Accurate simulations of surface pressure fluctuations and flow-induced noise near bluff body at low mach numbers

Y. P. Wang a, J. Chen b, H. C. Lee b K. M. Li b

aSchool of Automotive Engineering, Wuhan University of Technology, Wuhan, Hubei, China
bSchool of Mechanical Engineering, Purdue University, West Lafayette, IN, USA

ABSTRACT: For low Mach number flow around a bluff body, the far-field sound pressure is mainly dominated by dipole sources. The dipole sources are induced by the unsteady surface pressure distributions due to the presence of a turbulent boundary layer in the vicinity of the solid body. The far-field sound pressure can be calculated by integrating the time derivative of the wall-pressure fluctuations at the surface of the rigid body. Recent development in computational fluid dynamics (CFD) brings a powerful tool for predicting unsteady turbulent flow fields and the generation mechanism of aerodynamic noise. The present study investigates the use of an improved CFD method to accurately simulate the flow fields near a solid body. In particular, a large eddy simulation technique is adopted to predict the unsteady surface pressure fluctuation. The influences of various subgrid-scale models and near-wall treatments are investigated. The information of the flow field is then used to calculate the far field sound pressures. Extensive computations have been conducted to calculate the noise induced by the flow over a circular cylinder and an automobile's rear view mirror. The calculated sound pressure levels have been shown to agree well with published experimental results.

KEYWORDS: Low Mach Number Flow, Dipole, CFD, Aerodynamic Noise, Subgrid-scale model, Near-wall Treatment.

1 INTRODUCTION

For low Mach number flow around a solid body, the far-field sound pressure is dominated by dipole sources and quadrupole components play little role in the total sound field [1][2]. The dipole sources are created by unsteady surface pressure distributions due to the presence of a turbulent boundary layer in the vicinity of the solid body. The far-field sound pressure can be calculated by integrating the time derivative of wall-pressure fluctuations at the surface of the rigid body [3][4]. Recent developments in the field of computational fluid dynamics (CFD) have provided a powerful tool for predicting unsteady turbulent flow fields and the generation mechanism of aerodynamic noise. The direct numerical simulation (DNS) approach [5] solves the Navier–Stokes equation “exactly” in the temporal-spatial domain (resolving to the smallest scale). To avoid the extremely high computational cost associated with DNS, temporal development of averaged field parameters can be assessed using unsteady Reynolds-averaged Navier–Stokes (U-RANS) equations. Appropriate turbulence closure models are then required to close these equations. Different turbulence closure models, ranging from simple algebraic models to more complicated ones involving multiple equations [6], are applied to simulate flow-induced aerodynamic noise [7]-[9].

An alternate approach is to apply the large eddy simulation (LES) methodology [10] where large energy-containing scales of the flow fields are computed numerically and flow parameters
in unresolved subgrid scales are modeled by a subgrid-scale (SGS) model. Among the various proposed SGS schemes[10]-[12], the most popular model is the eddy-viscosity type developed by Smagorinsky [13], where the model constant can either have a static value or be determined dynamically (see detailed discussion in section 2). LES generally provides better accuracy than U-RANS for flows that feature unsteady components and large-scale eddies. When LES is used to simulate a flow field for analyzing flow-induced noise, it is very important to select an appropriate SGS model. This is because mid-to-high frequency noise is generated by turbulent structures of a very fine resolution at the subgrid scale, and the energy contained in the SGS eddies may be comparable to the acoustic energy.

The performance of SGS models can usually be appraised by a priori or a posteriori tests, as suggested by Piomelli et al. [14]. The validity of static and dynamic Smagorinsky-type SGS models for LES simulations has been well established using either numerical simulations or experimental results on homogenous isotropic turbulence [10][12]. LES simulations show that the dynamic Smagorinsky model gives better prediction results as compared to the static Smagorinsky model [15]. However, a priori tests on SGS models of a homogenous but less complicated, time-evolving, anisotropic, turbulent flow field show that the dynamic Smagorinsky model, as well as several other advanced SGS models, do not always yield numerical results that are better than those predicted by the static Smagorinsky model [16]. It is therefore important to examine the performance of a number of SGS models and establish guidelines for their use in studying flow-induced noise at low Mach numbers.

The current study is also motivated by the need to reduce the cabin noise level in automobiles operating at cruising speeds. The aerodynamic noise inside the cabin increases considerably with the vehicle’s speed. This type of flow-induced noise has broadband spectral content ranging from tens of Hertz at low frequencies to a few hundreds or thousands of Hertz at mid-to-high frequencies. This is attributed to time-varying flow separations and the breaking of large vortical structures into fine turbulent structures. The low frequency noise is generated by periodic convection of large-scale vortices over the open cavities where an unsteady shear flow exists, e.g., buffeting noise due to an open sunroof or window. The high frequency noise is induced by the shedding of unsteady vortices due to the interaction between body details, e.g., the flow interaction between the outside rear-view mirror and the A-pillar. This type of aerodynamic noise is a major contributor to the total noise levels inside the passenger cabins of vehicles. It can cause passenger discomfort, and more importantly, it may lead to driver fatigue. Therefore, it is crucial for the success of automobile development to predict the aerodynamic noise at the design stage and control it within an acceptable level. In the past few years, some researchers have incorporated the boundary element formulation in Curle’s analogy [17]–[19] to predict aerodynamic noise for a non-compact solid body interacting with flow at low Mach numbers. However, we adopt a different route by examining the accuracy of the flow fields to address the weakness of the traditional approach at high Helmholtz numbers. We note that the first requirement for predicting the sound fields accurately is to develop an effective and precise numerical scheme to simulate the unsteady flow fields.

This paper aims to assess the effectiveness of combinations of SGS models and near-wall treatments in predicting aerodynamic noise. We conduct LES simulations on 2D and 3D canonical problems. The simulations are configured in such a way that the results can be directly compared with well-documented acoustic measurements. This evaluation is expected to aid the selection of effective simulation tools at the design stage for predicting flow-induced noise. The paper is organized as follows. A brief review of LES is given in section 2. Following that, the simulation results of flow over a 2D cylinder and 3D generic side-view mirror model are discussed in sections 3 and 4; comparisons of the experimental data are also presented therein. Conclusions and future prospects are presented in the final section.
2. UNSTEADY FLOW SIMULATION BY LES FOR PREDICTING FLOW-INDUCED NOISE—A REVIEW

In LES, a filtering operation is applied initially to field variable \( g(x, t) \) to obtain its resolved component \( \tilde{g}(x, t) \) in the spatial domain \( D \):

\[
\tilde{g}(x, t) = \int_D g(x - x', t) G_\Delta(x') \, dx'
\]

(1)

where \( x = (x_1, x_2, x_3) \) denotes the coordinates, \( t \) is time, and \( G_\Delta(x) \) is the filtering kernel with a characteristic length scale \( \Delta \). The representative flow variable \( g(x, t) \) can be either the flow velocity \( u_i(x, t) \) or the instantaneous pressure \( p(x, t) \). For low Mach number (incompressible) flow, \( \tilde{u}_i(x, t) \) and \( \tilde{p}(x, t) \) can be solved using the filtered incompressible Navier–Stokes equations [20]:

\[
\begin{align*}
\frac{\partial \tilde{u}_i}{\partial x_i} &= 0 \\
\frac{\partial \tilde{u}_i}{\partial t} + \tilde{u}_j \frac{\partial \tilde{u}_i}{\partial x_j} &= -\frac{1}{\rho} \frac{\partial \tilde{p}}{\partial x_i} + \nu \frac{\partial^2 \tilde{u}_i}{\partial x_j \partial x_j} - \frac{1}{\rho} \frac{\partial \tau_{ij}}{\partial x_i} 
\end{align*}
\]

(2)

where \( \rho \) is the fluid density and \( \tau_{ij} \) is the subgrid-scale stress given by

\[
\tau_{ij} = 2\nu_r \tilde{S}_{ij}
\]

(3)

and the subscripts \( i, j = 1, 2, 3 \) (free indexes of index notation) represent the respective components along the coordinate axes \( x_1, x_2, \) and \( x_3 \). Once the SGS stress given in equation (3) is modeled properly, e.g., using the resolved parameters \( \tilde{u}_i(x, t) \) and \( \tilde{p}(x, t) \), it can be solved using equation (2). Then, the instantaneous flow fields are approximated by their resolved counterparts, namely,

\[
\begin{align*}
\tilde{u}_i &\approx \tilde{u}_i(x, t) \\
p &\approx \tilde{p}(x, t)
\end{align*}
\]

(4a) and (4b)

The predominant influence of the SGS stress on the resolved flow field parameters is reflected in the transport of kinetic energy from large resolved scales to the small unresolved subgrid scales [20],[21]. As a result, accurate modeling of the SGS stress is crucial for the success of LES in predicting the flow fields. Among the various SGS models that have been proposed [10]-[12], the most popular model is the eddy-viscosity type wherein the deviatoric part of the SGS stress is given by

\[
\tau_{ij} = \frac{1}{3} \tau_{ik} \delta_{ij} = -2\nu_r \tilde{S}_{ij}
\]

(5a)

where, \( \delta_{ij} \) is the Kronecker delta function, \( \nu_r \) is the subgrid-scale eddy viscosity, and the resolved strain is given by

\[
\tilde{S}_{ij} = \frac{1}{2} \left( \frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right)
\]

(5b)

where the repeated subscript \( k \) in the tensor represents the sum of the respective components from 1 to 3 (summation index of index notation).

The Smagorinsky model [13] is typically used for this class of approximations for the SGS stress, in which the eddy viscosity is

\[
\nu_r = \left( C_S \Delta \right)^2 \tilde{S}
\]

(6a)

where \( C_S \) is the Smagorinsky coefficient, and the magnitude of the resolved strain, \( \tilde{S} \), is calculated as

\[
\tilde{S} = \sqrt{2 \tilde{S}_{ij} \tilde{S}_{ij}}
\]

(6b)

In the traditional (static) Smagorinsky–Lilly model, \( C_S \) usually takes a static value of 0.16 in...
many practical situations simulated using scale invariant models [22]. However, the value of this Smagorinsky coefficient is questionable in highly complex flow problems [12]. A dynamic approach is proposed to determine the local value of $C_S$ by applying a second filtering operation at a scale of $\alpha \Delta$ where $\alpha$ is usually taken as 2 [23],[24]. This approach is referred to as the (standard) dynamic Smagorinsky model. Other approaches to model $C_S$ and $\tau_g$ have been proposed [10]-[12]. Some of these advanced models have been incorporated into commercial CFD programs. Users may choose a specific SGS model to simulate different flow problems.

Once the unsteady flow field is determined from LES, the Ffowcs Williams–Hawkings (FW-H) acoustic analogy [25] can be applied to compute the far-field sound pressure for flow over the rigid body, where the dipole term is dominant over the monopole and quadrupole terms. Then, the FW-H equation can be simplified as follows [26]:

$$p'(x,t) = \frac{1}{4\pi c_0} \int \frac{(x_i - y_i)n_i \partial p(y,\tau)}{r^2} \, dS(y) + \frac{1}{4\pi} \int \frac{(x_i - y_i)n_i p(y,\tau)}{r^3} \, dS(y) \quad (7)$$

where $c_0$ is the speed of sound in air, $\tau$ is the emission time ($\tau = t - r / c_0$), $r$ is the distance between the source and the receiver, and $y$ is the source on the surface of the rigid body $S$.

An accurate prediction of surface pressures in an unsteady flow field is crucial for the successful prediction of the far-field aerodynamic noise [15]. Flow fields in the viscous near-wall regime must be treated properly because of the distribution of dipole-type sources located on rigid surfaces. The presence of small but dynamically important eddies in the near-wall region demands that LES meets a number of challenging requirements. The computational grid must be sufficiently fine to resolve at least 80% of the energy-containing flow eddies. In addition, grid cells of high aspect ratio in the near-wall regime should be avoided [27]. Estimation suggests that an extremely fine grid is needed to accurately predict the surface pressure fluctuation, and the number of required grid points is proportional to the second power of the flow Reynolds number, which is comparable to the grid requirement in DNS [28]. It is evident that LES becomes increasingly impractical for fully resolving the near-wall regime in high Reynolds number flows.

To extend the applicability of LES at high Reynolds number flows, the dynamic effects of a semi-viscous near-wall layer can be estimated by means of a simple model that leads to an approximate boundary condition for LES at the outer regions; thus, the need to construct very fine grid points near the wall surface is avoided [29]-[31]. Among the various options proposed, the “wall-function” (WF) method uses an assumed instantaneous velocity profile between the nodes closest to the wall and the wall surface. A commonly used velocity profile is the Werner–Wengle wall function [32], which assumes a 1/7th power law outside the viscous sublayer interfaced with the linear profile within the viscous sublayer,

$$u^+ = \begin{cases} y^+ & \text{if } y^+ < 11.8 \\ 8.3(y^+)^{1/7} & \text{if } y^+ > 11.8 \end{cases} \quad (8a)$$

Here, $u^+=u / u_\infty$, and $u$ is the resolved velocity tangential to the wall at the nearest point. The non-dimensionalized distance $y^+$ measured from the wall is given by

$$y^+ = u_n y / v \quad (8b)$$

where $u_n$ is the friction velocity, $y$ is the normal distance measured from the cell centers, and $v$ is the kinematic viscosity of the fluid. The tangential velocity can be related to the wall shear stress by integrating the velocity profile given in equation (8a) over the distance between the first cell and the wall surface. A detached eddy simulation (DES), which matches U-RANS simulation for the near-wall layer and is equivalent to LES in the outer region, is also frequently used. The LES-WF combination and the DES method have been used to study unsteady separated flows in channels and over cylinders at high Reynolds numbers [29]-[31],[33].
3 FLOW OVER CIRCULAR CYLINDER

Flow over a 2D circular cylinder is a canonical problem for studying the unsteady effect of flow over a solid body and the induced aerodynamic noise. We simulate air flow with a free stream velocity \( U_0 = 69.2 \text{ m s}^{-1} \) over a cylinder with diameter \( D = 19 \text{ mm} \); Reynolds number \( Re = U_0 D / \nu \approx 90,000 \). These parameters are set such that they are identical to the values in the experimental study conducted by Revell et al. [34]. The computational domain extends from \(-8.5D\) (upstream) to \(+20.5D\) (downstream) with the origin set at the center of the cylinder. As suggested by Kim [35],[36], the distance between the top and the bottom of the computational domain is \( 21.0D \). An O-mesh topology is applied around the cylinder for better quality mapping of computational results. Thus, a fine mesh of \( 94,651 \) quadrilateral cells are employed, with a total of \( 360 \) mesh nodes in the circumferential direction and \( 80 \) mesh nodes in the radial direction. Numerical accuracy is checked by having a finer mesh consisting of \( 162,543 \) quadrilateral cells and a coarser grid consisting of \( 47,449 \) quadrilateral cells. Surface pressure and skin friction coefficient of the circular cylinder on different computational grids are compared in figure 1 and figure 2. As shown in figure 1 and 2, the fine and finer grid results are in good agreement, proving that the resolution is sufficient for an LES [47]. An additional comparison of the results between different meshes is presented in Table 1, where drag coefficients are compared. The difference in drag coefficient between fine and finer grids is approximately 0.4\%. Thus, fine computational grid is used for the entire analysis.

![Figure 1](image1.png)
**Figure 1.** Distribution of average coefficient of pressure \( C_p \) over surface of circular cylinder (\( \theta = 0^\circ \) is leading edge and \( \theta = 180^\circ \) is trailing edge). Square: coarse grids; Triangle: fine grids; Circle: finer grids.

![Figure 2](image2.png)
**Figure 2.** Distribution of scaled skin-friction coefficient \( Cr\sqrt{Re} / 2 \) (where \( Cr=2\tau_w / (\rho U_0^2) \) and \( \tau_w \) is local wall shear stress) over circular cylinder. Square: coarse grids; Triangle: fine grids; Circle: finer grids.

<table>
<thead>
<tr>
<th>Number of cells</th>
<th>LES SLM+WF ( \hat{C}_d )</th>
</tr>
</thead>
<tbody>
<tr>
<td>Coarse</td>
<td>47,449</td>
</tr>
<tr>
<td>Fine</td>
<td>94,651</td>
</tr>
<tr>
<td>Finer</td>
<td>162,543</td>
</tr>
</tbody>
</table>

**Table 1.** Comparisons of drag coefficient on different computational grids.

In this set of numerical simulations, the boundary conditions are (i) no-slip and no-penetration conditions on the cylinder surface, (ii) a uniform free stream velocity with no perturbations at the inlet boundary, (iii) a constant free stream pressure on the outlet boundary, and (iv) free-slip conditions on the upper and lower boundaries of the computational grids. To predict the sound pressure, an implicit pressure-based finite volume method is used to solve the incompressible Navier–Stokes equations (equation 2) on the unstructured grids. The numerical solver in Fluent...
CFD software is used for the present study. The discretized transport equations are advanced in time using the non-iterative time-advancement (NITA) algorithm. In addition, the fractional-step method (FSM) is chosen so that the momentum equations are decoupled from the continuity equation. To avoid numerical diffusion, the convection terms in all transport equations are discretized using bounded central-differencing schemes (BCD). Thus, this methodology preserves the non-dissipative properties of the fluid flow and avoids possible unbounded solutions and non-physical oscillations caused by other central-differencing schemes. Furthermore, the diffusion terms are discretized using a central-differencing scheme of second-order accuracy. The unsteady pressure is interpolated using PRESTO (PREssure STaggering Option), available in Fluent.

In LES the timescale of the smallest resolved eddies determines the time step needed. According to Kim and Mohan [35], the smallest resolved eddy, \( \ell \), should be set approximately at \( \ell = 0.05D \) and the corresponding characteristic velocity, \( u' \), at 0.2\( U_0 \). A non-dimensional time step, \( U_0 \Delta t / D = 0.0036 \), is used to ensure that the maximum local Courant-Friedriches-Lewy (CFL) number does not exceed 1.5 in most grid points of the computational domain. Although using this time resolution to predict sound pressure is a second-order approach, it has been demonstrated that the numerical solutions are as accurate as those directly computed by a sixth-order compressible flow solver in the frequency range in which the numerical method accurately resolves the flow structures [15].

To obtain converged results, it is essential that the CFL criterion is met. In fact, the CFL number reflects the portion of a cell that a fluid traverses by advection in one time step. The Courant number should preferably be chosen to be less than 1.5 in order to reduce oscillations and numerical dispersion and improve accuracy. In general, each cell in the computations has a different CFL number. A summary of statistical data used in our numerical simulations shows that over 95% of cells have CFL numbers that are less than 1.5.

In the present study, we investigate the importance of simulation methods in efficiently and accurately predicting the aerodynamic noise in low Mach number flows. In particular, a variety of SGS models are used in LES and DES to obtain the time histories of flow fields over a circular cylinder. Predictions from the following models are compared: (i) Smagorinsky-Lilly model (SLM, equation (6) with \( C_S = 0.16 \)), (ii) wall-function equation (8) along with Smagorinsky-Lilly model (SLM+WF), (iii) dynamic kinetic energy subgrid-scale model (DKESM), (iv) wall-adapting local eddy-viscosity model (WALEM), (v) renormalization group model (RNGM) [37], (vi) dynamic Smagorinsky-Lilly model (DSLM) [37], and (vii) DES with Spalart-Allmaras RANS model and Smagorinsky-Lilly model [37].

The prediction results are evaluated by comparing them with the experimental data obtained by Cantwell and Coles [38]. As shown in figure 3, the pressure coefficients predicted by DKESM, WALEM, and RNGM are less than the measured data in most angular positions. On the other hand, DSLM performs better than DKESM, WALEM, and RNGM. DSLM underpredicts as compared to the experimental data when \( \theta < 120^\circ \) but overpredicts when \( \theta > 120^\circ \). DES predictions are generally less accurate than those of LES when \( \theta \) is small, but DES yields improved results near the trailing edge when \( \theta > 110^\circ \).

Figure 3. Distribution of average coefficient of pressure \( C_p \) over surface of circular cylinder (\( \theta = 0^\circ \) is leading edge and \( \theta = 180^\circ \) is trailing edge). Solid triangle: experimental data [Ref. 38]; solid line: SLM+WF; cross: SLM; square: DKESM; circle: WALEM; asterisk: RNGM; diamond: DSLM; dashed line: DES.
The remaining two models, SLM and SLM+WF, both give excellent predictions $C_p$, although the predictions of SLM+WF are marginally better than those of SLM. However, it is interesting to note that SLM alone does not yield better predictions for the coefficient of mean skin friction (figure 4). Thus, SLM+WF out of all the methods tested is the best numerical model for accurately predicting the coefficient of skin friction. Both DKESM and WALEM agree closely with the experimental data. The prediction of RNGM is of the same order of accuracy as that of SLM. DSLM performs marginally better than RNGM and SLM and generally underpredicts the coefficient of mean skin friction at the leading edge.

![Figure 4. Distribution of scaled skin-friction coefficient over circular cylinder. Solid triangle: experimental data [Ref. 39]; solid line: SLM+WF; cross: SLM; square: DKESM; circle: WALEM; asterisk: RNGM; diamond: DSLM; dashed line: DES.](image)

The Strouhal number ($St = fD/U_0$) for the shedding frequency of vortices, mean and r.m.s. drag coefficients ($\bar{C}_d$ and $\sqrt{C_d^2}$), lift coefficients ($\sqrt{C_l^2}$), and angle of flow separation ($\theta_s$) are also used as benchmark parameters for evaluating the quality of the CFD results. The predicted values of these parameters are listed in table 2 for comparison with the experimental data. Except for the r.m.s. drag coefficient, the SLM+WF model agrees better with the measured data. All other models give comparable results for these parameters. The predictions according to LES with different subgrid models are consistently better than the DES predictions in all parameters except the Strouhal number. However, the computational cost for LES is generally higher than that for DES.

<table>
<thead>
<tr>
<th>Experimental Data</th>
<th>$St$</th>
<th>$\bar{C}_d$</th>
<th>$\sqrt{C_d^2}$</th>
<th>$\sqrt{C_l^2}$</th>
<th>$\theta_s$</th>
</tr>
</thead>
<tbody>
<tr>
<td>SLM+WF</td>
<td>0.173</td>
<td>1.26</td>
<td>0.26</td>
<td>0.55</td>
<td>80°</td>
</tr>
<tr>
<td>SLM</td>
<td>0.230</td>
<td>1.06</td>
<td>0.25</td>
<td>0.52</td>
<td>82°</td>
</tr>
<tr>
<td>DKESM</td>
<td>0.126</td>
<td>0.96</td>
<td>0.25</td>
<td>0.47</td>
<td>83°</td>
</tr>
<tr>
<td>DSLM</td>
<td>0.226</td>
<td>0.86</td>
<td>0.25</td>
<td>0.46</td>
<td>84°</td>
</tr>
<tr>
<td>WALEM</td>
<td>0.235</td>
<td>0.85</td>
<td>0.24</td>
<td>0.39</td>
<td>84°</td>
</tr>
<tr>
<td>RNGM</td>
<td>0.230</td>
<td>0.78</td>
<td>0.21</td>
<td>0.31</td>
<td>103°</td>
</tr>
<tr>
<td>DES</td>
<td>0.205</td>
<td>0.8</td>
<td>0.21</td>
<td>0.35</td>
<td>99°</td>
</tr>
</tbody>
</table>

Table 2: Comparisons of simulation results with experimental data.

The comparisons indicate that SLM+WF shows better agreement between the predictions and the experimental data. The SLM differs from SLM+WF only in terms of the respective near-wall treatments. As shown in the computational results, the interaction between the flow and the solid surface greatly affects the accuracy of the numerical model. This result indicates the superior performance of the analytical integration of the power law in the velocity distribution for the near-wall treatment in dealing with separating flows.

Thus, in light of the comparisons described above, SLM+WF is suitable for performing LES to compute unsteady lift and drag forces on a circular cylinder as well as to predict the surface

---

The Seventh International Colloquium on Bluff Body Aerodynamics and Applications (BBAA7)
Shanghai, China; September 2-6, 2012
pressure and mean drag and the point of flow separation on the cylinder’s surface. Furthermore, by substituting pressure into equation (7), the flow-induced noise can be calculated for an observer located at the far field. Before we proceed, it is of interest to demonstrate the correlation between the flow fields and acoustic field by examining snapshots of flow fields. Figure 5(a) and (b) shows the instantaneous streamlines and pressure distribution in the vicinity of the circular cylinder, where a periodic flow pattern is identified. This repeated alternate shedding of vortices suggests that the frequency of the far-field sound pressure can be characterized by the Strouhal number of the flow (corresponding to a frequency of 625 Hz and period of 0.0016 s).

![Figure 5. Predictions of instantaneous flow field from LES simulations (SLM+WF). (a) Instantaneous streamlines; (b) instantaneous static pressure.](image)

To further support the above observation, time histories of the lift coefficient and the surface pressure at selected points on the cylinder’s surface are analyzed (results not shown here for brevity). It is noted that the magnitudes of the lift force and surface pressures vary sinusoidally with a dominant period of approximately 0.0016 s, which is consistent with the change in flow patterns observed in figure 5. However, the phase angles of the surface pressures at different locations differ because the surface pressures change both spatially and temporally. In figure 6, we plot the power spectral density of the lift coefficient of the cylinder versus Strouhal number, and a distinct peak is observed at $St = 0.173$. Figure 7 shows spectra of relative surface pressure levels, $L$, at three azimuthal locations of 10°, 90°, and 160°, where $L$ is defined as

$$L = 20 \log \frac{p}{p_{ref}}$$

with the reference pressure $p_{ref} = 2 \times 10^{-5}$ Pa. Figures 6 and 7 show the broadband characteristics of the lift force and surface pressure. These are largely due to the presence of a turbulence field in the flow around the cylinder.

![Figure 6. Lift coefficient of 2D cylinder versus Strouhal number.](image)
Next, we compare the predictions of noise induced by this flow over a 2D circular cylinder with experimental results. The data is from Revell et al. [34], where a microphone is used to measure a sound field 128$D$ away from the longitudinal axis at an angle $\theta = 90^\circ$ in an acoustic wind tunnel. The tests are conducted at a Mach number of 0.2 ($U_0 \approx 69.2$ m s$^{-1}$) and Reynolds number of 89,000. The flow simulations and acoustics analysis are repeated under identical conditions. The numerical results are presented in terms of the overall sound pressure level (OASPL):

$$OASPL = 10 \log \left[ \sum 10^{\frac{SPL_i}{10}} \right]$$

(10)

where $SPL_i$ represents the far-field sound pressure level at the receiver location for the $i$th frequency band.

The relative sound pressure level SPL is represented by equation (9) where $p$ is replaced with the far-field acoustic pressure in the time domain, which is computed via equation (7) using the time histories of the surface pressure predicted by CFD. The narrow-band acoustic pressures in the frequency domain can then be computed from the time-domain data by means of fast Fourier transform (FFT). Summing contributions from all frequency bands leads to the OASPL given in equation (9). Recall that the integration given in equation (7) is performed over a 3D surface. Hence, a further assumption is needed to compute the sound fields based on the 2D input data. To allow a reasonable comparison between numerical and experimental results, a source correction length (SCL) is introduced where the sound sources are assumed to be perfectly correlated within the SCL region but uncorrelated outside it. Based on the strategy employed by Cox et al. [42], the SCL region can be adjusted for evaluating its effect on the predicted OASPL. For a Reynolds number of 90,000, Norberg [40] suggests that the SCL can be set experimentally as 3.16$D$. Here, according to the conclusion of Orselli et al. [8], the SCL of 5.0$D$ provides the best agreement with the experimental results for 3D flow over a long cylinder of length 25.3$D$. The OASPLs obtained from the LES using SLM+WF and DES are 101.9 dB and 102.09 dB, respectively, compared to the experimental data 100 dB obtained by Revell et al. [34]. This further supports the notion that, in order to predict flow-induced noise at a far-field location, accurate predictions of the near-field unsteady flow fields are essential.
4 FLOW OVER CIRCULAR CYLINDER

In the previous section, the 2D simulation results suggest that LES with SLM+WF agrees better with the experimental data. In this section, this approach is extended to a 3D model problem. In particular, we consider the noise induced by flow over a generic side-view mirror mounted on a flat plate. The flow has a free stream velocity \( U_1 = 39 \text{ m s}^{-1} \) and a Reynolds number \( Re = 5.2 \times 10^5 \).

As shown in figure 8, the generic side-view mirror consists of a half-cylinder of diameter \( D = 0.2 \text{ m} \) and height \( H = 0.2 \text{ m} \). A quarter sphere is placed on top of a half-cylinder, making the total height \( 1.5D \). The mounting plate has a width of \( 8D \) and a length of \( 12D \). The side mirror is located centrally at \( 4.5D \) downstream of the plate’s leading edge.

The numerical solver is set similar to the 2D case described in section 2. The computational mesh is composed of a structured grid with a multi-block arrangement. A total of three different computational grids are created using ICEM-HEXA software [43]. The purposes of having three different computational grids (Coarse, Fine, and Finer) are to ensure that the results obtained are grid independent. There are 2.3 million of nodes in the coarse computational domain, 5.2 million of nodes in the fine computational domain and 7.2 million of nodes in the finer computational domain. Figure 9 shows the surface pressure coefficient at different sensor position for coarse, fine and finer meshes. Fine and finer mesh produce similar results, meaning that the results obtained is converged and is grid independent. Thus, fine mesh is be used throughout the entire analysis. The height and circumferential direction of the cylinder surface in the fine grid computational domain are discretized by 100 nodes, and the first off-wall node is located \( 2 \times 10^{-4} \text{ m} \) away from the solid surface along the normal direction, \( n \), of the local wall. A fine mesh is extended in the wake of the mirror for a distance of \( 10D \) as measured from the mirror’s rear face. The mesh is progressively coarsened from the vicinity of the mirror to the domain boundary, where the mesh size is set from 0.01 m to 0.1 m.

The boundary conditions are as follows: (i) no-slip and no-penetration conditions on the generic mirror and the flat plate, (ii) a uniform free stream velocity with no perturbations at the inlet boundary, (iii) a constant free stream pressure at the outlet boundary, and (iv) free-slip conditions at the top and two lateral sides of the computational domain. The flow fields and fluctuating pressures are computed by SLM+WF. To improve numerical efficiency, a steady RANS approach is first employed to initialize the transient computations. The pressure time histories are then calculated and recorded after an initial transient period \( \sim 90D/U_1 = 0.46 \text{s} \) with a time step...
of $5 \times 10^{-5}$ s for a period of 0.85 s, i.e., the recording time period is between $90D/U_1$ and $255D/U_1$. The time-domain pressures are then converted to frequency-domain pressures by standard FFT numerical routines. These results are then compared later with the published experimental data [44] and DES predictions [45].

For ease of comparison, figure 10 gives the locations of multiple pressure transducers for measuring surface pressures. The measured mean surface pressures (circles) from transducers 1 to 34 are presented in figure 11. The predictions of LES with SLM+WF and DES, also shown in figure 11, show reasonable agreement with the measurement results. The predicted mean pressure from LES and DES over the mirror front side (locations 10–25) shows good agreement with the measured results. Sensors at locations 1–9 and 30–34 are deployed downstream of the separation line where the flow has complex temporal-spatial characteristics, and thus, it is difficult to obtain ideal agreement with the experimental data. DES underpredicts the mean pressure at these locations, whereas LES (with SLM+WF) overpredicts. At locations 5 and 28 the underprediction from DES is more pronounced, which is attributed to a misrepresentation of the separation zone, as highlighted by Ask and Davidson [46]. LES with SLM+WF provides accurate predictions at these two locations.

![Figure 10](image)

Figure 10. Locations of pressure measurements on plate surface (A, B, C, and D) and mirror surface (1–34) of generic side-view mirror model: (a) top, (b) front, and (c) rear view.

![Figure 11](image)

Figure 11. Comparison of predicted and measured time-mean pressure coefficient. Circle: experimental; solid line: LES + wall function; cross: DES.

The breaking vortex behind the side mirror will induce air noise and may trigger vibration of the exterior structure. Thus, it is necessary to evaluate the capacity of a variety of simulation schemes to accurately predict surface pressure fluctuation. The simulation results are compared with the experimental data [43] and the numerical results from DES [45]. Again, the relative
pressure level, which is defined in equation (9), is used for the presentation of data. Figure 12 shows comparisons of experimental measurements with numerical predictions at four selected locations (A–D, figure 11). Predictions of the surface pressures according to SLM+WF agree well with the experimental data for all these locations. At the upstream location A, numerical predictions according to DES provide the worst results, with discrepancies in excess of 40 dB even at low frequencies. Numerical predictions according to SLM and DSLM are somewhat better than those of DES, but the errors are typically more than 25 dB at high frequencies.

Figure 12. Predicted and measured relative pressure levels [equation (9)] at various locations. Cross: SLM+WF; circle: SLM; solid line: DES; dashed line: DSLM; triangle: experimental [Ref. 44].

The propagation of sound is computed at four receiver locations, shown in figure 13. The far-field acoustic pressures at the same locations are measured by Rung et al. [45], and the comparison is shown in figure 14. It is evident that the LES with SLM+WF offers the best predictions at these four locations. DES significantly underpredicts SPL, particularly at high frequency range (>300 Hz), which may contribute to the underprediction of unsteady pressure level (thus dipole strength in equation (7)) at high frequency (figure 14). LES with DSLM overpredicts SPL at the same high frequency range, which further demonstrates the inadequacy of the dynamic Smagorinsky SGS model in simulating unsteady surface pressure fluctuation.

Figure 13. Locations of microphones (1–4) employed by Rung et al. [Ref. 45] for far-field acoustic measurements.
5 CONCLUSIONS

In this study, the implications of various SGS models and near-wall treatment schemes used in LES of low Mach number flows are investigated for the accurate prediction of unsteady flow fields and far-field acoustic fields. The unsteady LES predicts surface pressure fluctuations, which gives a distribution of the dominant dipole source. The far-field acoustic fields are assessed using the FW-H analogy. Benchmark studies are conducted on flow over a 2D cylinder and a generic side-view mirror model. Numerical predictions of various schemes are compared with the measured surface pressure and far-field acoustic data. It is identified that LES combined with the Smagorinsky–Lilly SGS model \((C_s = 0.16)\) and the Werner–Wengle wall function provides the best predictions in a wide frequency range \((10–1000 \text{ Hz})\). On the other hand, the DES approach (a combination of RANS in the near-wall regime and LES in the outer regime) considerably underpredicts the unsteady surface pressure and acoustic field at high frequencies \((>300 \text{ Hz})\). Advanced SGS models such as the dynamic Smagorinsky model show no superiority over the other models in predicting surface pressure fluctuations. These results aid the selection of appropriate CFD techniques for accurately simulating low Mach number flows. The technique identified in the present study is currently being applied to investigate the aerodynamic noise of automobiles operating at cruising speeds, including low frequency buffeting noise due to flow over open cavities and high frequency noise associated with the breaking of large-scale vortical structures.
6 ACKNOWLEDGEMENTS

The research was supported by “the Fundamental Research Funds for the Central Universities” (2012-JV-068) and China Scholarship Council of the National Natural Science Foundation of China (NSFC 50975083).

7 REFERENCES


