APPENDIX 18.1
COMPUTATIONAL FLUID DYNAMICS (CFD) METHODOLOGY AND ANALYSIS

Appendix 18.1 CFD Analysis

The CFD analysis has the following main features:

- Simulation software: OpenFOAM v1706, is well recognised in industry and has been used by Arup in many engineering applications, including wind studies.
- Model scale: The simulation is nominally at 1:1 scale with sufficient detail and extent such that full dynamic similarity was achieved and problems associated with scaling avoided.
- Steady-state simulations: Reynolds-Averaged Navier-Stokes (RANS). The SimpleFoam steady-state solver for incompressible, turbulent flow was used for all of the CFD simulations presented.
- Initial conditions: The simulation domain was initialised using an inlet wind velocity profile derived from curve fitting to the wind profiles derived from the ESDU [3] analysis at each of the 16 wind directions.
- Turbulence: RANS simulations typically solve for the turbulent content of the flow by introducing new flow variables to represent turbulent kinetic energy and turbulence dissipation. This provides a statistically averaged representation of the turbulent flow. The SST turbulence model was used in all of the CFD simulations presented.
- Simulation metrics: The computational meshes used in the existing Site simulation had approximately 8.4 million cells, the Proposed Development simulation had 12.1 million cells

Boundary conditions:

- Inlet: The wind inlet wind velocity profile was derived from curve fitting to the wind profiles derived from the ESDU [3] analysis at each of the 16 wind directions.
- Outlet: An inlet/outlet boundary condition was applied, which switches between zero gradient when the fluid flows out of the domain and fixed value (no-slip) when fluid is flowing into the domain.
- Atmosphere: A zero gradient and slip boundary condition was applied at the top of the domain for pressure and velocity respectively.
- Walls: A no-slip boundary condition was applied at the solid boundaries. This is a standard condition that states that at a solid boundary, the air has no velocity relative to the boundary.

Exclusions and limitations

- The resolution of the detail in the models used in the CFD simulations is constrained by the mesh size and the turbulence models which were used. A balance between computational effort and accuracy was found.
- In the case of the present study which uses current best-practice, the limitations are unlikely to be significant. In critical cases further confirmation of significant wind conditions should be sought. The potential sheltering effect of the trees was not represented in the models. This implies that the CFD simulations represent a conservative scenario.