# An Overview of Arup Computational Fluid Dynamics Projects

Jennifer Keenahan[1], Cristina Paduano[1], Réamonn Mac Réamoinn[1]
[1]Arup, 50 Ringsend Road, Dublin 4, Ireland
email: jennifer.keenahan@arup.com, cristina.paduano@arup.com, reamonn.macreamoinn@arup.com

ABSTRACT: In recent years, end users have become more concerned with the human experience and the personal comfort of the individual is becoming more important in the design of the built environment. Computational Fluid Dynamics (CFD) is a tool that permits assessment of personal comfort. CFD is a stream of fluid mechanics that utilises numerical methods to analyse and solve problems involving fluid flows. The fundamental basis for these problems are the Navier-Stokes equations. While research applications of CFD are developing, as it the use of CFD in the aerospace industry, the use of CFD in Civil Engineering applications is currently at the cutting edge. Arup have recently developed a team to work on such projects and some of these will be presented in this paper. The paper will discuss the research question of each project, the methodology adopted in developing the models, the results of the simulations and some lessons learned going forward. Based on the projects done to date, it can be concluded that CFD has been shown to be a powerful tool that adds valuable information to fluid flow problems.

KEY WORDS: Computational Fluid Dynamics (CFD); Navier-Stokes; thermal comfort; pedestrian comfort.

## 1 INTRODUCTION

Computational Fluid Dynamics (CFD) is the use of computers and numerical methods to solve problems in fluid flow. It is a method for solving partial differential equations in continuum mechanics using numerical techniques. It involves breaking the problem down into a discrete number of volumes that can be analysed more easily [1]. Combining the solutions from these small volumes permits the generation of the complete solution. The equations governing fluid motion are based on the fundamental physical principles of the conservation of mass, momentum and energy. CFD has been successfully applied in many areas of fluid mechanics, including, heat and mass transfer [2], chemical reaction and combustion [3], aerodynamics of cars and aircrafts, and pumps and turbines. The use of CFD in civil engineering applications, although more recent, is currently at the cutting edge due to the direct practical applications. Applications of CFD to civil engineering include wind modelling and the dynamic response of structures [4], ventilation [5], fire, smoke flow and visibility [3], dispersion of pollutants and effluent [6], and heat transfer in buildings [7]. Traditionally, the interaction of these phenomena has been carried out experimentally, using scaled models and short calculations. However, it is rarely possible to accurately capture all phenomena guaranteeing repeatability. In the advent of improving computational power, and the development of numerical techniques such as Finite Element Analysis, CFD offers an opportunity to model many variations of the same problem at full scale, more efficiently, in a virtual environment.

As with any computer simulation, the quality of the results is dependent on the quality of the inputs, the assumptions, modelling characteristics employed and the equations used to represent the phenomena. There will inevitably be approximations, and a robust model validation process is essential. A high level understanding of the modelling process and of the phenomena being modelled is necessary for the output to be of any practical use.

In order to capture the phenomena in sufficient detail, large finite volume models are necessary requiring significant processing power. These large models generate a great amount of data, which in turn needs to be stored and analysed. The advantages of CFD however, far out-weigh the disadvantages. It is a non-intrusive, virtual modelling technique with powerful visualisation capabilities. Results can be captured across the entire domain. There are also significant cost and time savings with CFD as there is opportunity to assess comparisons between alternative systems quickly and efficiently, without the disruption of making physical changes on site.

This paper will discuss some projects recently undertaken by the new CFD team at Arup, namely, a Dublin Docklands tall building wind study, a Hospital thermal comfort study, and a Dundrum Ventilation Shaft fire study. The research question of each project will be discussed, the methodology adopted in developing the models presented, and finally the results illustrated.

## 2 CFD MODELLING METHODOLOGY

Modelling in CFD comprises three main stages: pre-processing, simulation and post-processing.

Pre-processing firstly involves the construction of the geometric model for the flow domain of interest, and the subsequent division of this domain into small control volumes (cells), a process often called 'meshing'. The flow field and the equations of motion are discretized, and the resulting system of algebraic equations is solved to give values at each node. Once the model and the mesh have been created, appropriate initial conditions and boundary conditions are then applied.

The Navier-Stokes equations, the governing equations for the behavior of fluid particles, are solved iteratively in each control volume within the computational domain until the solution converges. The field solutions of pressure, velocity, air temperature and other properties can be calculated for each control volume at cell centers and interpolated to outer points in order to render the flow field.

Post-processing involves graphing the results and viewing the predicted flow field in the CFD model at selected locations, surfaces, or planes of interest. The Navier-Stokes equations, used within the CFD analysis, apply a numerical representation to approximate the laws of physics to produce extremely accurate results, providing that the scenario modelled is representative of reality. In each of the following examples, Fire Dynamic Simulator Version 5.5.3 was used.

## 3 DUBLIN DOCKLANDS TALL BUILDING WIND STUDY

Computational Wind Engineering (CWE) is a branch of CFD concerned with behavior of wind. Similar to wind tunnel tests, it can be used to understand the wind flow through an urban environment and the effect of a proposed development on the local wind microclimate. Unlike boundary layer wind tunnel tests, a virtual topographical model can be constructed at full scale and therefore, avoids any similarity problems (i.e. Reynolds number violation). In addition, the wind speed profile, which is consistent with reality, can be specified directly in CWE rather than generated artificially in a wind tunnel using additional roughness elements.

The proposed development at the Dublin Docklands consists of the construction of an office building ranging in height from 8 to 17 storeys, known as the Dublin Docklands tall building. The construction of new buildings and the alteration of the existing landscape may alter the flow of the wind in the surrounding area [8]. The proposed development can generate a wind environment at ground level that is discomforting or even possibly dangerous to pedestrians. The assessment of discomfort and distress of pedestrians was carried out in accordance with the Lawson Comfort Criteria [9] (Table 1). The main source of pedestrian discomfort relates to the force of the wind felt on their body and their clothing, as additional effort is required to negotiate the wind. Even though the force acting on a person is related to the wind speed, the level of pedestrian discomfort depends on frequency of occurrence and the activity being undertaken as well as the wind speed.

Table 1. Lawson Comfort Criteria.

| Wind effect | Threshold | Stationary | Strolling | Transit |
|---|---|---|---|---|
| Calm | | | | |
| Felt on face | | | | |
| Leaves move | 4.0 m/s | | | |
| Dust raised | 6.0 m/s | | | |
| Felt on body | 8.0 m/s | | | |
| Hard to walk | 10 m/s | | | |
| Trees moving | | | | |
| Storm | 15 m/s | | | |
| Dangerous | 20 m/s | | | |

A boundary layer wind tunnel study of the proposed development at the Dublin Docklands was previously conducted by BMT Fluids to assess the impact of the proposed development on the wind microclimate. Arup were commissioned to conduct a further wind assessment to examine the locations identified as causing pedestrian discomfort or distress and to evaluate the effectiveness of the proposed mitigation measures.

The local wind climate was determined from historical meteorological data recorded at Dublin Airport. Two different datasets were analysed, namely; the data associated with the maximum daily wind speeds recorded over a 30 year period between 1985 and 2015, and the mean hourly wind speeds recorded over a 10 year period between 2005 and 2015. The wind speeds in the vicinity of the development will differ from the wind recorded in Dublin Airport. It is necessary to transform the wind speeds to take account of local conditions [10]. From this, a single wind speed profile was determined for each direction for both comfort and distress criteria, as illustrated in Figures 1 and 2.

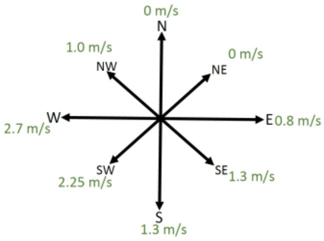

Figure 1. Comfort criteria

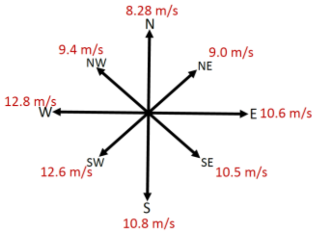

Figure 2. Distress criteria

The geometry for the CFD model was developed in 'Sketchup' drawing package using drawings received from the Client and Google Earth images of the surrounding area. The geometry was imported into 'Fire Dynamics Simulator' (FDS) software [11-12], used in the CFD modelling. A view from the East of the Dublin Docklands tall building is illustrated in Figures 3 and 4. The Dublin Docklands tall building was taken at the center of the computational domain which extended 550m in the North-South direction and 410 m in the East-West direction. The domain was modelled as 102.4 m in height.


Figure 3. Sketchup Geometry

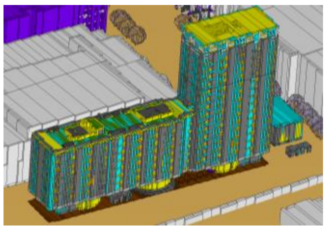
Figure 4. CFD model

As part of the computational process, the domain was divided into a total of 16million cells. The cells range in size from 0.4 m at the Dublin Docklands tall building to 3.2 m at the outer domain. Closed boundary conditions were applied on the bottom face of the domain, and open boundary conditions were modelled elsewhere. Data sampling locations were chosen to be as close as possible to the exact sensor locations used in the wind tunnel tests.

For the purposes of validation of the CFD modelling process, the conditions in the wind tunnel were modelled using CFD and comparisons were drawn. The results of this validation showed good consistency with the original wind tunnel test results.

The assessment of the pedestrian comfort and distress in the pedestrian thoroughfare between the 3Arena and the Dublin Docklands tall building indicates that the main areas of concern are at the southern end. The equivalent wind speeds from the perspective of comfort and distress are 12m/s and 24 m/s respectively (Figure 5). These speeds arise when high westerly winds are directed to ground level by the building façade. These high winds cause discomfort and distress between the 3Arena and the Dublin Docklands tall building (Figure 6). Midway along the building, the equivalent hourly average gust wind speed from a comfort perspective is 8.3 m/s. At the northern end, the analysis reveals that the wind speeds are insufficient to cause either discomfort or distress.

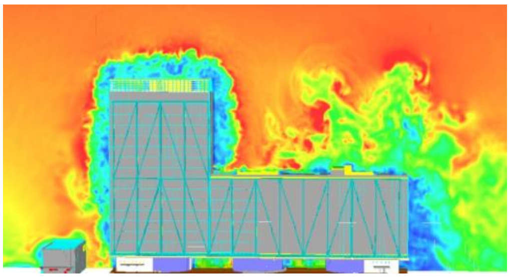

Figure 5. Distress Criteria, Wind from the West, Minimum values in blue (0m/s), Maximum values in red (20m/s)

In addition, it is possible that the winds from the south and north will cause distress to the general public. The equivalent hourly average gust speed is estimated at 15.4m/s and 16.3m/s from the North and Southwest, respectively. These wind speeds exceed the distress threshold for the general public of 15m/s.

The assessment of the pedestrian comfort and distress in the passageways between the Dublin Docklands tall building reveals that the wind conditions might be discomforting and distressing to pedestrian. However, the level of discomfort and distress is related to the wind direction. For instance, while westerly winds with an annual return period may produce conditions unsuitable for undertaking any activity within the undercroft, a slight shift in direction to the south results in an acceptable environment. It would appear that much of the pedestrian discomfort and distress is due to westerly winds. From the perspective of comfort, the equivalent hourly average gust speed in the undercroft passageways ranges between 10 m/s and 13.9 m/s for westerly winds, which is considered uncomfortable irrespective of the activity being undertaken. The main source of distress is due to westerly winds which range between 15m/s and 30m/s along the length of the building. These high wind speeds near ground level are due to the building funnelling high level winds downward. The wind speeds are further increased through the undercroft passageways as the wind is forced through narrower openings underneath the building (Figure 6). Although it might be expected that easterly winds might cause similar pedestrian discomfort and distress within the building undercroft, it is apparent from Figure 7 that this is not the case. The wall on the western boundary of Dublin Port acts to disturb the wind. The bluff nature of the wall causes the flow to separate and the formation of large vortex between the wall and the Dublin Docklands tall building (Figure 8). The vortex acts to push much of the wind over the Dublin Docklands tall building. As a consequence there is less flow passing under the Dublin Docklands tall building.

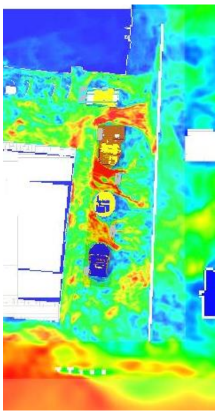

Figure 6. Distress Criteria, Wind from the West, Minimum values in blue (0m/s), Maximum values in red (20m/s)

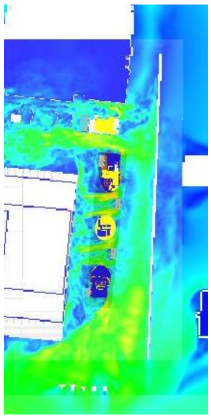

Figure 7. Distress Criteria, Wind from the East, Minimum values in blue (0m/s), Maximum values in red (20m/s)

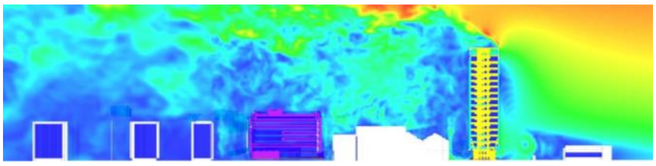

Figure 8. Distress Criteria, Wind from the East, Minimum values in blue (0m/s), Maximum values in red (20m/s)

The assessment of the Dublin Docklands tall building indicated that the southern extent of the pedestrian thoroughfare between the 3Arena and the Dublin Docklands tall building is prone to pedestrian discomfort and distress. Pedestrian discomfort and distress is also predicted in the undercroft of the Dublin Docklands tall building. The analysis reveals that high speed winds at ground level responsible for possible pedestrian discomfort were largely due to westerly winds, where the building acts to direct high level wind towards the ground. The provision of a 2.5 m wide canopy with a 0.45 m upstand for the full width of the western side of the building is proposed as a mitigation measure.

## 4 HOSPITAL THERMAL COMFORT STUDY

Arup was commissioned to conduct a thermal comfort analysis of a proposed new Hospital. An initial analysis of historical weather data was carried out to identify thermal conditions. Computational Fluid Dynamics (CFD) modelling was utilized in order to understand the internal flow field in the bedrooms and to identify zones of patient discomfort.

Thermal comfort is a subjective evaluation by humans of their satisfaction with the thermal environment. Maintaining a standard of thermal comfort for occupants of enclosures is an important goal of HVAC systems [13-14]. Owing to the global initiative to improve carbon footprints, many building owners are opting for naturally ventilated systems. A desired internal temperature of 22°C for patient comfort has been specified by the Design Team. The objective of this study is to estimate whether an internal temperature of 22°C can be maintained:

- During winter conditions, with an ambient outside temperature of -4.5°C
- With the louvers in the open position
- Assuming the heating system of radiant panels prescribed by the Design Team is fully operational
- Assuming the geometry and layout of the rooms as received from the Design Team.

The temperature climate in the vicinity of the new hospital was estimated from almost 20 years of historic temperature data (December 1996 until December 2015) recorded at a local airport. Analysis of the data revealed that the most frequent minimum temperatures were between 4-8°C, however is was not uncommon to have temperatures as low as -4°C. On seven occasions in 20 years, the temperature fell below -8°C, and the minimum temperature recorded over the analysis period was -10°C which occurred twice (Figure 9).

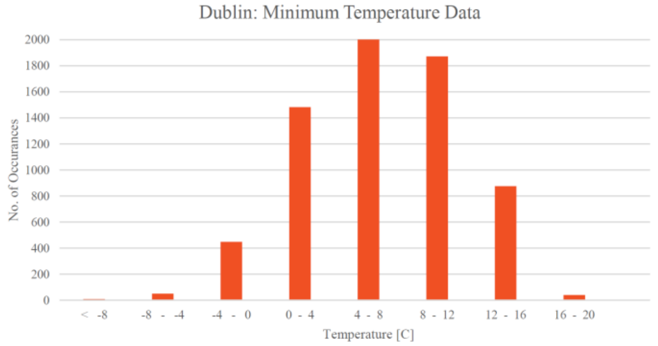


Figure 9. Frequency of Minimum Temperature Data

EN1991-1-5 provides information on ambient temperature for thermal modelling. The indicative outdoor temperature may be determined from Figure NA.1, Table 5.1 and Table 5.2. Taking into account the weather maps provided by the Eurocode, winter conditions, a light coloured façade surface, and a North-East facing orientation, a minimum air temperature of -9°C was determined.

The geometry for the CFD model was created using the Revit model previously developed for the project. A total of three top-floor bedrooms were modelled, each with a floor area of 23 m$^2$. The slab-to-slab height was taken as 4.0 m and a false ceiling was included at a height of 3.0 m above floor level. Radiant panels were modelled as surface mounted on the false ceiling, as illustrated in the plan view of the model (Figure 10). Each bedroom has three 3.0 m x 0.6 m panels, one 1.2 m x 0.6 m panel and one 1.8 m x 0.6 m panels giving a total area of radiant panels per bedroom of 7.2 m$^2$. Glazing units 2.5 m x 1.95 m and aluminum spandrel panels 2.5 m x 1.0 m and 2.5 m x 0.75 m were modelled, all of which can be seen in the elevation view of the model (Figure 11).

Extractor units with a diameter of 0.15 m were modelled in the vicinity of the shower in the ensuite of each room. A transfer filler of area 0.6 m x 0.6 m with 50% free area was modelled in the lower portion of the ensuite doors. A louver of 2.5 m wide and 0.3 m high was modelled in each room. The free area of these louvers was taken as 2.5 m x 0.2 m.

As specified by the Design Team, the heat flux of each 3 m x 0.6 m panel was taken as 1 kW/m$^2$. The heat flux of the smaller panels were factored down by area. The extractor flow rate, also specified by the Design Team, was taken as 0.065 m$^3$/s.

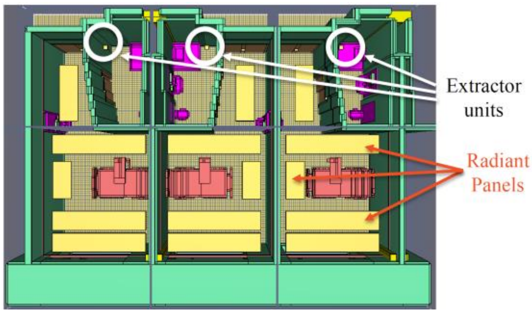


Figure 10. Plan view of bedroom

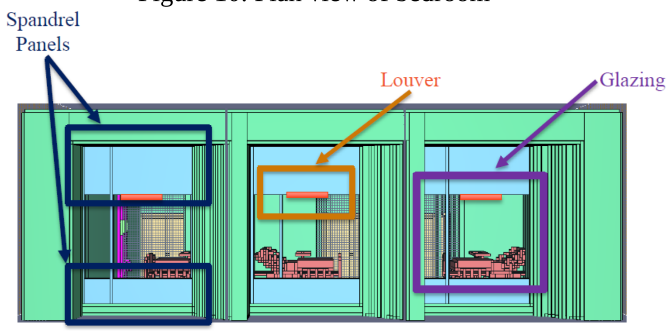


Figure 11. Elevation view of bedroom

The boundaries to the model at the floor, the ceiling, the back (along the internal corridor) and the sides (internal partitions with neighbouring bedrooms) were modelled as closed boundaries. The internal initial temperature was taken as 22°C. The front of the model (the side containing the windows) was modelled as an open boundary, with an applied external ambient temperature of -4.5°C. This is represented by the blue portion in Figure 12.

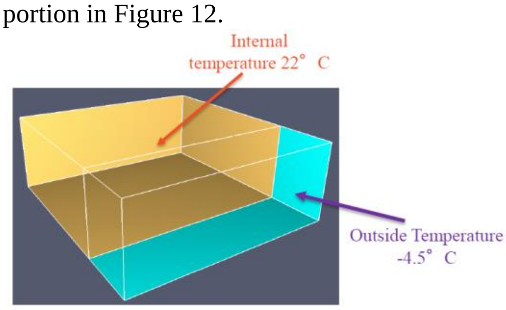


Figure 12. Initial conditions

The results from the CFD modelling at a height of 0.5m above the floor are illustrated below. Figure 13 presents the internal and external temperatures when the louver is in the open position and the external temperature is set at -4.5°C. Results indicated that the internal temperature at a height of 0.5 m above the floor reaches 17.2°C when the external temperature is 0°C and 18.4°C when the external temperature is +5°C.

The results from the CFD modelling at a height of 1.5 m above the floor were also captured when the louver is in the open position and the external temperature is set at -4.5°C, 0°C and +5.0°C respectively. Results indicated that the internal temperature at a height of 0.5 m above the floor reaches 17.8°C when the external temperature is 0°C and 19.0°C when the external temperature is +5°C.

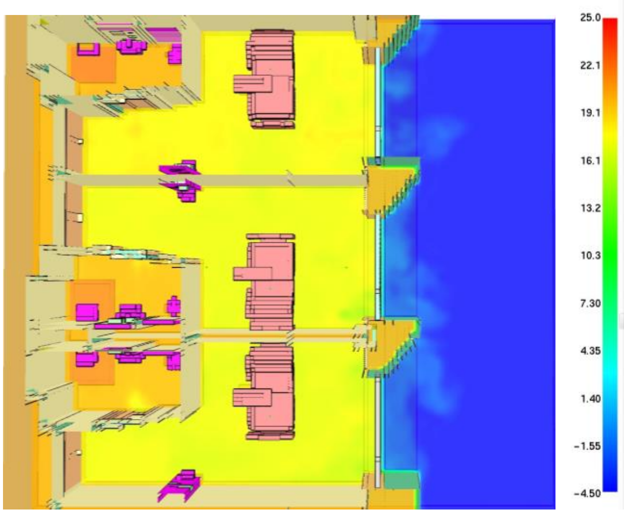

Figure 13. External Temperature -4.5°C

Figure 14 is an elevation view from outside the windows of the bedroom. It serves to highlight the contrast between the temperatures near the radiant panels and the temperatures in the vicinity of the bed when the external temperature was taken at -4.5°C. The results of the CFD modelling demonstrate that the heat flux produced by the prescribed radiant panels in each room are not capable of maintaining an ambient internal temperature of 22°C with either size louver, when they are in the open position and the external temperature is -4.5°C. Results also indicated that an ambient internal temperature of 22°C cannot be maintained with the larger louver and en external temperature of +5.0°C.


Figure 14. External Temperature -4.5°C

From the analysis of temperature data carried out for the local area, peak temperatures of +5.0°C occur approximately 5% of the time, average temperatures of +5.0°C occur approximately 16% of the time and minimum temperatures of +5.0°C occur approximately 34% of the time. Referring back to Figure 9, a temperature of +5.0°C occurs in the most frequent band. Given this, and the results of the CFD simulations – the proposed configuration of radiant panels and louvers will not deliver an ambient temperature of 22°C and will result in a thermally uncomfortable environment. Results from this work resulted in a revised consideration of the proposed ventilation system for the hospital.

## 5 DUNDRUM VENTILATION SHAFT FIRE STUDY

Smoke Management System Limited commissioned Arup to produce a CFD modelling analysis for the mechanical differential pressure shaft proposed for the Dundrum Apartment block, Dublin. The building is an existing 3 storey high residential development with a basement car park. The height of the highest floor is 8.575 m above ground.

The proposed system (Figure 15) comprises a mechanical extract system of 2 $m^3/s$, which is linked to the lobby area with a vent situated at high level close to the ceiling, and a naturally ventilated inlet shaft, which connects to the lobby area via a vent that is located at low level (i.e. with an inlet point close to the floor).

The system for the lobby area is composed of two extract fans (one in operation and one on standby) that are sized in order to not exceed a differential pressure of -50 Pa inside the lobby area. This is in order to avoid leakage of smoke into the staircase area while permitting the opening of the doors as the system continues to extract air smoke. The two extract fans are located on the roof. The fans are connected to each lobby through a shaft, which has an approximate area of 0.5 $m^2$, with a damper situated close to the ceiling in each lobby.

The replacement air supplied into the lobby is provided through a small inlet shaft. This shaft, which supplied the make-up air, is connected to each lobby though a low level louver or transfer grille. The purpose of the provision of make-up air is to prevent the pressure differential of -50 Pa being exceeded.

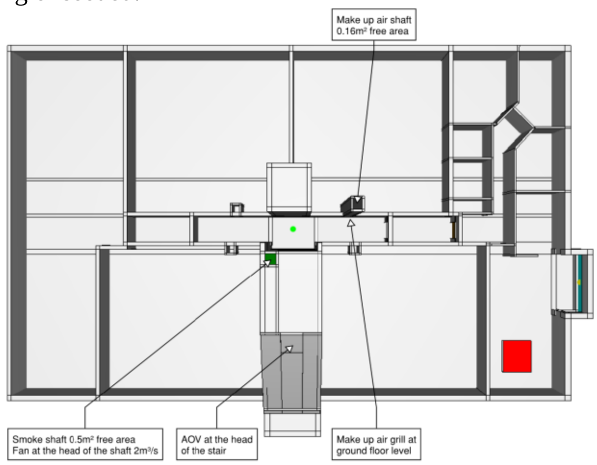

Figure 15. Proposed depressurisation system

The pressure is maintained by the imbalance between the extraction rate and the inlet rate of airflow. Upon detection of fire in any of the lobbies, the extraction fans come into operation. The smoke shaft damper opens on the level where the fire is detected, while the dampers on the other levels remain closed. Air from the make-up shaft will be drawn into the appropriate lobby area automatically by the extraction system. The system will extract continuously and it will be sized in order to not exceed the -50 Pa with the staircase door closed.

Through a CFD analysis, the capability of the de-pressurisation mechanical smoke system to achieve the following tenability criteria [15] for the lobby area is examined:

- Visibility: minimum 10m to light-reflecting signage (e.g. wall);
- Temperature: maximum 60°C at 1.8 m (head height) and below for occupants;
- Velocity: velocity 5 m/s at all locations on the egress route, except at the inlet/ extract vents. A velocity of 10

m/s is acceptable in close proximity to the inlet and extract vents;

- Pressure: a pressure difference of < 50 Pa should ensure that the door opening force does not exceed 100 N at the handle of a typical (0.8 m x 2.0 m) single leaf door. The pressure difference across the stair door (and doors to apartments other than that of fire origin) will be limited such that the force required to open the door(s) does not exceed 100 N.

Two models were developed for the purpose of this study:

1) Fire Model: this scenario aims to demonstrate that the proposed depressurization system is capable of preventing smoke entering the evacuation staircase in the event of a fire in one of the apartments.
2) Pressure assessment model: the aim of this scenario is to demonstrate that the proposed depressurization system is capable of maintaining a sufficient pressure differential (i.e. <50 Pa) within the lobby so that doors can be opened and closed when the smoke extract system is operating.

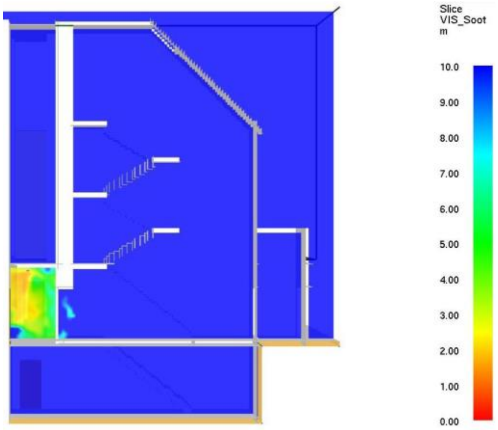

Figure 16. Visibility in the escape stair

The CFD results for the fire scenario have shown that smoke does not spread to the escape staircase. Tenable conditions for visibility (more than 10 m) and temperature (less than 60°C) are maintained at all times inside the escape staircase (Figure 16). Therefore, this indicates that the proposed despressurisation system is capable of preventing smoke entering the staircase. In addition, smoke spreading within the lobby is cleared at 210 s which corresponds to less than 120 s after the apartment door is closed. The time taken to clear the smoke inside the corridor is less than 2 minutes, which is deemed an acceptable performance of the smoke control system. In addition, the velocity levels are kept below 10 m/s except on the vicinity of the inlet grill which is considered acceptable.

The CFD results for the pressure assessment model have shown that the pressure levels are maintained between -12.5 and 7.5 Pa when the proposed depressurisation system is activated and reached steady state conditions. These pressure levels are within the range of -50 and 50 Pa which corresponds to the acceptance criteria so that occupants can open or close doors in the lobby. Therefore this indicates that the proposed despressurisation system is capable of maintaining a sufficient pressure differential within the corridor so that doors can be opened and closed when the smoke extract system is operating.

## 6 CONCLUSION

This paper presented three projects recently completed by the Arup Ireland CFD team. These three examples serve to highlight the wide breadth of applications possible with CFD modeling, providing a depth of understanding and insight. In the case of the Dublin Docklands wind study, CFD had a role to play in informing design such to mitigate unpleasant wind conditions. Permission was granted by the planning authority placing CFD analysis as equivalent to wind tunnel testing. For the Hospital thermal comfort study, CFD modelling indicated that the initial design would result in uncomfortable conditions in the bedrooms, where they would be warm enough but without inflow of fresh air, or too cool but with inflow of fresh air. The outcome of the CFD study was to modify the design. For the smoke shaft in Dundrum, CFD modelling was able to show that the modified smoke shaft had the capacity to perform adequately and satisfy tenability criteria. In summary, CFD has the potential to improve quality of design, is of immediate practical use and has a significant role to play in consulting engineering in the future.

## REFERENCES


[1] J.H. Ferziger and M Peric, *Computational Methods for Fluid Dynamics*, Springer, third edition, 2002.

[2] Lien, J. and Ahmed, N., (2012), 'Indoor air quality measurement with the installation of a rooftop turbine ventilator', *Journal of Environmental Protection*, 3, 1498-1508.

[3] Senveli, A., Dizman, T. and Celen, A., (2014), 'CFD analysis of smoke and temperature control system of an indoor parking lot with jet fans', *Journal of Thermal Engineering*, 1(2), 116-130.

[4] Montazeri, H and Blocken B., (2013), 'CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: validation and sensitivity analysis', *Building and Environment*, 60, 137-149.

[5] Meroney, R.N., (2009), 'CFD prediction of airflow in buildings for natural ventilation', *in Proceedings of the 11th Americas Conference on Wind Engineering*, San Juan, Puerto Rico.

[6] Prasad, K., Li, K., Moore, E.F., Bryant, R.A., Johnson, A. and Whetstone, J.R., (2013), 'Greenhouse gas emissions and dispersion', *NIST special publication 1159*.

[7] Kobayashi, N., and Chen, Q., (2003), 'Floor-Supply displacement ventilation in a small office', *Indoor and Built Environment*, 12(4), 281-292.

[8] Penwarden, A.D., (1973), 'Acceptable wind speeds in towns', *Building Science*, 8, 259-267.

[9] T.V. Lawson, *Building Aerodynamics*, Imperial College Press, London, UK, 2001.

[10] E. Simiu, *The design of buildings for wind: a guide for ASCE 7-10 Standard users and designers for special structures*, second edition, John Wiley & Sons Inc, Hoboken, New Jersey, USA, 2001.

[11] K. Mc Gratten et al., Fire Dynamic Simulator (Version 6) User's Guide 2014: NIST Special Publication 1019.

[12] K. Mc Gratten et al., Fire Dynamic Simulator (Version 6) Technical Reference Guide, NIST Special Publication 1018.

[13] Papakonstantinou, K.A., Kiranoudis, C.T., Markatos, N.C., (2000), 'Computational analysis of thermal comfort: the case of the archaeological museum of Athens', *Applied Mathematical Modelling*, 24, 477-494.

[14] Webb, M., (2013), 'Building energy and CFD simulation to verify thermal comfort in under floor air distribution (UFAD) design', *in Proceedings of the 13th international building performance simulation association*, Chambery, France,

[15] PD 7974-6:2004 Applications of fire safety engineering principles to fire safety design of buildings. Human factors: Life safety strategies – occupant evacuation, behaviour and condition (sub-system 6).