

NUMERICAL APPROXIMATION OF AEROACOUSTICS INDUCED BY FLOW OVER A TWO-DIMENSIONAL SQUARE CYLINDER

T. Marhan¹ P. Sváček¹

¹ Department of Technical Mathematics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technická 4, 166 01 Prague 6, Czechia

Abstract

This study outlines a possible approach to aeroacoustics. An unsteady simulation of a 3D flow field in vicinity of a square cylinder is performed within the OpenFOAM framework using $k - \omega$ SST turbulence model. Afterwards, an in-house implementation of Lighthill's acoustic analogy is discussed using FEM in 2D. The study concludes by presenting results both in the fluid and acoustic domains.

Keywords: fluid-acoustics interaction, computational aeroacoustics, lighthill's acoustic analogy, computational fluid dynamics, finite volume method, PIMPLE algorithm, turbulence modeling, finite element method

1 Introduction

Turbulent flow around specifically shaped obstacles, such as cylinders (whether rectangular or circular) is of great interest in various technical and scientific applications, see e.g. [1]. Furthermore, describing the turbulent air flow over structures such as buildings, bridges, or bluff-body vehicles holds significant importance in many technical disciplines, such as aeroelasticity or aeroacoustics, see e.g. [2], or more recent [3].

In this paper, we focus on computational aeroacoustics. In particular, the problem of 3D flow over a square cylinder is approximated, where separation of the flow appears, resulting in the formation of strong vortices in the wake region, famously known as the von Kármán vortex street. This phenomenon creates acoustic waves, which are further propagated throughout space. Computational Fluid Dynamics (CFD) simulations are carried out using the OpenFOAM framework, employing a finite volume solver. In contrast, for the Computational Aeroacoustics (CAA) part, an in-house code based on the finite element method is utilized.

2 Mathematical model

In this section, the hybrid CAA methodology is applied, offering the advantage of a distinct treatment for fluid and acoustic computations. The computational domain is divided into two regions: a non-linear source region (near field), where the sound is generated from unsteady fluid flow, and a wave propagation region (far field), where the fluid is at rest and the sound is only propagated. Different physical models and discretization methods are applied for the flow and acoustic computations.

In the source region, the unsteady flow is computed using the $k - \omega$ -SST DES turbulence model. Subsequently, acoustic sources are extrapolated from the flow field, serving as inputs for the acoustic propagation computation. Throughout this process, it is assumed that acoustic propagation has no significant influence on the flow field.

2.1 Finite volume discretization of incompressible Navier-Stokes equations

The finite volume method (FVM) is used to evaluate partial differential equations describing the fluid domain. It involves the subdivision of the computational domain $\mathcal{D}$ into a finite number of closed, non-overlapping cells or control volumes. By using the integral form of the equations, partial differential equations are transformed into a system of algebraic equations in matrix form with a finite number of unknowns.

Partial differential equations describing the flow of an incompressible Newtonian fluid in a domain $\mathcal{D} \times (0, T)$ are known as Navier-Stokes equations, in the form

$$\frac{\partial u_i}{\partial x_i} = 0 \quad (1)$$

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i}{\partial x_j} u_j + \frac{\partial p}{\partial x_i} - \frac{\partial}{\partial x_j} \left(\nu \frac{\partial u_i}{\partial x_j} \right) = f_i. \quad (2)$$

Here, u_i represents the velocity, components p denotes kinematic pressure (pressure divided by density), and ν stands for kinematic viscosity. External volume forces f_i are assumed to be zero. Eq. (1) and (2) are supplemented by appropriate boundary and initial conditions to ensure a unique solution, see e.g. [4]. The momentum equations (2) can be reformulated as

$$\frac{\partial u_i}{\partial t} + \frac{\partial}{\partial x_j} \left(u_i u_j + p \delta_{ij} - \nu \frac{\partial u_i}{\partial x_j} \right) = 0. \quad (3)$$

The basic idea of the FVM is to enforce the momentum equation (3) and the continuity equation (1) for a control volume V_k with its boundary surface S_k located in the computation domain $\mathcal{D} = \bigcup_{k \in J} V_k$. Integration of Eqs. (1) and (3) together with the use of the Gauss's theorem yields

$$\oint_{S_k} u_i n_i dS = 0 \quad (4)$$

$$\int_{V_k} \frac{\partial u_i}{\partial t} dV + \oint_{S_k} \left(u_i u_j + p \delta_{ij} - \nu \frac{\partial u_i}{\partial x_j} \right) n_j dS = 0, \quad (5)$$

where $\mathbf{n} = (n_1, n_2, n_3)$ represents the unit normal vector to the surface outward from the volume. The integrated Eqs. (4) and (5) can be expressed in the conservative vector form

$$\int_{V_k} \frac{\partial \mathbf{W}}{\partial t} dV + \oint_{S_k} (\mathbf{F}_c - \mathbf{F}_v) dS = 0. \quad (6)$$

Here, $\mathbf{W}$, $\mathbf{F}_c$, $\mathbf{F}_v$ are vector of the variables, vector of the convective fluxes, and vector of the viscous fluxes, respectively. Each of these vectors comprises in three dimensions the following four components: $\mathbf{W} = [0, u, v, w]^T$ and

$$\mathbf{F}_c = \begin{bmatrix} U \\ u_1 U + n_1 p \\ u_2 U + n_2 p \\ u_3 U + n_3 p \end{bmatrix}, \quad \mathbf{F}_v = \nu \begin{bmatrix} 0 \\ \frac{\partial u_1}{\partial x_1} n_1 + \frac{\partial u_1}{\partial x_2} n_2 + \frac{\partial u_1}{\partial x_3} n_3 \\ \frac{\partial u_2}{\partial x_1} n_1 + \frac{\partial u_2}{\partial x_2} n_2 + \frac{\partial u_2}{\partial x_3} n_3 \\ \frac{\partial u_3}{\partial x_1} n_1 + \frac{\partial u_3}{\partial x_2} n_2 + \frac{\partial u_3}{\partial x_3} n_3 \end{bmatrix},$$

where U represents the contravariant velocity defined as $U = u_i n_i$ (i.e. the velocity normal to the surface S_k), see e.g. [4]. By volume-averaging of the variables $\mathbf{W}$, the average value of $\mathbf{W}$ in V_k reads

$$\mathbf{W}_k = \frac{1}{|V_k|} \int_{V_k} \mathbf{W} dV \quad (7)$$

and assuming that the control volume V_k remains constant over time, the time derivative of the conservative variables $\mathbf{W}$ can be cast in the form

$$\int_{V_k} \frac{\partial \mathbf{W}}{\partial t} dV = |V_k| \frac{d\mathbf{W}_k}{dt}. \quad (8)$$

Herewith, Eq. (6) becomes

$$\frac{d\mathbf{W}_k}{dt} = -\frac{1}{|V_k|} \left[\oint_{S_k} (\mathbf{F}_c - \mathbf{F}_v) dS \right], \quad (9)$$

for any $k \in J$. For more details, refer to sources such as [5].

Implementation of PIMPLE algorithm within the OpenFOAM framework

The spatial and temporal discretization is implemented within the OpenFOAM framework, see e.g. [6]. OpenFOAM predominantly adopts a co-located variable arrangement on the grid, where fluid dynamic properties are stored at a single point within a control volume, which is the centroid of the control volume. Values at other locations are obtained through interpolation.

Solving the Navier-Stokes equations is intricate due to the absence of an independent equation for pressure, whose gradient contributes to each of the three momentum equations. While the momentum equations distinctly determine the velocity components, the continuity equation is left to determine the pressure. A *pressure-based* method is employed, which derives the Poisson equation for pressure from the momentum equations, see e.g. [7]. Within OpenFOAM, implicit pressure-correction methods are utilized to solve the pressure equation, namely the semi-implicit method for pressure-linked equations (SIMPLE) algorithm and the pressure-implicit algorithm with the splitting of the operator (PISO). The combination of these methods results in the PIMPLE algorithm, which is used in this study. The solution algorithm for these methods can be summarized as follows

1. Start the calculation of the fields at the new time t_{n+1} using the latest solution u_i^n and p^n as starting estimates for u_i^{n+1} and p^{n+1} .
2. Assemble and solve the linearized algebraic equation systems for the velocity components (momentum equations) to obtain u_i^{m*} (predicted velocity for outer iteration m , which does not yet satisfy the continuity equation).
3. Assemble and solve the pressure-correction equation to obtain p' .
4. Correct the velocities and pressure to obtain the velocity field u_i^m (which satisfies the continuity equation) and the new pressure p^m .
 - a. SIMPLE algorithm only makes one correction
 - b. Whereas the PISO algorithm solves the pressure-correction a few more times, but typically not more than 4 and always corrects both velocities and pressure.
5. Return to step 2 and repeat, using u_i^m and p^m as improved estimates for u_i^{n+1} and p^{n+1} , until all corrections are negligibly small.
6. Advance to the next time step.

Solving both pressure and velocity equations involves the use of the *under-relaxation* technique, which limits the amount a variable can change between each outer iteration. Further details about these algorithms can be found in [7].

2.2 Turbulence modeling

Detached-Eddy Simulation (DES) serves as a hybrid modeling approach for treating turbulence, focusing on predicting unsteady separated flows. This method bridges Large Eddy Simulation (LES) with Reynolds-Averaged Navier-Stokes (RANS) and emphasizes a manageable computation effort. DES incorporates RANS in the attached boundary layer and LES in the separated region within a single formulation. In essence, the turbulence model functions as a sub-grid-scale (SGS) model in regions where the grid density is fine enough for LES and as a RANS model in regions where it is not. The DES was initially introduced in [8] for one-equation Spalart-Allmaras (S-A) turbulence model, later, it was applied to the two-equation $k - \omega$ Shear Stress Transport ($k - \omega$ SST) turbulence model, see [9]. For the present study, the $k - \omega$ SST DES turbulence model with wall functions, is utilized to simulate separated flows, particularly for noise prediction.

In the context of LES, a fine grid is defined where the largest dimension of the grid cell $\Delta = \max(\Delta x, \Delta y, \Delta z)$ is significantly smaller than the flow turbulence length-scale δ_t (integral length scale of the turbulence, much larger than the Kolmogorov scale). In these LES regions,

the turbulence model has little control, while in the RANS regions, it exerts full control over the solution. The RANS $k - \omega$ SST model is formulated as follows

$$\begin{aligned} \frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} &= \tilde{P}_k - \beta^* \rho k \omega + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_k \mu_t) \frac{\partial k}{\partial x_i} \right] \\ \frac{\partial(\rho \omega)}{\partial t} + \frac{\partial(\rho u_i \omega)}{\partial x_i} &= \alpha \rho S^2 - \beta \rho \omega^2 + \frac{\partial}{\partial x_i} \left[(\mu + \sigma_\omega \mu_t) \frac{\partial \omega}{\partial x_i} \right] + 2(1 - F_1) \rho \sigma_{\omega^2} \frac{1}{\omega} \frac{\partial k}{\partial x_i} \frac{\partial \omega}{\partial x_i}, \end{aligned} \quad (10)$$

where k represents turbulence kinetic energy, and ω is the turbulence dissipation rate. The turbulent eddy viscosity is defined as

$$\nu_t = \frac{a_1 k}{\max(a_1 \omega, S F_2)}. \quad (11)$$

For the model parameters $\tilde{P}_k, \sigma_k, \beta, \beta^*, S, \sigma_\omega, \sigma_{\omega^2}, a_1, F_1, F_2$ and detailed description of the model refer to [10]. To adapt the $k - \omega$ SST model into the DES version, the length scale involved in the RANS version is appropriately modified. The length scale of the model is given by

$$d_{k-\omega} = \frac{k^{1/2}}{\beta^* \omega}. \quad (12)$$

The DES length scale is then defined as

$$\tilde{d} = \min(d_{k-\omega}, C_{DES} \Delta), \quad (13)$$

where C_{DES} is the only new adjustable model constant, as described in [9]. The only term of the $k - \omega$ SST model that has been modified to transform it into the DES version is the dissipative term of the k -transport equation, that is

$$D_{RANS}^k = \beta^* \rho k \omega = \frac{\rho k^{3/2}}{d_{k-\omega}}, \quad (14)$$

where $\tilde{d}$ is substituted for $d_{k-\omega}$, resulting in

$$D_{DES}^k = \frac{\rho k^{3/2}}{\tilde{d}}. \quad (15)$$

For computation, the constant C_{DES} is chosen to be equal to one.

2.3 Aeroacoustics

The employed hybrid CAA approach in this study is based on the Lighthill's acoustic analogy, see [11]. The inhomogeneous wave equation governing the generation and propagation of sound, written in terms of the acoustic density fluctuation, ρ' reads

$$\frac{\partial^2 \rho'}{\partial t^2} - c_0^2 \frac{\partial^2 \rho'}{\partial x_i^2} = \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j}, \quad (16)$$

where T_{ij} is the Lighthill's tensor defined as

$$T_{ij} = \rho u_i u_j + [(p - p_0) - c_0^2 (\rho - \rho_0)] \delta_{ij} - \tau_{ij}. \quad (17)$$

Here, ρ_0, p_0 and c_0 denote the atmospheric values of density, pressure and speed of sound respectively. For isentropic flows, the relation

$$(p - p_0) = c_0^2 (\rho - \rho_0) \quad (18)$$

is employed. Moreover, the effects of viscosity have a significant influence only over very large distances, allowing for the possibility to neglect the viscous term τ_{ij} entirely, see [12]. Consequently, the resulting approximation of Lighthill's tensor is given by

$$T_{ij} \approx \rho u_i u_j. \quad (19)$$

For linear acoustic propagation, the isentropic relation (18) can be rewritten as

$$p' = c_0^2 \rho'. \quad (20)$$

Using this relation leads to the acoustic analogy formulation in terms of pressure perturbations

$$\frac{1}{c_0^2} \frac{\partial^2 p'}{\partial t^2} - \frac{\partial^2 p'}{\partial x_i^2} = \frac{\partial^2 T_{ij}}{\partial x_i \partial x_j}. \quad (21)$$

Assuming that the density fluctuations are negligible in the unsteady flow, an estimate for the Lighthill's tensor is obtained as $T_{ij} \simeq \rho_0 u_i u_j$ and Eq. (21) takes the form

$$\frac{1}{c_0^2} \frac{\partial^2 p'}{\partial t^2} - \frac{\partial^2 p'}{\partial x_i^2} = \rho_0 \frac{\partial^2 (u_i u_j)}{\partial x_i \partial x_j}. \quad (22)$$

In this study, we presume that the pressure fluctuation p' is equal to the acoustic pressure p^a both in the source region and far-field region.

3 Finite element approximation of Lighthill's acoustic analogy

Finite element method (FEM) is utilized in this study to solve the variation formulation of Lighthill's acoustic analogy. The acoustic solution is obtained at discrete points within the numerical domain, as illustrated in Fig. 1. Therein, Ω_1 denotes the computation domain, where the inhomogeneous wave equation needs to be solved using acoustic sources obtained from the hydrodynamic velocity field, and Ω_2 denotes the domain outside the source region, where acoustic sources become zero.

For the mixed-value problem, solving for $p' : \bar{\Omega} \times (0, T) \rightarrow \mathbb{R}$, we consider initial conditions $p'(\vec{x}, 0) = p'_0$, $\frac{\partial p'}{\partial t}(\vec{x}, 0) = \dot{p}'_0$ for $\vec{x} \in \Omega$ and boundary conditions given as

$$\frac{\partial p'}{\partial x_i} n_i = -\frac{1}{c_0} \frac{\partial p'}{\partial t} \quad \text{on } \Gamma_2 \times (0, T) \quad (23)$$

$$\frac{\partial p'}{\partial x_i} n_i = 0 \quad \text{on } \Gamma_B \times (0, T). \quad (24)$$

Condition (23) is a first-order approximation of the absorbing boundary condition, where the unit normal vector n_i points out of the boundary towards the infinite domain, indicating the direction of the outgoing wave. Condition (24) represents the sound-hard boundary condition, describing behavior where sound waves are fully reflected back to the acoustic domain.

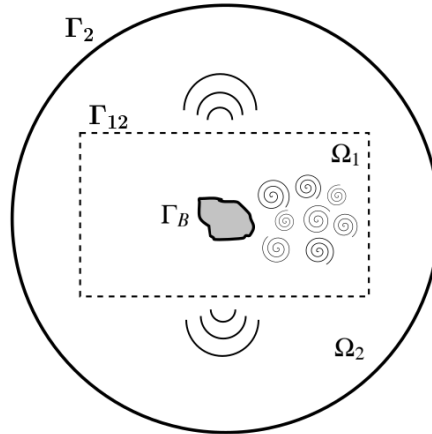


Figure 1: Schematic of hybrid domain

In order to approximate the inhomogeneous wave equation (22) using FEM, the equation is multiplied by a test function $w \in \mathcal{V} = H^1(\Omega)$ and integrated over the entire acoustic domain $\Omega = \Omega_1 \cup \Omega_2$. This yields

$$\frac{1}{c_0^2} \left(\frac{\partial^2 p'}{\partial t^2}, w \right)_{\Omega} - \left(\frac{\partial^2 p'}{\partial x_i^2}, w \right)_{\Omega} = \rho_0 \left(\frac{\partial^2 (u_i u_j)}{\partial x_i \partial x_j}, w \right)_{\Omega}, \quad (25)$$

where by $(\cdot, \cdot)_{\Omega}$ the dot L_2 product is denoted. After applying Green's integration theorem to the second spatial derivate of p' as well as to the acoustic source term on the right-hand side, the equation can be rearranged into the variational (weak) formulation of Lighthill's acoustic analogy

$$\frac{1}{c_0^2} \left(\frac{\partial^2 p'}{\partial t^2}, w \right)_{\Omega} + \left(\frac{\partial p'}{\partial x_i}, \frac{\partial w}{\partial x_i} \right)_{\Omega} - \left(\frac{\partial p'}{\partial x_i} n_i, w \right)_{\partial\Omega} = \rho_0 \left(\frac{\partial (u_i u_j)}{\partial x_j} n_i, w \right)_{\partial\Omega} - \rho_0 \left(\frac{\partial (u_i u_j)}{\partial x_j}, \frac{\partial w}{\partial x_i} \right)_{\Omega}. \quad (26)$$

Now we apply boundary conditions (23) and (24) for the boundary term on the left-hand side of Eq. (26), whereas the boundary term on the right-hand side equals zero, see [13]. This leads us to the weak formulation of Eq. (22), which may be stated as, find $p' \in \mathcal{V}$ such that

$$\frac{1}{c_0^2} \left(\frac{\partial^2 p'}{\partial t^2}, w \right)_{\Omega} + \frac{1}{c_0} \left(\frac{\partial p'}{\partial t}, w \right)_{\Gamma_2} + \left(\frac{\partial p'}{\partial x_i}, \frac{\partial w}{\partial x_i} \right)_{\Omega} = -\rho_0 \left(\frac{\partial (u_i u_j)}{\partial x_j}, \frac{\partial w}{\partial x_i} \right)_{\Omega_1}, \quad (27)$$

is fulfilled for all $w \in \mathcal{V}$.

The semi-discrete Galerkin formulation is obtained from the weak formulation of an equation after discretization of the domain and the introduction of finite element spaces. A finite-dimensional finite element space $V_h \subset \mathcal{V}$ with dimension n is chosen, and the solution $p' \in \mathcal{V}$ is approximated by $p_h \in V_h$ written as a time-dependant linear combination of coefficients $p'_j(t)$ and basic functions $\varphi_j(x) \in V_h$, i.e.

$$p'(x, t) \approx p_h(t, x) = \sum_{j=1}^n p_j(t) \varphi_j(x). \quad (28)$$

Using this relation in Eq. (27) with $w_h = \varphi_i$ for $i = 1, \dots, n$ leads to the second-order system of ODEs for an unknown vector $\mathbf{p}(t) = \{p_j\}_{j=1}^n$ in matrix form

$$\frac{1}{c_0^2} \mathbb{M} \ddot{\mathbf{p}}(t) + \frac{1}{c_0} \mathbb{D} \dot{\mathbf{p}}(t) + \mathbb{K} \mathbf{p}(t) = \mathbf{b}(t), \quad (29)$$

where the matrices $\mathbb{M} = \{m_{ij}\}_{i,j=1}^n$, $\mathbb{D} = \{d_{ij}\}_{i,j=1}^n$, $\mathbb{K} = \{k_{ij}\}_{i,j=1}^n$ and the vector $\mathbf{b} = \{b_i\}_{i=1}^n$ are computed as follows

$$m_{ij} = (\varphi_j, \varphi_i)_{\Omega}, \quad d_{ij} = (\varphi_j, \varphi_i)_{\Gamma_2}, \quad k_{ij} = \left(\frac{\partial \varphi_j}{\partial x_l}, \frac{\partial \varphi_i}{\partial x_l} \right)_{\Omega}, \quad b_i = -\rho_0 \left(\frac{\partial (u_j u_l)}{\partial x_l}, \frac{\partial \varphi_i}{\partial x_j} \right)_{\Omega_1}.$$

Problem (29) is discretized in time with the aid of Newmark's scheme, see e.g. [14]. The results presented have been obtained using bilinear finite elements and quadrilaterals.

4 Numerical results

The numerical domain is depicted in Fig. 2, where the center of the square cylinder is situated at $(x, y) = (10B, 5.5B)$ with $B = 20$ mm. The boundary conditions are detailed in Tab. 1. The dimensions of cells at the outer boundary equal to about 5 mm. The mesh gradually refines until it reaches a refinement region around the cylinder, with a cell size of around 1 mm. Finally, a thin boundary layer with a stretching factor of 1.1 is present, with the smallest dimensions reaching $5 \cdot 10^{-2}$ mm. This results in a mesh with a total number of approximately 300 000 control volumes.

The inlet velocity $U_{\infty} = 10$ m/s, cylinder dimension $B = 20$ mm, and the kinematic viscosity for air at 20°C result in a Reynolds number of around 13 300. The fluid computation utilizes a second-order spatial discretization for velocity and a first-order for turbulence characteristics. At

the inlet, the turbulence kinetic energy is set to $k = 0.01 \text{ m}^2/\text{s}^2$, along with a turbulence dissipation rate of $\omega = 1000 \text{ 1/s}$. Temporal discretization is performed with a first-order Euler time-stepping scheme. The computation in the fluid domain is run for 0.2 s with a time step of $20 \text{ }\mu\text{s}$.

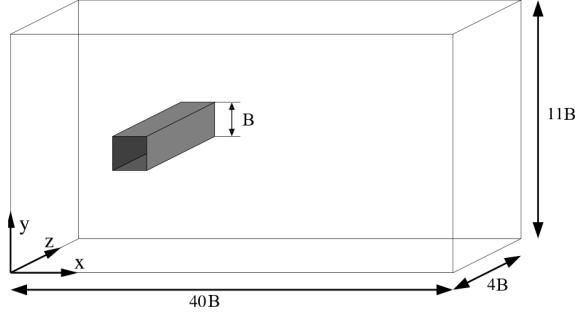


Figure 2: Numerical domain

Table 1: Boundary Conditions

position	boundary condition
$x = 0$	inlet, fixed velocity U_∞
$x = 40B$	outlet, convective exit
$y = 0, y = 11B$	symmetry
$z = 0, z = 4B$	periodic
cylinder	no slip

Lift coefficient in time is displayed in Fig. 3. The Fourier transform of the lift coefficient variation reveals the main frequency of vortex-shedding, i.e. 70 Hz , which results in a Strouhal number of 0.14 . The observed value lies in the range of experimentally measured values, see Escobar et al. [15]. Fig. 4 presents the time-averaged axial velocity component at the centerline of a plane at $z = 2B$. The obtained results are compared with experimental data from Durão et al. [16] with a good agreement. In addition, a snapshot of the velocity and pressure field is depicted at a specific time in Fig. 5, highlighting the presence of strong vortices in the cylinder's wake.

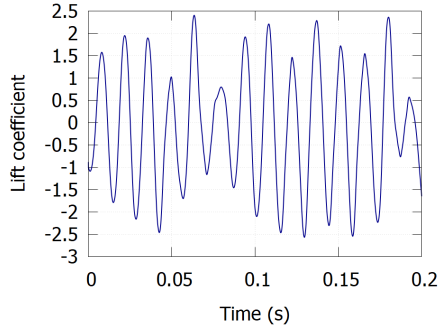


Figure 3: The lift coefficient variation on the square cylinder as a function of time

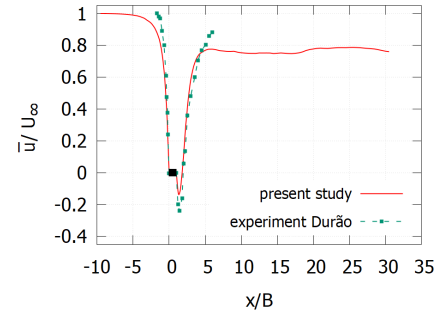


Figure 4: Time-averaged axial velocity component alongside the centerline of a plane at $z = 2B$

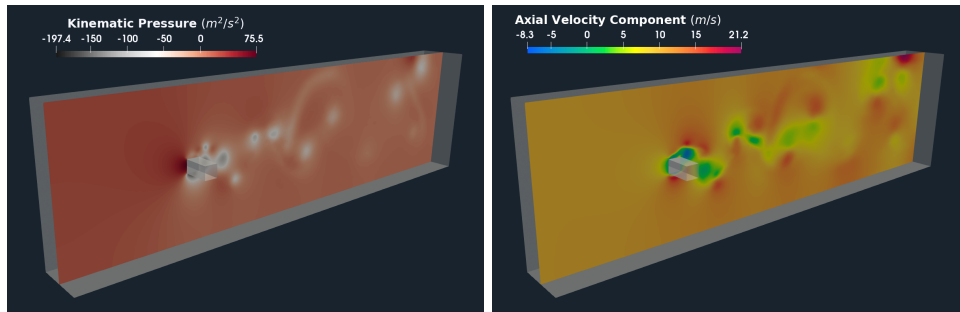


Figure 5: Slice of pressure and velocity field at $z = 2B$ at $t = 0.115 \text{ s}$

In the finite element implementation of Lighthill's acoustic analogy, as discussed in Chap. 3, the acoustic source term $\frac{\partial(u_i u_j)}{\partial x_j}$ from Eq. (27) is directly computed on the finer grid of the fluid

domain. A 2D field is then extracted at $z = 2B$ from the 3D fluid simulation for each time step. These acoustic sources are subsequently interpolated to the near-field region Ω_1 of the acoustic grid, see Fig. 6. The numerical grid for the acoustic computation has an element size equal to 10 mm in the near-field region, and around the cylinder, the element size progressively decreases to 0.9 mm. The radius of the far-field region is $r = 10$ m, with the cylinder located at the center. The mesh becomes noticeably coarser in this region, with an element size reaching 30 cm at the outermost boundary. The total number of elements in the mesh for the acoustic computation is approximately 19 000.

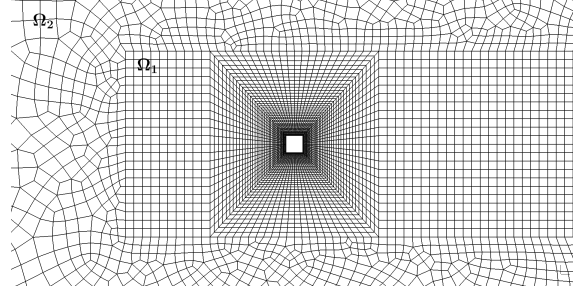


Figure 6: Section of acoustic grid depicting near-field region Ω_1 , where sources are interpolated and the far-field region Ω_2

The acoustic source values are assessed at two different points $P_1 = (15B, 5.5B)$ and $P_2 = (20B, 8.5B)$. The first point is positioned downstream along the x -axis, second point is located further away, outside the central region of the wake. Fig. 7 displays the results in both the time and frequency domains. The frequency spectrum reveals that 140 Hz stands out as the dominant frequency for point P_1 . In contrast, several other frequency components are found within the spectrum for point P_2 , with 70 Hz being the primary one, which is the main frequency for vortex-shedding. Additionally, in [15], it was experimentally observed that the frequency of fluctuations is twice as high in the wake center compared to outside, aligning with our study.

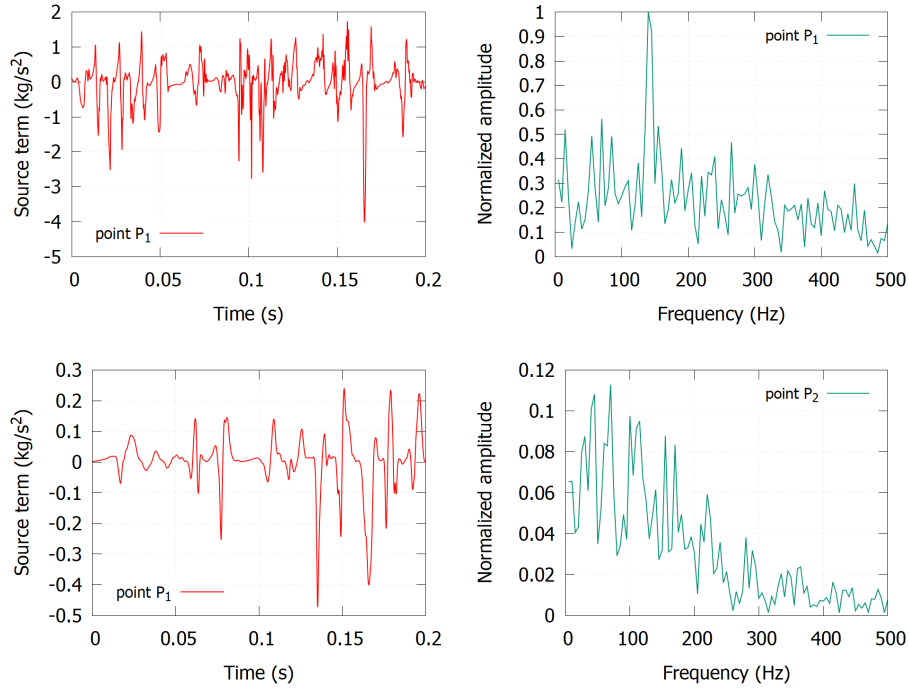


Figure 7: Acoustic source values in time and frequency domain at point P_1 and point P_2 in the central region along the x -axis and outside of the central region

The acoustic propagation is computed in the complete domain $\Omega_1 \cup \Omega_2$. The chosen time step for the acoustic computation corresponds to about 30 steps per period at the main frequency. The expected main frequency component is 70 Hz, resulting in a time step of approximately 0.5 ms. Contour plots of the acoustic pressure in the near-field region and the entire domain are presented in Fig. 8. In the near-field region Fig. 8a displays two separate time frames at $t = 0.115$ s and at $t = 0.137$ s, which highlight the induced sound from the upper and lower main vortices. Fig. 8b presents the acoustic pressure radiation in the entire domain at time $t = 0.115$ s. Lastly, Fig. 9 depicts acoustic pressure values at a point P_3 located at (0, 5) m, directly above the cylinder. To apply the Fourier transform, moving average is calculated with a window of 4 time steps and subtracted from the obtained acoustic pressure values. Frequency spectrum in Fig. 9 reveals a dominant frequency component of 136.7 Hz, accompanied by a secondary frequency of 278.5 Hz. Although this does not seem to correspond to the expected frequency of vortex-shedding at 70 Hz, it does resemble certain similarity to the fluid domain, in which the second highest frequency of the observed fluctuation was twice as high in the central region along the x -axis.

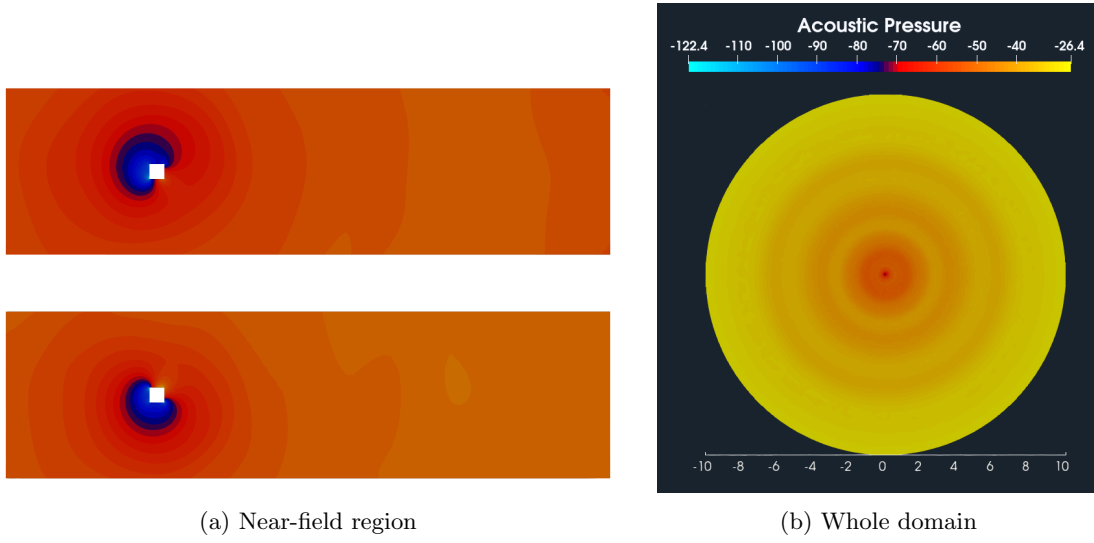


Figure 8: Acoustic pressure distribution at $t = 0.115$ s and $t = 0.137$ s

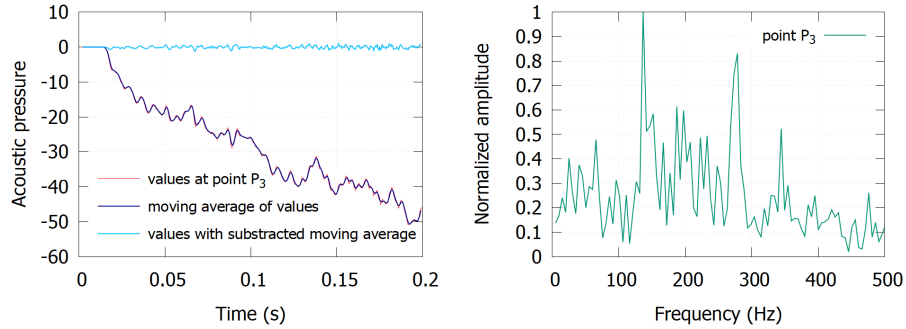


Figure 9: Acoustic pressure results in time and frequency domain at point P_3 , located directly above the cylinder

5 Conclusion

In this study, we have introduced a computational aeroacoustic methodology that establishes a foundation for future investigations. To address the fluid domain, we utilized the open-source software OpenFOAM, employing the FVM solver for solving the incompressible Navier-Stokes equations. Subsequently, the acoustic investigation involved solving the inhomogeneous wave equation based on Lighthill's acoustic analogy, utilizing an in-house FEM solver. Our primary focus was on

a 2D cylinder situated in a fluid field with a Reynolds number of about 13 300. The presence of the cylinder resulted in the formation of strong vortices in the downstream region, modeled using the $k-\omega$ SST DES turbulence model. These vortices induced acoustic waves, which were explored and analyzed in both the near-field and far-field regions. To verificate the numerical implementation, simulation results have been compared against the experimental measurements.

Acknowledgment

This research work was supported by the *Czech Technical University in Prague* under the Grant No.SGS22/148/OHK2/3T/12.

References

- [1] Norberg, C.: Flow around a circular cylinder: aspects of fluctuating lift. *Journal of Fluids and Structures*, vol. 15, no. 3: (2001) pp. 459–469
- [2] Bearman, P.: Bluff Body Flow Research with Application to Road Vehicles. *The Aerodynamics of Heavy Vehicles II: Trucks, Buses, and Trains*: (2009) pp. 3–13
- [3] Li, Z. & Li, J.: Numerical Simulation Study of Aerodynamic Noise in High-Rise Buildings. *Applied Sciences*, vol. 12, no. 19: (2022)
- [4] Blazek, J.: Chapter 2 - Governing Equations. *Computational Fluid Dynamics: Principles and Applications (Second Edition)*: (2005) pp. 5–28
- [5] Ferziger, J. & Perić, M.: Finite Volume Methods. *Computational Methods for Fluid Dynamics*: (2002) pp. 71–89
- [6] Greenshields, J.: OpenFOAM v8 User Guide. *The OpenFOAM Foundation*: (2020)
- [7] Ferziger, J. & Perić, M.: Solution of the Navier-Stokes Equations. *Computational Methods for Fluid Dynamics*: (2002) pp. 157–216
- [8] Spalart, P. & Allmaras, S.: Comments on the Feasibility of LES for Wings, and on a Hybrid RANS/LES Approach.: (1997)
- [9] Strelets, M.: Detached eddy simulation of massively separated flows. *39th Aerospace Sciences Meeting and Exhibit*: (2001)
- [10] Menter, F., Kuntz, M. & Langtry, R.: Ten years of industrial experience with the SST turbulence model. *Heat and Mass Transfer*, vol. 4: (2003)
- [11] Lighthill, M. J.: On Sound Generated Aerodynamically. I. General Theory. *Proceedings of the Royal Society of London. Series A, Mathematical and Physical Sciences*, vol. 211, no. 1107: (1952) pp. 564–587
- [12] Goldstein, M. E.: Aeroacoustics.: (1976)
- [13] Caro, S., Ploumhans, P., & Gallez, X.: Implementation of Lighthill’s Acoustic Analogy in a Finite/Infinite Elements Framework. *10th AIAA/CEAS Aeroacoustics Conference*: (2012)
- [14] Hughes, T.J.R.: The Finite Element Method: Linear Static and Dynamic Finite Element Analysis.: (2000)
- [15] Escobar, M., Ali, I., Hahn, C. & Kaltenbacher M.: Numerical and Experimental Investigation on Flow Induced Noise from a Square Cylinder. *10th AIAA/CEAS Aeroacoustics Conference*: (2004)
- [16] Durao, D. F. G., Heitor, M. V. & Pereira, J. C. F.: Measurements of turbulent and periodic flows around a square cross-section cylinder. *Experiments in Fluid*, vol. 6: (1988) pp. 298–304