

Wind Engineering Using Computational Fluid Dynamics Modelling

By: Dr Keith Liow (DreamCatcher Associate for CFD)

Wind engineering is a specialist engineering field which analyses the effects of natural wind in the built-environment. It covers a broad range of applications such as the impact of wind in and around built-structures. Wind engineering is important for the design of high-rise buildings, cable suspension bridges, towers, and chimneys. It is therefore incorporated into major design codes and standards for built-structures. Wind-load could be a significant load on high-rise buildings and wind-navigating around these structures may cause discomfort or even outright danger to the pedestrians, especially those with walking difficulties. At extreme wind-driven conditions such as tornadoes, hurricanes, or thunderstorms, structures can be susceptible to a dangerous level of static and dynamic loadings. At low to moderate wind speeds, studies of pollutant dispersion, pedestrian wind comfort, mechanical ventilation design, and wind farm design are some examples of wind engineering applications.



Figure 1 – Wind tunnel test facility at Clemson University

In the context of the design of buildings and structures, notable relevant engineering parameters are wind impact, pedestrian comfort, and dispersion transport. As these parameters are dependent on the aerodynamic forces exerted onto the building, thus a good appreciation of the complex airflow, which is caused by the interactions of the atmospheric effects (wind, terrain, etc.) with the buildings, is required. In this regard, the wind-tunnel has been the standard laboratory tool used by wind engineers. While the wind-tunnel has been the main tool of choice for many decades, computational fluid dynamics (CFD) is increasingly gaining traction as a complementary tool in wind engineering,

not least because of the rapid advance of computational power and the development of sophisticated numerical techniques.

In this case study, the drag-force caused by a uniform flow profile past a singular tall structure of a circular cross-section is being investigated. This is studied by using a combination of wind-tunnel and CFD techniques. Flow specifications corresponding to the isolated tall structure that is being subjected to sustained wind speeds in excess of 200 km/h. This example is perhaps indicative of offshore towers which can often be exposed to high cyclonic speeds.

The CFD models have to be designed to capture the flow gradients to ensure accurate resolution of the shear-and-wake effects. This can be implemented through a structured grid that is designed for cell clustering in the downstream region of the cylinder and yet, only incurs a reasonable computational cost (see Figure 2). A moderate grid stretching ratio should also be used to prevent artificial gradients. High-order discrete schemes are also required to dampen numerical diffusion. To reproduce the transient behaviour of the aerodynamic fluctuations, the more advanced *unsteady Reynolds-averaged Navier Stokes* (URANS) turbulence model of *kw-SST* was used in this particular study.

The flow visualisation shows the 'turning' of the flow and subsequent wake formation of the circular structure. The periodic shedding of the vortices as the flow separates past the cylinder can be observed, which implies a drag-force of a harmonic nature (see Figure 3). The mean drag, which ultimately is the engineering parameter of interest, is obtained through time-averaging the drag fluctuations over several shedding periods. The results compare favourably with those obtained from wind-tunnel testing. This validation exercise has provided yet another clear indication that the physical phenomena of an externally driven wind induced aerodynamic loading on a structure can indeed be satisfactorily addressed by CFD modelling.

In this conclusion, while it is true that CFD can potentially produce 'misleading/wrong' results if it is applied without due consideration to factors that are specific to wind-engineering problems, nevertheless continuing research and development (R&D) in expanding the realms of CFD from its traditional application areas (aerospace, automotive, etc.) towards wind engineering implies that CFD could have a strategic advantage as a complementary analysis tool.

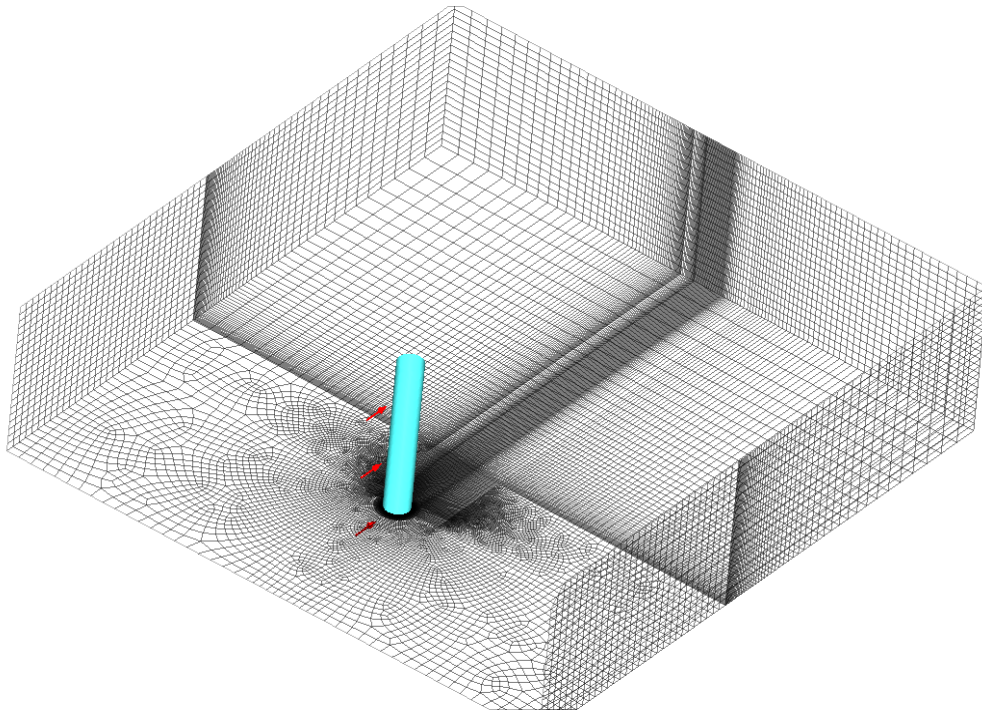


Figure 2 – Structured grid used for the model

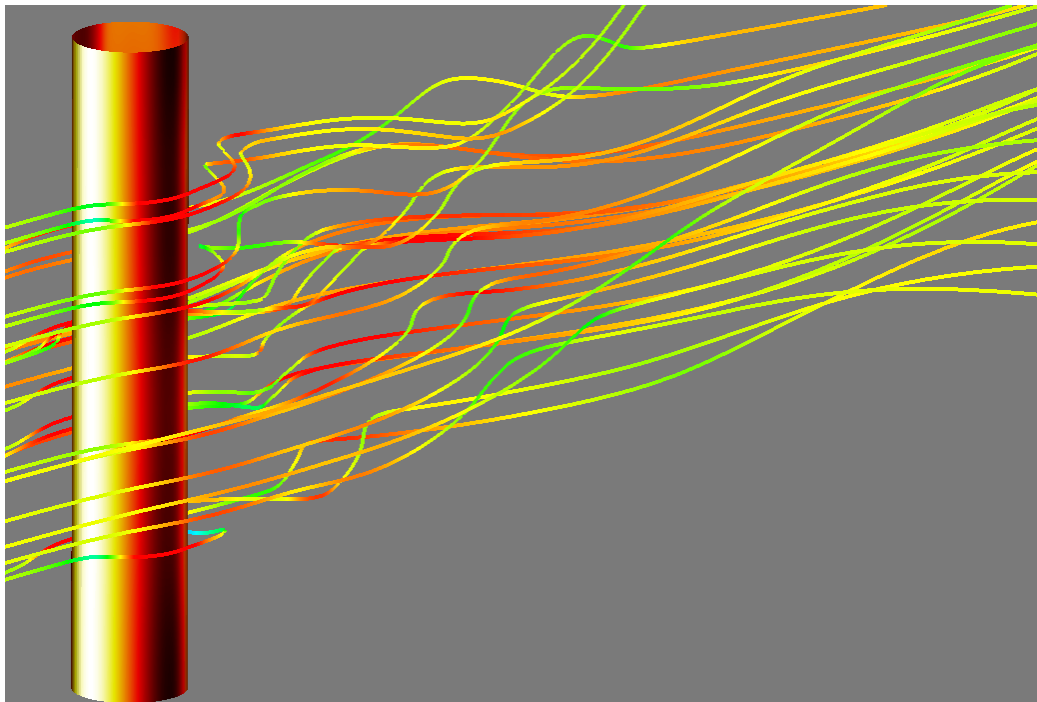


Figure 3 – Periodic shedding of vortices past the cylinder