of the dynamics of turbulent flows.

In this work we present a computational framework, and carry out numerical simulations of buffeting. For the flow simulation, we use the semi-discrete version of the Residual-Based Variational Multiscale (RBVMS) formulation of the Navier–Stokes equations of incompressible flows, proposed and further developed. The space–time version of RBVMS, named ST-VMS, was proposed and further developed more recently. Both VMS formulations proved their accuracy and efficiency through numerous high-Reynolds-number flow computations for applications such as parachutes, turbomachinery, thermo-fluids, flows with topology change, wind turbines, hydro turbines, bio-inspired propulsion, cardiovascular hemodynamics, and bridge aerodynamics.

To simulate the incident turbulence, a Multi-Domain Modeling (MDM) approach is employed, where the turbulence is generated in a separate domain, and then used as the inflow boundary condition (BC) for the bridge-section domain. The incident turbulence is generated by a pressure-driven turbulent channel flow computed using Isogeometric Analysis (IGA) based on Non-Uniform Rational B-Splines (NURBS). The flow bulk velocity and friction-velocity-based Reynolds number are used as two parameters that define the turbulent inflow and, in part, govern the turbulence intensity.

To demonstrate the accuracy and efficiency of the framework, aerodynamic admittance functions and pressure distributions on a rectangular section with aspect ratio 1:10 subjected to uniform and turbulent inflow conditions are computed. Although most long-span bridges have a more streamlined shape, rectangular sections are often considered as a reference, and have been extensively studied both experimentally and numerically. In addition, as experience has shown, accurately capturing the flow separation and reattachment for this geometrically simple bluff-body shape is a challenging task. The bridge section is simulated using standard low-order finite elements (FEM), and the framework of weakly-enforced essential BCs is employed to couple the non-matching IGA and FEM discretizations.

The paper is outlined as follows. Sec. 2 gives an overview of the buffeting theory for bridge sections. In Sec. 3 we present the computational framework. Numerical results are shown in Sec. 4 for the turbulent channel flow and in Sec. 5 for the bridge section. Conclusions are drawn in Sec. 6.

2. Buffeting and aerodynamic admittance

Wind loads acting on line-like bridge girders are non-linear functions of structural motions and incident turbulence. These are, however, commonly linearized and expressed in terms of time-averaged static, self-excited and buffeting components. In this work, we focus on the latter. For a comprehensive description of the dynamic wind loads on bridges, the readers is referred to.

As shown in Fig. 1, we decompose the wind speed into a mean component $U$, acting
in the drag direction of the bridge, and zero-mean turbulent fluctuations \( u(t) = [u, w]^T \) in the drag and lift direction, respectively. Under the assumption of isotropic turbulence, these components are regarded as independent.

The total aerodynamic forces due to a turbulent wind field, \( \mathbf{q}_T(t) \), are given as

\[
\mathbf{q}_T(t) = \mathbf{q} + \mathbf{q}_b(t),
\]

where \( \mathbf{q} = [\bar{D}, \bar{L}, \bar{M}]^T \) contains the mean drag, lift and pitching moment, respectively, and \( \mathbf{q}_b(t) = [D_b, L_b, M_b]^T \) is the buffeting component.

The drag, lift and pitching-moment coefficients are defined as

\[
C_D = \frac{\bar{D}}{1/2 \rho U^2 H}, \quad C_L = \frac{\bar{L}}{1/2 \rho U^2 B}, \quad C_M = \frac{\bar{M}}{1/2 \rho U^2 B^2},
\]

where \( 1/2 \rho U^2 \) is the stagnation pressure and \( H \) and \( B \) are the height and the width of the bridge girder, respectively. For the buffeting part we denote \( \mathbf{G}_v(K) = [G_u, G_w]^T \) the temporal Fourier transform of \( u(t) \) as a function of the reduced frequency \( K = \omega B/U \), and \( \mathbf{G}_b(K) = [G_D, G_L, G_M]^T \) the Fourier transform of \( \mathbf{q}_b(t) \). The buffeting wind action can then be expressed in the following form:

\[
\mathbf{G}_b(K) = \frac{1}{2} \rho U B \chi(K) \mathbf{G}_v(K),
\]

where

\[
\chi = \begin{bmatrix}
2(H/B)\bar{C}_D \chi_{Du} & ((H/B)\bar{C}'_D - \bar{C}_L) \chi_{Dw} \\
2\bar{C}_L \chi_{Lu} & (C''_L + (H/B)\bar{C}_D) \chi_{Lw} \\
2\bar{C}_M \chi_{Mu} & BC'_M \chi_{Mw}
\end{bmatrix}
\]

is the matrix of aerodynamic admittances. The bars and primes on the aerodynamic load coefficients refer to their mean value and inclination with respect to the angle-of-attack, respectively. The coefficients \( \chi_{ij}, i \in \{D, L, M\}, j \in \{u, w\} \), are the aerodynamic admittance functions, varying with the frequency of the turbulent fluctuations. These can be interpreted as transfer functions between the fluctuating wind velocities and aerodynamic forces.
The cross-spectral density matrix from Eq. (2.3) reads

\[ S_b(K) = \left( \frac{1}{2} \rho UB \right)^2 \chi(K) S_v(K) \chi^T(K), \]  

(2.5)

where \( S_b(K) \) and \( S_v(K) \) are the cross-spectral density matrices of the forces and velocities, respectively, and superscript * denotes the complex conjugate of its attribute. Equation (2.5) serves as a starting point for most methods of identifying the aerodynamic admittance functions, such as auto-spectral methods\(^{11,17} \) and cross-spectral methods.\(^9\)

In the following, we assume that the velocity cross spectrum can be neglected, i.e., \( S_{uw}(K) = 0 \). Further, we recognize that, for the symmetric bridge section considered in this work, the components \( \bar{C}_L, \bar{C}_M \) and \( C_D' \) are exactly zero. The diagonal components of Eq. (2.5) then reduce to

\[ S_{DD}(K) = \left( \frac{1}{2} \rho UB \right)^2 \left( \frac{2}{B} H_B \bar{C}_D \right)^2 |\chi_{Du}|^2 S_{uu}(K), \]  

(2.6a)

\[ S_{LL}(K) = \left( \frac{1}{2} \rho UB \right)^2 \left( C_L' + \frac{H}{B} \bar{C}_D \right)^2 |\chi_{Lw}|^2 S_{ww}(K), \]  

(2.6b)

\[ S_{MM}(K) = \left( \frac{1}{2} \rho UB \right)^2 (C_M')^2 |\chi_{Mw}|^2 S_{ww}(K). \]  

(2.6c)

We note that Eq. (2.6) coincides with the auto-spectral method in buffeting analysis, in which the admittance components \( \chi_{Du}, \chi_{Lw} \) and \( \chi_{Mw} \) are treated independently. This is also the governing equation for identifying the aerodynamic admittance functions.

For an airfoil passing through a sinusoidal vertical-gust pattern, the analytical expression for the lift admittance is given by the Sears function\(^2\)

\[ \chi_{Lw}(K) := \chi(K) = \frac{2}{\pi KH_+(K)}, \]  

(2.7)

where \( H_+ \) is the Bessel function of the third kind. Being derived from a smooth and fully correlated flow, it is a good approximation for streamlined decks and often regarded as the reference solution, however, for bluff bodies it may underestimate the admittance for low frequencies.\(^{10} \)

Lastly, it should be mentioned that simplified models of buffeting use admittances of unity. This corresponds to quasi-steady theory, and represents the upper bound for the buffeting forces and are often used in design codes (see, e.g.,\(^{100} \)). However, as demonstrated in,\(^9\) this simplification may render an excessive buffeting response.

Remark 2.1. To simplify the above relations we have intentionally omitted the span-wise coordinate. However, in a modal superposition approach, which is a widely used method in structural dynamics, the extension is straightforward. See, e.g.,\(^4\) for a practical implementation.
3. Modeling and discretization approaches

3.1. Navier–Stokes equations of incompressible flows

We consider the weak formulation of the Navier–Stokes equations for incompressible flows on the domain $\Omega$. Let $S$ and $V$ denote the trial and test function spaces, respectively. The variational formulation is then stated as follows: Find a velocity-pressure pair $\{u, p\} \in S$, such that for all test functions $\{w, q\} \in V$

$$B(\{u, p\}, \{w, q\}) - F(\{w, q\}) = 0,$$

(3.1)

where

$$B(\{w, q\}, \{u, p\}) = \int_{\Omega} w \cdot \left( \rho \frac{\partial u}{\partial t} + u \cdot \nabla u \right) d\Omega$$

$$+ \int_{\Omega} \varepsilon(w) : \sigma(u, p) d\Omega + \int_{\Omega} q \nabla \cdot u d\Omega,$$

(3.2)

and

$$F(\{w, q\}) = \int_{\Omega} w \cdot \rho f d\Omega + \int_{\Gamma_h} w \cdot h d\Gamma.$$

(3.3)

Here, $\rho$ is the fluid density, $f$ is the body force per unit mass and $h$ is the traction acting on the $\Gamma_h$ part of the boundary $\Gamma$. The Cauchy stress $\sigma$ is given

$$\sigma(u, p) = -pI + 2\mu \varepsilon(u),$$

(3.4)

where $I$, $\mu$ and $\varepsilon(u)$ are the identity tensor, dynamic viscosity and the strain rate tensor, respectively, where the latter is given by

$$\varepsilon(u) = \frac{1}{2} \left( \nabla u + \nabla u^T \right).$$

(3.5)

3.2. RBVMS and weak BCs

At the discrete level, the fluid domain is partitioned into $n_{el}$ subdomains $\Omega_e$ such that $\Omega = \bigcup_{e=1}^{n_{el}} \Omega_e$. Accordingly, the fluid boundary is partitioned into $n_{eb}$ boundary elements $\Gamma^h_b$. We now define finite-dimensional trial- and test functions as $S^h \subset S$ and $V^h \subset V$, respectively. The variational formulation of (3.1) is then stated as: Find $\{u^h, p^h\} \in S^h$, such that for all test functions $\{w^h, q^h\} \in V^h$

$$B(\{w^h, q^h\}, \{u^h, p^h\}) + B_{VMS}(\{w^h, q^h\}, \{u^h, p^h\})$$

$$+ B_{WBC}(\{w^h, q^h\}, \{u^h, p^h\}) - F(\{w^h, q^h\}) = 0.$$

(3.6)
In Eq. (3.6), \( B \) and \( F \) are given by Eqs. (3.2) and (3.3), respectively. The stabilization terms that arise from the RBVMS formulation are given by \( B_{VMS} \) as
\[
B_{VMS} (\{w^h, q^h\}, \{u^h, p^h\}) = \sum_{e=1}^{n_{el}} \int_{\Omega^e} \tau_{SUPS} \left( u^h \cdot \nabla w^h + \frac{\nabla q^h}{\rho} \right) \cdot r_M (u^h, p^h) \, d\Omega \\
+ \sum_{e=1}^{n_{el}} \int_{\Omega^e} \rho_{LSIC} \nabla \cdot w^h r_C (u^h) \, d\Omega \\
- \sum_{e=1}^{n_{el}} \int_{\Omega^e} \tau_{SUPS} w^h \cdot (r_M (u^h, p^h) \cdot \nabla u^h) \, d\Omega \\
- \sum_{e=1}^{n_{el}} \int_{\Omega^e} \nabla w^h : (\tau_{SUPS} r_M (u^h, p^h) \otimes (\tau_{SUPS} r_M (u^h, p^h))) \, d\Omega,
\]
(3.7)
in which \( \tau_{SUPS} \) and \( \rho_{LSIC} \) are stabilization parameters that ensure stability and optimal convergence (see e.g., \(^{102,103,104,105}\)). In this work we use the definitions given in.\(^{106}\) \( r_M \) and \( r_C \) are the residuals of strong form linear-momentum and continuity equations, respectively, defined as
\[
r_M (u^h, p^h) = \rho (\frac{\partial u^h}{\partial t} + u^h \cdot \nabla u^h) + \nabla p^h - \nabla \cdot \sigma (u^h, p^h)
\]
(3.8)
and
\[
r_C (u^h) = \nabla \cdot u^h.
\]
(3.9)
The last form, \( B_{WBC} \), augments the terms associated with the weakly-enforced boundary conditions.\(^{107,108,65}\) Given a prescribed velocity \( g \) on the boundary \( \Gamma_g \), whose outward normal vector is denoted \( n \), this form reads
\[
B_{WBC} (\{w^h, q^h\}, \{u^h, p^h\}) = - \sum_{b=1}^{n_{el}} \int_{\Gamma_b \cap \Gamma_g} w^h \cdot \sigma (u^h, p^h) \, n \, d\Gamma \\
- \sum_{b=1}^{n_{el}} \int_{\Gamma_b \cap \Gamma_g^-} (2\mu \varepsilon (w^h) n + q^h n) \cdot (u^h - g) \, d\Gamma \\
- \sum_{b=1}^{n_{el}} \int_{\Gamma_b \cap \Gamma_g^-} w^h \cdot \rho (u^h \cdot n) (u^h - g) \, d\Gamma \\
+ \sum_{b=1}^{n_{el}} \int_{\Gamma_b \cap \Gamma_g} \tau_{TAN} (w^h - (w^h \cdot n) n) \cdot ((u^h - g) \cdot (u^h - g) \cdot n) \, n \, d\Gamma \\
+ \sum_{b=1}^{n_{el}} \int_{\Gamma_b \cap \Gamma_g} \tau_{NOR} (w^h \cdot n) ((u^h - g) \cdot n) \, d\Gamma.
\]
(3.10)
Here, \( \tau_{TAN} \) and \( \tau_{NOR} \) are boundary penalty parameters\(^{107}\) in the tangential and normal directions, respectively. \( \Gamma_g^- \) is defined as the inflow part of \( \Gamma_g \):
\[
\Gamma_g^- = \{ x \mid u^h \cdot n < 0, \forall x \in \Gamma_g \}.
\]
(3.11)
The generalized-α method\cite{109,110,106} is employed to integrate the RBVMS equations in time.

### 3.3. Turbulence-generating domain: Pressure-driven wall-bounded channel flow

For the channel computations we characterize the flow using the friction-velocity Reynolds number, Re$_\tau$, and the mean bulk velocity, $\overline{U}_b$. The friction-velocity Reynolds number Re$_\tau$ is given by:\cite{83}

$$Re_\tau = \frac{u^* D}{\nu},$$

where $D$ is the channel half-height, $\nu$ is the kinematic viscosity, and $u^*$ is the friction velocity. The latter is given by

$$u^* = \sqrt{\frac{\nu \partial u}{\partial n}},$$

The non-dimensional wall distance, $y^+$, and velocity, $u^+$, are now defined as

$$y^+ = \frac{yu^*}{\nu} \quad \text{(3.14)}$$

and

$$u^+ = \frac{U}{u^*} \quad \text{(3.15)}$$

where $y$ and $U$ are the dimensional wall distance and stream-wise velocity, respectively. From Eq. (3.14) it follows that $y^+|_{cl} = Re_\tau$, where subscript $cl$ refers to the channel center line (i.e., $y = D$).

The relation between $y^+$ and $u^+$ is given by the "law of the wall", for which we use Spalding’s parameterization\cite{111} given by

$$y^+ = g(u^+) = u^+ + e^{-\chi B} \left( e^{\chi u^+} + 1 - \chi u^+ - \frac{(\chi u^+)^2}{2!} - \frac{(\chi u^+)^3}{3!} \right),$$

where $\chi = 0.4$ and $B = 5.5$.

Combining Eqs. (3.14)–(3.16), we obtain the centerline velocity as

$$\overline{U}_{cl} = u^+ u^+|_{cl} = u^* g^{-1}(Re_\tau),$$

which may be related to the mean bulk velocity by Dean’s correlation\cite{112} as

$$\overline{U}_{cl} = 1.28 \overline{U}_b Re_b^{-0.0016},$$

where $Re_b = 2\overline{U}_b D/\nu$ is the bulk Reynolds number. The kinematic viscosity may be obtained from Eq. (3.12)

$$\nu = \frac{u^* D}{Re_\tau}.$$
In addition, in a statistically-stationary flow, the mean wall shear force is balanced by the applied stream-wise pressure gradient $f$ as

$$\int_{\Gamma_{ch}} \nu \frac{\partial p}{\partial n} d\Gamma = \int_{\Omega_{ch}} f d\Omega.$$  \hspace{1cm} (3.20)

A direct evaluation of the integrals in Eq. (3.20) yields

$$f = \frac{u^*}{D}.$$  \hspace{1cm} (3.21)

In practice, the relations above are incorporated in the channel-flow computations through the following initialization steps:

1. Start with a desired $U_b$ and $Re_\tau$;
2. Determine $u^+$ from Eq. (3.16) with $y^+ = Re_\tau$;
3. Solve for $u^*$, $U_{cl}$, and $\nu$ using the equation system formed by Eqs. (3.17)–(3.19);
4. Solve for the volumetric forcing $f$ from Eq. (3.21).

This procedure produces the dimensional values of the kinematic viscosity and forcing needed to carry out the channel-flow calculations at wind speeds and bridge-deck sizes corresponding to the full-scale or wind-tunnel-scale cases.

### 3.4. Domain coupling and data transfer

![Fig. 2: Mesh and processor partitioning of the channel outflow (left) and bridge-section inflow (right).](image)

Projection of the velocity field from the channel outflow surface to the bridge domain inflow surface involves non-matching grids, non-conforming processor partitioning, and different element topologies, as illustrated in Fig. 2. To accommodate these, we present a strategy based on weakly enforced BCs, which only requires the velocity-field information at the quadrature points of the inflow of the bridge-section grid.
Following the illustration in Fig. 3, we let $\mathbf{x}^B_j$ denote the Gauss point coordinates of the bridge domain inflow surface (superscript $B$) associated with the local processor ID $j$, $j \in [1, M_{\text{proc}}]$. Next, we let $\mathbf{u}^A_i(\mathbf{x}, t)$ denote the velocity field for the channel outflow surface at time $t$ and the local processor ID $i$, $i \in [1, N_{\text{proc}}]$. As an initialization step in the channel simulation we establish the connectivity between $i$ and $j$, and allocate global velocity vectors associated with $\mathbf{x}^B_j$; $\mathbf{u}^A_j$. We then evaluate $\mathbf{u}^A_i(\mathbf{x}^B_j, t_m)$, where $t_n$ is the desired time level, for all $i \cap j$, and export to global velocity vectors $\mathbf{u}^A_j(\mathbf{x}^B_j, t_n)$. Finally, we export these to a database read as part of the bridge-section simulation.

In the bridge-section domain, Eq. (3.10) governs the weak imposition of the velocity field $\mathbf{g}$ on $\Gamma_g$. As a result, at a desired time level $t_{m}$, we set $\mathbf{g}^B_j(\mathbf{x}^B_j, t_m) = \mathbf{u}^A_j(\mathbf{x}^B_j, t_m)$, determined by a linear-in-time interpolation from the neighboring time steps in the inflow velocity database as:

$$
\mathbf{g}^B_j(\mathbf{x}^B_j, t_m) = (1 - \xi)\mathbf{u}^A_j(\mathbf{x}^B_j, t_n) + \xi\mathbf{u}^A_j(\mathbf{x}^B_j, t_{n+1}),
$$

(3.22)

$$
t_m = (1 - \xi)t_n + \xi t_{n+1}, \quad t_n < t_m < t_{n+1}.
$$

(3.23)

In addition, because the flow is incompressible, we ensure that the flow rate at the inlet remains constant and consistent with the set bulk velocity $\overline{U}_b$, by explicitly enforcing the following condition on the inflow velocity at each time level:

$$
\int_{\Gamma_g} (\mathbf{g} \cdot \mathbf{n} - \overline{U}_b) \, d\Gamma = 0.
$$

(3.24)

Remark The data projection approach presented here is very similar to that described in. However, in the present work, the step where the velocity is projected onto the inlet nodal space is not needed since the quadrature-point values are used directly in the enforcement of weak BCs.

4. IGA simulations of turbulent channel flows

The channel height is set to 2 m, which is the vertical dimension of the wind tunnel described in the authors’ earlier work. At the walls, no-slip BCs are enforced weakly and
no-penetration BCs are enforced strongly. The channel stream-wise and span-wise dimensions are 6 m and 2 m, respectively. Periodic BCs are applied in these directions. The bulk velocity is set to $U_b = 2.0$ m/s, and $Re_\tau$ of 395 and 2000 are considered in this work.

For the discretization we use $C^1$-continuous quadratic NURBS and a uniform mesh of $192 \times 64 \times 64$ elements. It should noted that the smallest length scales resolved for the given mesh then becomes approximately 30 mm, or $B/16$. The domain is partitioned into $16 \times 4 \times 4$ processors and the time step is set to $\Delta t = 0.01$ s.

The simulations are carried out until a statistically stationary flow is achieved. The flow patterns in this state are depicted in Fig. 4, showing instantaneous vertical velocity contours for a channel cross section for both Reynolds numbers. For $Re_\tau = 395$ and $Re_\tau = 2000$, the turbulence intensities are approximately 4.4 % and 2.9 %, respectively.

The mean stream-wise velocity and root mean square (RMS) of the velocity fluctuations are shown in Figs. 5 and 6, respectively. For $Re_\tau = 395$ both the mean and RMS profiles match the DNS results very well. For $Re_\tau = 2000$, while the mean velocity is very accurate, we see more deviation in RMS profile. However, considering that the first knot is located at $y^+ = 63$, the results are very good, and underscore the effectiveness and accuracy of RBVMS and weak BCs discretized with NURBS for this problem class.

Figure 7 shows the vertical vs. longitudinal velocity fluctuations at the channel centerline. The uncorrelated fluctuations reveal an isotropic turbulence field in the channel center, which is the flow that will be directly impacting the bridge deck. Velocity spectra for the stream-wise ($u$) and span-wise ($w$) velocity components are shown in Figs. 8 and 9, respectively. The velocities were sampled at a fixed spatial point for a 50 s period. The same plots also show the least-squares fitted curves of the classical von Kármán one-point spectra, which for the $u$ and $w$ component can be written on the following form:

$$
\frac{fS_{uu}(f)}{\sigma_u^2} = \frac{4f_u}{(1 + 70.8f_u^2)^{5/6}},
$$

(4.1)
and

\[
\frac{f S_{ww}(f)}{\sigma_{w}^2} = \frac{4f_w \left(1 + 755.2f_w^2\right)}{(1 + 283.2f_w^2)^{11/6}}.
\]  

(4.2)

Here, \( f \) is the frequency, \( \sigma_i, i = \{u, w\} \) is the variance of \( i(t) \) and \( f_i = f^x L_i / U \) where \( x L_i \) is the characteristic length scale of the \( i \)th component. The spectra show good correspondence to the Kolmogorov energy cascade and also follow the -5/3 slope of the inertial subrange in the range of resolved scales. It is also clear that RBVMS, unlike pure Galerkin methods, provides effective dissipation at high frequencies.

**Remark 4.1.** We emphasize that the state of isotropic turbulence is only expected in the channel core.

**Remark 4.2.** Because the cutoff frequency is inversely proportional to the mesh element size, the cost for resolving higher frequencies increase rapidly. The channel flow computa-
Fig. 7: Scatter plot of the stream-wise and span-wise velocity fluctuations at the channel centerline. Left: $Re_\tau = 395$; Right: $Re_\tau = 2000$.

Fig. 8: Spectra of the stream-wise velocity fluctuations for $Re_\tau = 395$ and $Re_\tau = 2000$ with von Kármán fitting.

Simulations were set up with buffeting forces in mind, for which the current frequency content is in the range of interest.

5. FEM simulations of the bridge section

In this section, we report the results of the simulations of the rectangular bridge section subjected to the wind fields presented in Sec. 4. We also carry out a simulation using a uniform inflow BCs for comparison.
5.1. Problem setup

The computational setup for this case consists of a bridge sectional model that is centered between the walls that mimic those of the wind tunnel. For the cross-section we consider a simple rectangular cylinder with aspect ratio 1:10, shown in Fig. 1, with physical dimensions of 0.5 × 0.05 m. The section is referred to as “BD10”. The inflow surface is placed 2.5 B upwind and the outflow surface is placed 6 B downwind from the section centroid. The domain width is set to 1.0 m and the section width is that of the entire domain.

The simulations make use of linear tetrahedra. To resolve the incoming turbulence a refinement region that extends from the inlet and 3 B downwind from the section is defined. A smaller refinement region around the shear layers is also introduced. At the section’s surface, four layers of prismatic boundary-layer elements of thickness 0.3 mm are constructed. In total, the mesh has 1.2 M nodes and 6.4 M elements. The mesh is partitioned into 512 subdomains using METIS, and each subdomain is assigned a compute core.

The boundary conditions are defined as follows. At the walls, no-slip BC is enforced weakly, as a continuation of the channel computation. The outlet is set to have a zero traction BC, and, at the lateral boundaries, no-penetration BCs are adopted. Weakly-enforced no slip and no-penetration BCs are set at the bridge-section surface. At the inlet, velocity field from the channel flow simulation is imposed weakly. Fluid properties of air, with density $\rho = 1.225$ kg/m$^3$ and dynamic viscosity $\mu = 1.848 \times 10^{-5}$ kg/ms, are employed in the bridge section simulations. The time step is set to $\Delta t = 1 \times 10^{-4}$ s for all cases.

The aerodynamic forces and pressures are integrated at seven strips of 10 mm width. The strips are positioned such that we obtain span-wise distances of $nB/50$, $n = \{1, 2, 4, 8, 16, 32\}$, as shown in Fig. 10. The strips are centered in the domain as to eliminate any effects that arise from the span-wise boundary conditions. A sufficiently narrow
force measurement strips were chosen so that the buffeting forces can be assumed fully correlated, and the span-wise coherence can be set to unity. The corresponding velocity sampling points were placed at a distance $B$ upwind of the bridge-section leading edge.

5.2. Buffeting and aerodynamic admittance results

![Diagram of velocity and force sampling layout.](image)

**Fig. 10:** Velocity and force sampling layout.

The $Re_f = 395$ and $Re_f = 2000$, with $U_b = 2 m/s$, are simulated for 50 s, of which we discard the first 2.5 s to let the flow fully develop. A 5 s window of the drag and lift coefficients for the force measuring strip located in the middle of the section is shown in Fig. 11. Visualization of the flow for the $Re_f = 395$ and uniform-inflow simulations are
shown in Fig. 12.

The time series give a good first impression of the effect of incident turbulence. First of all, we notice that besides the vortex shedding frequency there is an additional, and more irregular component that we recognize as a buffeting force. The time series further suggests that the buffeting components of lift and pitching moment are highly correlated, and seemingly independent of the turbulence-induced drag, which speaks in favor of the auto-spectral assumption of buffeting forces (see Eq. (2.6)). Finally, we note that the average drag coefficient increases from approximately 1.05 to 1.08 when the bridge section is subjected to turbulent flow. The turbulent wakes shown in Fig. 12 further prove the irregularity the that incident turbulence triggers. We clearly see that whereas the vortices are almost fully correlated for uniform inflow, they have a much more 3D and random character for the turbulent inflow case.
The spectra for the drag, lift and pitching moment, averaged over the measurement strips, are shown in Fig. 13. The buffeting is recognized as the differences between the curves of turbulent and uniform inflow, which do reflect the velocity spectra well (Figs. 8 and 9), limited by the domain size at the lower end ($K \approx 0.4$) and the mesh density as the upper limit ($K \approx 15$). We further see that, at higher frequencies, the force spectra are equal for all inflow conditions and recognize this as the “signature turbulence” of the bridge. The
vortex shedding frequency ($K = 12.1 \Rightarrow St = 0.192$), which contains almost all the energy of the uniform inflow, is hardly notable for the turbulent cases. This observation is supported by Fig. 12, and also clearly demonstrates the well-known fact that turbulence disturbs the vortex formation and no regular shedding is seen.86

We now use the velocity and force spectra to identify the aerodynamic admittance functions by Eq. (2.6). The results are shown in Fig. 14. For comparison, we have also

Fig. 14: Aerodynamic admittance functions.
plotted the Sears function given by Eq. (2.7), which represents the airfoil solution for lift admittance. The drag coefficient was set to $C_D = 1.05$, taken from the uniform inflow time series, and the slope of the lift and pitching moments were set to $C'_L = 8.60$ and $C'_M = 1.089$, respectively. The latter were determined through wind tunnel experiments as a part of our previous work.\textsuperscript{21}

By utilizing the essential property of linearity in the frequency domain,\textsuperscript{118} we isolate the buffeting action by subtracting the uniform spectra from the turbulent ones. This effect is negligible for lower frequencies, but makes a significant difference when approaching the vortex shedding frequency. It should be remarked that the validity of the computed admittance functions relies on sufficient frequency content of the velocity spectra, which, in our case, corresponds to the range of $K \approx 0.4$ to 15.

The computed admittance functions suffer from relatively low resolution due to limited duration of the time series, which is a limitation for numerical simulations in general. Nevertheless, we still capture the admittance properties of the bridge deck. For the drag, we see that the admittance has a constant value of unity, i.e., the horizontal velocity fluctuations are directly transferred to the drag forces as in the quasi-steady theory. For the admittances associated with lift and pitching moment, however, we see that they exhibit the same asymptotic behavior as the Sears function. The lift has a slightly lower magnitude, which is consistent with the observations reported in,\textsuperscript{10} where the same section was studied. For the pitching moment, which lies above the Sears function, the section seems to receive full admittance to a certain level before it decays. The computed admittance functions generally confirm that the BD10 section exhibit very similar buffeting response to that of an airfoil.

5.3. \textit{Temporal statistics}

In this section we study the impact of incident turbulence on the time-averaged flow characteristics. The data are sampled from $t = 15$ s to $t = 20$ s with a sampling frequency of 0.1 s. The pressure, reported in terms of the pressure coefficient $C_p = p/(1/2 \rho \overline{U^2})$, is averaged in the span-wise direction over 75% of the span in order to eliminate boundary-condition effects at the bridge-section edges.

Figures 15 and 16 show the time-averaged pressure coefficient and velocity contours, respectively, for the uniform and turbulent inflow cases. It is clear from the figures that turbulence in the flow makes the reattachment length significantly lower. In addition, the peak base suction increases near the leading edge. This observation was also made experimentally for the same section in,\textsuperscript{86} where the authors explained the increased base suction with the shorter vortex formation length and thereby increased shedding of the vortices. The results are also consistent with a drastic reduction of the separation bubble for turbulent inflow around a sphere shown in.\textsuperscript{82} We further note that the pressure profiles for the two turbulence intensities are very similar. This suggests that the vortex formation depends as much on the presence of disturbances in the incoming flow, as the intensity or scales of the turbulence. These observations may be of significant interest for CFD applications that use experimental results for validation, in that classical assumption of uniform
inflow may overestimate the reattachment length. For non-zero angle of attack, or non-symmetrical sections, this effect gives an overestimation of the lift and pitching moment, which is a general, but not yet explained, result in many numerical simulation of bridges (see e.g.,\textsuperscript{119,19,21,25}).

6. Conclusions

A framework to carry out buffeting analysis of bridge sections is presented. The framework makes use the RBVMS formulation of the Navier–Stokes equations of incompressible flows, weakly enforced BCs, and an MDM approach that couples a turbulent inflow generation domain with a domain for the analysis of the bridge section. Coupling between the two domains makes use of weak BCs as a core technology, and naturally enables different discretizations in both subdomains. The turbulence-generating domain makes use of a pressure-driven channels flow setup, where the turbulent flow, and, in particular, the turbulence intensity, is controlled using the bulk velocity and friction-velocity-based Reynolds number. The channel simulations produced highly isotropic and realistic velocity spectra in the channel core with effective high-frequency dissipation. This accuracy was achieved due to the use of RBVMS, weak BCs, and a discretization using NURBS-based IGA. To illustrate the modularity of the framework, the bridge section was simulated using RBVMS and weak BCs, but discretized using traditional low-order FEM.

From the aerodynamic forces on the bridge section we separated buffeting from vortex shedding and signature turbulence by subtracting the uniform-inflow force spectra. The resulting aerodynamic admittance functions for the lift and pitching moment were shown to exhibit a buffeting response that follows the Sears analytical solution for airfoils.\textsuperscript{2} It was further shown that incident turbulence had a significant effect on the flow patterns around
the bridge section. As observed earlier in,\sup{86,82} under turbulent inflow, the separation-bubble reattachment length was reduced, which, in turn, gave rise to a pressure profile that differs significantly from the uniform-inflow case. Because the averaged drag, flow velocity, and pressure coefficient are very similar for two different values of \( \text{Re}_\tau \), we hypothesize that these quantities are more dependent on the presence of ambient turbulence than on the details of turbulent structures. This statement, however, needs further investigation. Nevertheless, these observations explain why numerical simulations of bluff body aerodynamics using uniform inflow generally overestimate the magnitudes of lift and pitching moment.
Acknowledgments

This work was carried out with financial support from the Norwegian Public Roads Administration. YB was partially supported by the NSF Award No. 1854436. All simulations were performed using resources provided by UNINETT Sigma2 - the National Infrastructure for High Performance Computing and Data Storage in Norway. TAH was partially funded by Dr.Techn. Olav Olsen AS. The authors greatly acknowledge this support.

References

14. D.E. Walshe and T.A. Wyatt. Measurement and application of the aerodynamic ad-


28. Y. Bazilevs, V.M. Calo, J.A. Cottrell, T.J.R. Hughes, A. Reali, and G. Scovazzi. Vari-
REFERENCES


42. K. Takizawa, T. E. Tezduyar, J. Boben, N. Kostov, C. Boswell, and A. Buscher. Fluid–
structure interaction modeling of clusters of spacecraft parachutes with modified geo-
for aerodynamic-moment calculations from parachute FSI modeling. *Computational
44. K. Takizawa, T. E. Tezduyar, C. Boswell, R. Kolesar, and K. Montel. FSI modeling
of the reefed stages and disreefing of the Orion spacecraft parachutes. *Computational
Multiscale methods for gore curvature calculations from FSI modeling of spacecraft
46. K. Takizawa, T. E. Tezduyar, and R. Kolesar. FSI modeling of the Orion spacecraft
47. K. Takizawa, T. E. Tezduyar, and T. Terahara. Ram-air parachute structural and fluid
methods for compressible-flow aerodynamics of parachutes with geometric porosity.
49. T. Kanai, K. Takizawa, T. E. Tezduyar, T. Tanaka, and A. Hartmann. Compressible-
flow geometric-porosity modeling and spacecraft parachute computation with
isogeometric discretization. *Computational Mechanics*, published online, DOI:
50. K. Takizawa, T. E. Tezduyar, and H. Hattori. Computational analysis of flow-driven
Turbocharger flow computations with the Space–Time Isogeometric Analysis (ST-
52. Y. Otoguro, K. Takizawa, and T. E. Tezduyar. Space–time VMS computational flow
analysis with isogeometric discretization and a general-purpose NURBS mesh generation
53. Y. Otoguro, K. Takizawa, and T. E. Tezduyar. A general-purpose NURBS mesh generation
55. K. Takizawa, T. E. Tezduyar, and T. Kuraishi. Multiscale ST methods for thermo-


68. A. Korobenko, J. Yan, S.M.I. Gohari, S. Sarkar, and Y. Bazilevs. FSI Simulation of


80. T. Tezduyar and Y. Osawa. The Multi-Domain Method for computation of the aero-


111. D.B. Spalding. A signel formula for the law of the wall. *Journal of Applied Mechan-
REFERENCES


