Appendix 10.4

CFD TECHNICAL ASSUMPTIONS

NOTE

Jackson's Row Developments Ltd (hereinafter referred to as ‘the Applicant’) is seeking to construct a major development proposal known as St Michael’s, Jackson’s Row, Manchester, UK

The planning application for the development was originally registered in January 2017. Following feedback, the decision was taken to revise the scheme. Revised plans and documents were submitted in November 2017 and the development was granted planning permission in June 2018.

Through the detailed design process, a number of relatively minor improvements to the original scheme have been identified; these will also provide technical and functional improvements through the next stage design process. The Applicant wishes to progress these proposed changes, which are considered to require amendments to the original scheme, through a Section 73 application.

An Environmental Statement, produced in accordance with the Town and Country Planning (Environmental Impact Assessment) (England and Wales) Regulations 2011, was submitted in support of the Approved Development in 2017. The 2017 Environmental Statement comprised three volumes, as follows:

Volume 1: Main Text and Figures;
Volume 2: Technical Appendices;
Volume 3: Non-Technical Summary

With reference to Volumes 1 and 2 of the 2017 Environmental Statement, it has been decided that rather than updating and re-submitting the entire Volumes in support the Section 73 application, an ‘Addendum’ to the 2017 Environmental Statement should be submitted. This is because the Proposed Amendments to the scheme are minor in nature and the potential to affect the environmental assessment is limited.

This Environmental Statement Volume 2 (2017) appendix formed part of the application package for the June 2018 approved scheme and remains valid.

CFD Technical Assumptions

CFD model
Computational Fluid Dynamics (CFD) solves the Navier Stokes equations governing fluid flow which is mapped out over a computational domain using the physics specific to the problem analysed; velocity, turbulence, pressure etc. The CFD code is solved via CD-adapco's STAR-CCM+ software, version 7.06. This program has been extensively used and widely validated and verified in many fluid driven applications, particularly in environmental wind cases involving atmospheric boundary layers such as this.
Using CFD to investigate the effects of a wind on a site enables a detailed understanding to be gained of the velocity distribution across the whole site. It also enables the local site microclimate to be compared with the annual weather data thus enabling assessments to be made of what activities could be comfortably undertaken at a particular location as a proportion of the year.

The fact that the flow is measured throughout the domain means that the cause of an elevated wind speed can be investigated and mitigation measures proposed that are suitable, for instance canopies or trees.

Boundary conditions
The CFD simulation has been set up in accordance with the COST Action 732 (2007)i detailing best practise guidelines for the CFD modelling of flows in the urban environment.

The CFD fluid domain has a minimum distance of 10 times the height of the tallest building from the extent of the modelled buildings to all inflow and outflow boundaries.
The top of the CFD fluid domain has been modelled as a no slip wall boundary condition with a minimum distance of 6 times the height of the tallest building from the ground.
The blockage ratio of the domain has been kept below 3% as detailed in Cost C14 (2004)ii.

Atmospheric Effects are accounted by the accurate modelling of the Atmospheric Boundary Layer profile as used in STAR-CCM+ by utilising the Zhang (2009)iii model of the expressions for turbulence and velocity.
Surface Roughness is accounted for in these equations by using the roughness parameter zo. This takes into account the effect of upstream buildings on velocity and turbulence formulation.

Meshing
A CFD model requires the fluid domain to be divided into discrete elements or "cells" that are linked together and communicate information between adjacent cells. The size, number and distribution of these cells define the level of accuracy that can be gained from the model.

In this model the domain has been meshed using hexahedral elements. The size of these hexahedral elements have been reduced around the buildings to ensure an adequate number of cells accurately resolve the geometry and flow field.
An inflation layer mesh has been used to increase the mesh density in areas of interest to ensure the flow field has been accurately resolved. This also ensures there is sufficient number of cells between the ground and the velocity slice plane.

Numerical Sub Models
Turbulence Modelling: Turbulent effects have been modelled using the Reynolds Averaged Navier Stokes (RANS) equations. The two equation k-ε RNG turbulence model was used as it is accepted to be accurate for swirling, turbulent flows such as the building immersed in an atmospheric boundary layer in this study.
Discretisation Scheme: Second order discretisation schemes have been used to spatially resolve flow and pressure distribution.

Convergence Criteria: Solutions to the Navier Stokes Equations are achieved using an iterative method to reduce the residual error within the solution, allowing closure of the governing equations for convective, diffusive and source terms. Solution convergence is a measure of the imbalance of the solution of each equation and relates to whether they have been sufficiently solved to an acceptable accuracy or not.

The residual error for this study is calculated using the RMS (root mean square) with a target of 0.0001 at selected locations around the buildings of interest.

The steady state CFD simulation will capture all major flow accelerations on the site that are likely to significantly impact pedestrian safety, it will not calculate velocity fluctuations expected in low speed turbulent wakes. Unsteady numerical modelling can account for this; however, it is computationally and time intensive to do so. Global site gusting is accounted for by running the simulation at higher range wind speeds at the site.

---

[i] COST Action 732, (2007) "*Best practice guideline for the CFD simulation of flows in the urban environment.*"
[ii] COST Action C14, (2004) *"Recommendations on the use of CFD in wind engineering."*
[iii] Xiaodong Zhang (2009) *"CFD simulation of neutral ABL flows"*