

## ABSTRACT

Long-span cable-supported bridge decks are susceptible to aerodynamic instability due to their inherent flexibility, which necessitates preliminary evaluation of their aerodynamic performance in the early design phases. Compared to traditional wind tunnel testing, computational fluid dynamics(CFD) demonstrates its great potential on this task by providing valid results in a more timely and cost-saving fashion. However, CFD simulation is still making its way into design codes as an accepted quantitative evaluation tool due to the lack of a standardized approach in simulation development. Intending to study and explore this topic, the thesis focused on the theoretical basis of bridge aerodynamics and typical considerations of CFD application in this field. To further demonstrate the CFD application, 2D unsteady Reynolds-Averaged Navier-Stokes (RANS) simulations were run using turbulence model SST  $k - \omega$  (Menter, 1994) on a fixed bridge section. Nondimensional quantities (coefficients of drag and lift forces and pitching moment) of the bridge section at multiple angles of attack ( $\alpha = -10^\circ \sim +10^\circ$ ) were obtained and showed a good agreement with another CFD study and experimental results available in the literature.