CWE-8
June 7-11, 2026
London, ON, Canada

CFD study on the aerodynamics of a square cylinder under time-varying accelerating flow conditions

Cheng-Yang Chung [a*], Cheng-Xuan Wu [b], Yuan-Lung Lo [b]

[a] *Aerospace Science and Technology Research Center, National Cheng Kung University, Tainan, Taiwan*
scychung@mail.ncku.edu.tw
[b] *Department of Civil Engineering, National Taipei University of Technology, Taipei, Taiwan*

SUMMARY

Unsteady accelerating flows exhibit strong time-varying characteristics, making traditional steady aerodynamic methods insufficient for accurate assessment. This study uses CFD to examine the aerodynamic response of a square cylinder in an accelerating flow, focusing on transient changes in aerodynamic forces and vortex shedding. The case considers a uniform, low-turbulence inflow accelerating from 4 m/s to 14 m/s. Time-varying lift and drag forces are obtained from instantaneous pressure integration to analyze how acceleration influences pressure fields and wake evolution. Additionally, the acceleration is initiated at various phases of the vortex-shedding cycle. Results show that the timing of acceleration causes only slight variations in vortex dynamics and transient loads, with no significant effect on overall lift and drag.

Keywords: *Non-stationary flow, Computational fluid dynamics, Accelerating flow field*

1. INTRODUCTION

Most previous wind tunnel studies have been conducted under low-turbulence, smooth-turbulence, or atmospheric boundary layer inflow conditions. However, with advancements in wind tunnel technology and numerical simulation capabilities, unsteady and accelerating flows—such as downbursts and tornadoes—have increasingly become important research topics. These flow types play critical roles in aviation, wind energy, and hydraulic engineering, yet their turbulence structures and energy transfer mechanisms remain insufficiently understood. To reproduce such unsteady conditions, specialized wind tunnels with active-flow control systems, such as WindEEE and multi-fan facilities, have been developed worldwide, enabling the generation of time-varying inflow profiles and leading to a growing body of aerodynamic studies [1–2]. Meanwhile, progress in computational fluid dynamics (CFD), particularly large-eddy simulation (LES) and direct numerical simulation (DNS), has allowed researchers to investigate the transient behavior of non-stationary flow fields beyond the limitations of conventional wind tunnels, thereby deepening our understanding of turbulence structures and energy transport.

2. NUMERICAL MODEL VALIDATION

In this study, a square cylinder is adopted as the primary body for simulation, and the computational domain configuration is shown in Figure 1. A three-dimensional large-eddy simulation (LES) is employed, with the cylinder width defined as D. The spanwise length is set to 2D and assigned a symmetric cyclic boundary condition. The top and bottom boundaries are specified as slip conditions, the cylinder surface is treated as a no-slip solid wall, and a zero-gradient condition is applied at the outflow boundary.

The present study employs the open-source software OpenFOAM to perform the flow-field simulations. To balance numerical accuracy and computational efficiency, a multi-layer, gradually varying mesh is designed. Finer grids are applied in regions near the cylinder where the flow is highly disturbed, while coarser grids are used farther away where flow variations are less significant. The total mesh count is approximately five million cells (Figure 2).

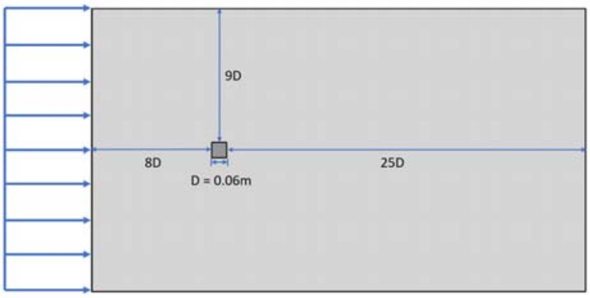

Figure 1: Computational domain.

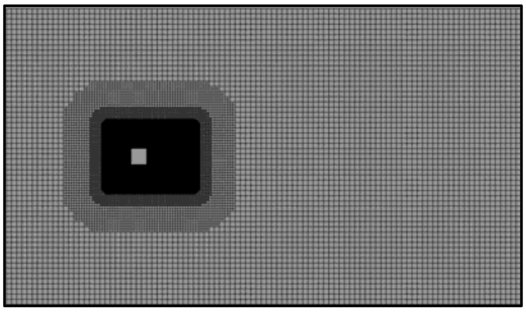

Figure 2: Numerical mesh system.

The validation case is conducted using a uniform inflow velocity of 12 m/s. The time step is set to 0.000025 s, and a total of 400,000 time steps are computed, corresponding to 10 seconds of real flow time. The simulation results show good agreement with experimental data in terms of the drag coefficient (Table 1). In addition, the predicted Strouhal number closely matches the values reported in the literature, indicating that the simulated vortex-shedding frequency is consistent with that observed in physical flows and successfully captures the alternate vortex formation in the wake of a bluff body. Furthermore, the pressure distribution statistics on the cylinder surface (Figure 3 and Figure 4) also demonstrate consistent trends between the numerical predictions and experimental measurements.

Table 1: Comparison of aerodynamic coefficients and Strouhal number

Case	Re	C_D	C_D’	C_L	C_L’	S_t
2022 Wu	47600	2.2	-	-	1.18	0.137
2015 Trias	22000	2.18	0.205	-	1.71	0.132
Presented (LES)	47600	2.31	0.209	0.002	1.475	0.135

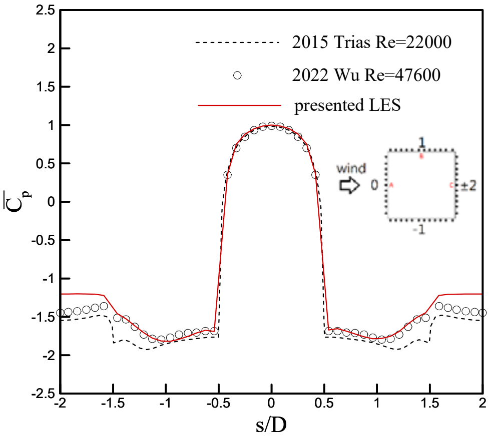

Figure 3: Mean surface pressure distribution on the cylinder for the validation case.

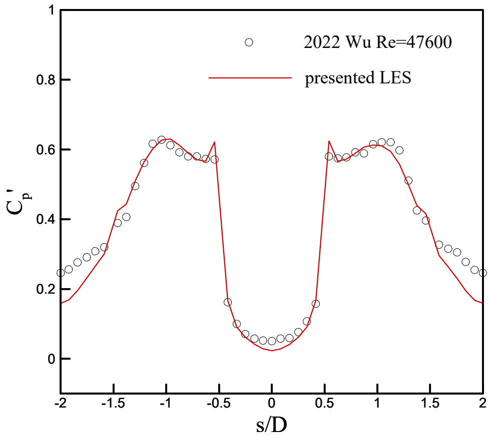

Figure 4: Fluctuating surface pressure distribution on the cylinder for the validation case.

3. ACCELERATING FLOW SIMULATION

A short-duration acceleration is simulated by increasing the inlet velocity from 4 to 14 m/s in 2s, with the internal flow maintaining spatio-temporal continuity governed by the Navier-Stokes equations. The acceleration is initiated at four characteristic phases within a single vortex-shedding cycle, corresponding to four simulations starting at intervals of one-quarter of the shedding period. Although the pressure coefficient exhibits a slight upward trend during acceleration, the magnitude of this increase is similar across all cases, resulting in no notable change in the fluctuation amplitudes of the lift and drag coefficients (Figure 5). The flow continues to exhibit alternate vortex shedding; however, the shedding frequency increases as the flow accelerates. Wavelet analysis of the lift coefficient (Figure 6) further confirms the rise in vortex-shedding frequency with increasing wind speed, while the Strouhal number under accelerating conditions remains close to that of the corresponding steady-flow results.

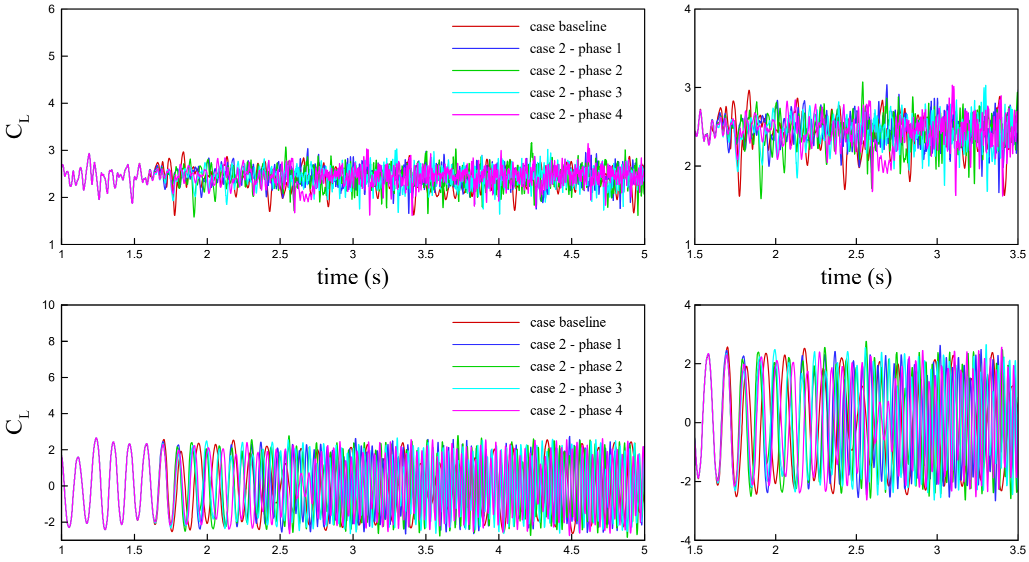

Figure 5: Time histories of drag and lift coefficients under accelerating-flow conditions.

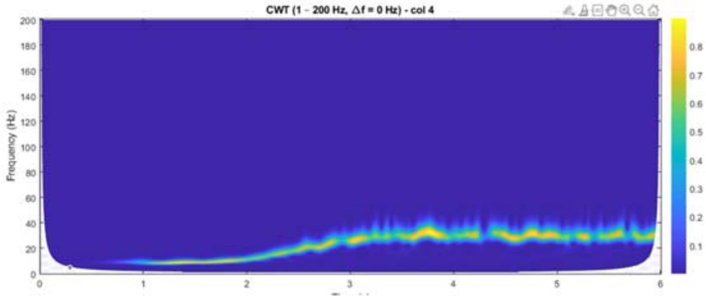

Figure 6: Wavelet transform results of the lift-coefficient time history.

The vorticity fields clearly illustrate the unsteady flow characteristics of a square cylinder subjected to accelerating inflow. As the velocity increases from 4 m/s to 14 m/s, the shear layers elongate and intensify due to the higher Reynolds number, resulting in stronger vorticity and delayed roll-up. During acceleration, the vortex formation length temporarily increases, indicating a stabilizing effect on the separated shear layers. The wake becomes progressively narrower and more turbulent at higher speeds, with vortex shedding occurring at a higher frequency, consistent with the near-constant Strouhal number behavior. Overall, the accelerating flow produces enhanced vorticity, increased shedding frequency, and modified wake development compared with steady-flow conditions.

Figure 7: Vorticity fields at different vortex shedding phases for various accelerating-flow process.

4. CONCLUSIONS

The simulations of four acceleration-initiation phases show that the aerodynamic responses exhibit a phase-shift behavior corresponding to the timing of acceleration, and no significant influence on transient statistical characteristics is observed. During the acceleration process, the pressure coefficients at various points on the cube surface display an increasing trend, returning to steady-state levels once acceleration ends. The vortex-shedding frequency increases as the flow accelerates, while the Strouhal number does not deviate noticeably from the steady-flow results.

ACKNOWLEDGEMENTS

The authors would like to thank to National Center for High-performance Computing (NCHC) for providing computational and storage resources.

REFERENCES

Lo, Y.-L., Mason, M., Li, Y., Zhang, Y., & You, Y.-C. 2024. Investigation of transient aerodynamic forces on a hemispherical dome under ramp-up flows. 9th International Colloquium on Bluff Body Aerodynamics & Applications

Brusco, V., Manes, C., Marusic, I., & Taylor, J. (2021). Transient aerodynamics of a square cylinder under downburst-like accelerating flows reproduced in a multiple-fan wind tunnel. Journal of Wind Engineering and Industrial Aerodynamics, 147, 263-275. https://doi.org/10.1016/j.jfluidstructs.2023.104038