Application of Computational Fluid Dynamics to Address Environmental and Buildings Design Challenges

Neihad Hussen Al-Khalidy
SLR Consulting
CFD, Wind and Energy Technical Discipline
2 Lincoln Street, Lane Cove 2066
AUSTRALIA
Nal-khalidy@slrconsulring.com http://www.linkedin.com/pub/neihad-al-khalidy/19/a5/341

Abstract: - Challenges facing commercial and high profile building projects are to achieve mandated local government and buildings codes for energy, ventilation, air quality, pedestrian wind comfort, thermal comfort and other specialist building services while meeting organisational objective to maximum return on investment. This paper introduces the role of Computational Fluid Dynamics (CFD) as a sophisticated thermo-fluid modelling method to solve building design problems and optimise the design of buildings. Case studies are provided for a number of topics including thermal comfort, combined internal-external air flow assessment, wind driven rain, pollutant dispersion study and natural ventilation design applications. The paper also discusses the application of CFD for entire city modelling. The use of CFD for entire city modelling will be a useful tool to help urban designers and environmental planners.

Key-Words: - *CFD, Building Design, Wind, Ventilation, Thermal Comfort, Air Quality, Urban Design*

1 Introduction

The building industry is currently facing plenty of challenges in terms of climate change and sustainability to meet community expectation and on the other hand the building project team is under increased pressure to meet organisational goal to maximize return on investment.

The increasing environmental concerns start influencing architectural design from a technical standpoint and a political influence that makes “green buildings” a good image for corporate architecture [1].

Since last decade Computational Fluid Dynamics (CFD) has been playing an important role in building design. CFD can be used to optimise the design of sustainable buildings, diagnose air flow problems during the concept design stage of building projects and promote innovative engineering solution [2].

This study provides insight into using Computational Fluid Dynamics to optimise the design of sustainable buildings, diagnose air flow and thermal problems during the concept design stage of building projects, promote innovative natural or mixed mode ventilation design and solve other complex wind driven rain and condensation problems.

2 Problem Formulation

CFD uses numerical methodologies to simulate and analyse fluid flow, heat transfer and pollutant dispersion in built environment.

The CFD model solves the continuity, momentum, energy and species concentration (if required) equations. The equations for a steady state case can be written as follows:

$$\frac{\partial}{\partial x_i}(\rho\, u_i) = 0$$

$$\frac{\partial}{\partial x_i}(\rho\, h) + \frac{\partial}{\partial x_i}(u_i\, \rho\, h) = \frac{\partial}{\partial x_i}(k_{effective}\,\frac{\partial T}{\partial x_i}) + S$$

$$\frac{\partial}{\partial x_j}(\rho\, u_i\, u_j) \;\; = - \;\frac{\partial p}{\partial x_i} + \frac{\partial \tau_{ij}}{\partial x_j}) + \rho\, g_i + F_i$$

$$\frac{\partial}{\partial x_j}(C u_i) = \frac{\partial}{\partial x_i}(\frac{\nu_t}{\sigma}\,\frac{\partial C}{\partial x_j})$$

Where ρ is the density, *u* is the velocity, *p* is the static pressure, ρg and *F* are the gravitational body and external body forces, τ_{ij} is the stress tensor, *h* is the enthalpy and $k_{effective}$ is the effective thermal conductivity, *C* is the pollutant concentration and *v* is the kinematic vescoisty and *S* is the volumetric heat source.

Turbulence is predicted using one of the following methods:

- Direct Numerical Simulation (DNS)
- Large Eddy Simulation (LES)
- Reynolds-Averaged Navier-Stokes (RANS) Equations

For most real world building problems turbulence is, in principle, described by the Navier-Stokes equations [3]. Commercial CFD codes provides a wide range of turbulence models including Spalart-Allmaras model, $k-\varepsilon$ models, k-w models, v2f model, Reynolds Stress models, Scale Adaptive Simulation (SAS) model, Detached Eddy Simulation, Large eddy simulation (LES) models.

The quality of CFD simulation depends on the selected turbulence model. In practical problems the turbulence model should be as simple as the relevant physics will permit.

3 Case Studies

3.1 Case Study 1: Combined Internal-External Wind Assessment

In the past, partially due to the limitations of commercial CFD software and processing power, it has been the norm to use CFD analysis on the interior airflow only. However, with advances in processing power, specialised wind engineering experience and specialised in house and commercial CFD software we were able to complete a combined real time internal-external flow analysis for a number of high profile buildings in Australia, UK and the UAE.

This case study presents an example of using CFD for a proposed commercial building site planning. It is one of the requirements of developers that they ensure pedestrian safety by creating safe wind levels in and around their projects. The CFD assessment in this case study was conducted in accordance with the following Lawson Comfort and safety Criteria.

"Comfort" criteria relate a number of typical pedestrian activities such as purpose-walking, strolling, sitting, etc, in terms of the mean wind speed which is exceeded 5% of the time, on an annual basis.

"Safety" criteria cover the circumstance where pedestrians might encounter difficulty in walking. They are defined by the incidence of mean wind speeds occurring once or twice per year.

Lawson criteria couple the probability of exceeding winds at given statistical levels with wind speed magnitudes originally related to the Beaufort Land Scale [4].

The CFD modelling involved the followings:

- Developing a 3D model for the proposed development;
- Adding topography and surrounding buildings to an approximate radius of 500 m;
- Developing mixed meshed cells (tetrahedral, hexahedra and pyramids) for the computational domain;
- Integrating the local weather data with the developed CFD model and defining the upwind free boundary inlet. At the upwind free boundary inlet, velocity profiles were derived from Met Bureau data and building code for the project site;
- Applying "constant pressure" boundary conditions at the downwind and upper free boundaries;
- Calculating turbulence quantities (kinetic energy and dissipation rate) required for the upwind free boundary from empirical relationships [5];
- Selecting a proper turbulence model and numerical scheme for the assessment. A Realizable k-epsilon (rke) turbulence model [6] was used for all analysed cases;
- Predicting air velocity in terms of three directions, pressure profile and turbulence parameters. For the pressure velocity coupling a global solver based on the SIMPLE algorithm was employed [7] and

- Providing guidance as to the areas where the adopted wind acceptability criterion had the potential to be exceeded and an indication as to the likely local optimum wind treatment strategy.

For the current analysis, between 5 million and 9.2 million mixed cells (tetrahedral, hexahedra and pyramids) were used to cover the computational domain. Mesh distribution on one of the proposed buildings is shown in Figure 1. One can see that fine structured meshes were used near the ground. Mesh distribution was then optimised using a Solution Adaptation Technique, eg refining the mesh based on the numerical results for each prevailing wind direction. A Realizable k-epsilon (rke) turbulence model combined with a wall function data group was used to avoid using a very fine mesh near the wall and improve turbulent flow simulation.

The software package utilised in the current CFD analysis is the commercially available code Fluent.

Figure 2 shows that the normalised residual of continuity was reduced by between five and six orders of magnitude while the normalised residuals of x-, y-, and z-velocity, k and epsilon were reduced between six and eight orders of magnitude demonstrating a valid numerical solution.

Part of the results is shown in Figure 3. Figure 3 shows mean airflow velocities ratios ($V_{local}/V_{reference\ at\ 10m}$) at pedestrian level (1.5 m above the ground) for the easterly wind direction. Velocity ratios are plotted on a colour coded scale between 0 and 1.82. One can see that the flow characteristic is captured well be the developed CFD model where the wind is approaching the site with Vref=1 as per the given boundary condition. Wind is then accelerated near the corners and a stagnation region is located immediately downstream of the proposed building and a number of neighbouring buildings.

The CFD results for all analysed wind directions have been combined with the wind probability information from the local wind rose and assessment predictions have been developed in terms of Comfort and Safety using the Lawson Criteria. In areas of elevated wind speeds wind mitigation treatments such as dense landscape, wind screens, etc. were recommended to satisfy the assessment criteria.

The production of such models is challenging due to a combination of large dimensions and the non orthogonal geometry of many small building features. In the last decade we have undertaken extensive R&D on completed development to confirm modelling accuracy against wind tunnel data.

A comparison between the CFD and the wind tunnel results using sound scouring technique for an iconic shopping centre in NSW Australia is shown in Figure 4 [8]. One can see that scoured regions underneath the footbridge in the wind tunnel test model are similar to that in the CFD model. The CFD model was used to assess wind movement though the retail levels of the proposed development including all building entrances and openings.

Fig.1 Sample Mesh

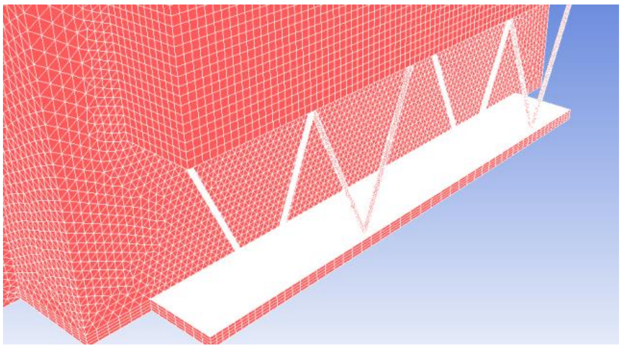
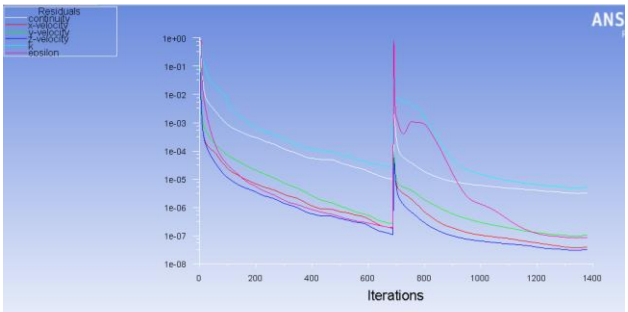


Fig.2 Scaled Residual History



ISBN: 978-960-474-398-8

Fig.3 Mean Velocity Ratio (V_{local}/V_{10m})

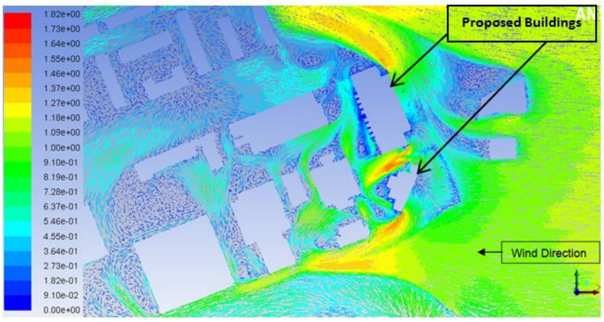
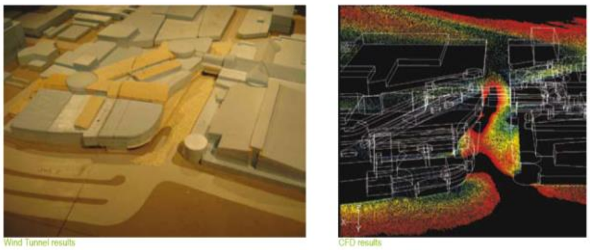


Fig.4 Comparison between Wind Tunnel and CFD Results



3.2 Case Study 2: Macro/Meso Scale Air Quality Modelling

The topic of this case study is the macro scale air quality in the built environment on the example of the dispersion of gases from gas powered electricity generators within and then downstream of building complexes in a city. The generators located on the roofs of commercial buildings throughout the city are proposed to be integrated with the CBD grid and to operate during peak power demand when the electricity transmission infrastructure has a shortfall in providing the required power. Initially, it has been proposed to integrate more than 30 generators with a total capacity of more than 65 MW with the CBD grid within inner city buildings. The objectives of this study therefore were to:

- Develop a model of the entire city (Refer Figure 5);
- Incorporate the generators into the model;
- Predict pollutants concentrations namely NOx on the city ground level;
- Predict pollutant concentrations on buildings walls and balconies for a number of prevailing winds directions;
- Study the impacts of stack heights on the city air quality and building air-conditioning intakes and
- Optimise number and locations of the proposed generators.

A three dimensional geometric model was assembled for CFD modelling to capture the complex air flow pattern and pollutant dispersion within the vicinity of the buildings in the city taking into account the often complex geometry of individual buildings and regional terrain.

A sensitivity analysis was carried out to define the domain of analysis and study the impact of approximating the geometry of the buildings in the computational domain. Higher pollutant concentrations are predicted when ignoring gaps in the order of 2 m between the buildings [9].

Figure 6 reveals the NOx ppm concentration profiles on the wall of the buildings and at the ground level for a north easterly prevailing wind direction.

In general the following conclusions have been reached based on the CFD results:

- A number of the generators with a total capacity of 25 MW were found to exceed the local government criteria and have unacceptable impact on the air quality of the city CBD.
- Other generators can be integrated with the CBD grid to provide electricity during peak hours without exceeding the criteria and regulations set down by the city council.

The CFD analysis has offered a comprehensive range of output including pollutant concentrations, velocity distribution, pressure profile, turbulence levels, etc.

Fig.5: Geometry of the Modelled City (river is shown in magenta colour)

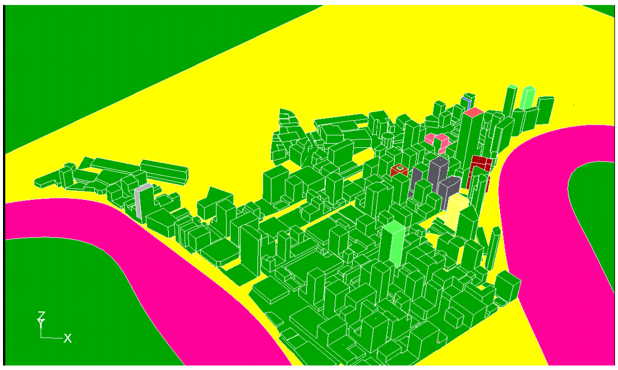
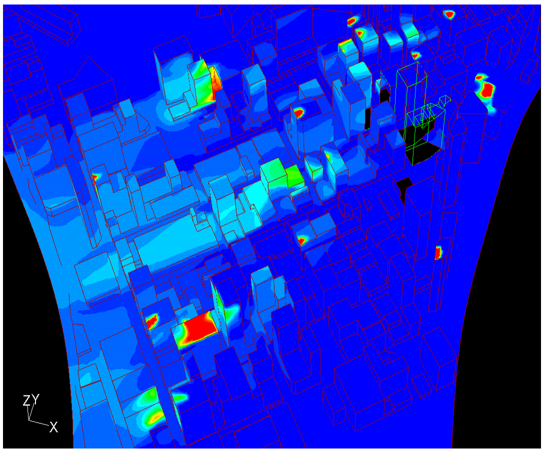


Fig.6: Nox Concentration on the Walls of the Buildings – North Easterly Winds



3.3 Case Study 3: Wind Driven Rain

The topic of this case study is the wind driven rain assessment of thermal chimneys of a proposed educational building in Australia. The proposed building is designed to use the concept of natural ventilation where air is drawn into the building through low level louvres and hot air is removed via the thermal chimneys in the common areas, using thermal buoyancy effects.

The impact of prevailing winds and rain on the proposed building has been carried out via 3-dimensional CFD simulations. These simulations provide quantitative data useful in determining the primary direction of the local windflow and whether rain particles can penetrate through the proposed louvres incorporated into the building's design.

After the steady wind-flow field has been obtained, the raindrop trajectories are calculated using a Lagrangian particle tracking approach to simulate the trajectories of the rain drops. In the Lagrangian approach discrete particles are released into the flow and then tracked by integrating the particle equation of motion. Only the drag and the gravity-buoyancy forces are considered since the rain drops have a much higher density than the surrounding air flow. The particle equation of motion can be simplified as follows:

$$\frac{\pi}{6} d^3 \rho_p \frac{d\bar{U}_p}{dt} = 3\pi \mu d(\bar{U}_f - \bar{U}_p) + \frac{\pi}{6} d^3 (\rho_p - \rho_f)\bar{g}$$

where d is the droplet diameter, U_f is the wind velocity, U_p is the particle velocity, r_f is the air density, ρ_p is the rain droplet density, μ is the air viscosity; and g is the acceleration of gravity.

For a given droplet diameter and initial droplet position and velocity, the trajectory of an individual droplet is computed by integrating the above equation. In order to predict the dispersion of rain particles due to turbulence, a stochastic method known as "Random Walk Model" is implemented to determine the instantaneous wind velocity [3].

Results of simulation are shown in Figure 7 and Figure 8. Figure 8 shows the computed trajectories for very fine rain drops of 0.2 mm diameter. One can see that most particles can penetrate through the louvres. Many particles will follow the wind field and leave from the leeward side. Approximately 12% of all released particles trapped in the recirculation region wet the base of the thermal chimney.

The study proposed and modelled a number of design modification options including solid vertical screens in front of the louvres, dampers and motorised louvres to prevent ingress rain water during persistent rains with fine droplets combined with unfavourable wind directions.

Fig.7 Contours of Velocity Magnitudes (m/s) at a 2D Section through a Selected Chimney

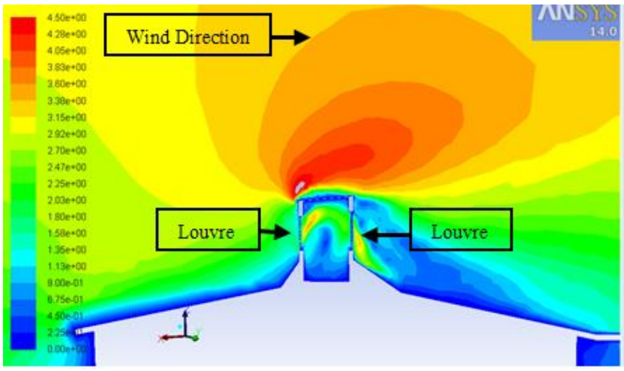
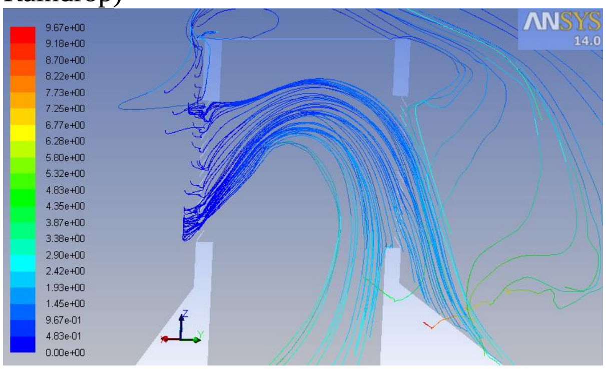


Fig.8: Particle Path Lines (0.2 mm Diameter Drizzle Raindrop)



ISBN: 978-960-474-398-8

3.4 Case Study 4: Thermal Comfort Assessment

The assessment of thermal comfort is usually approached by computing the Predicted Mean Vote (PMV) and the Predicted Percentage of Dissatisfied (PPD) [10]. Thermal environments are mainly affected by air temperature, relative air speed and humidity. Besides there environmental factors there are personal factors such as metabolic heat production and clothing that affect the thermal comfort.

Local air velocity magnitude, local air temperature, and local static air pressure for a mechanically, naturally or a hybrid ventilated area can be accurately predicted by CFD simulations.

The CFD approach can employ a user defined function or a thermal comfort module to calculate the PMV and PPD thermal comfort indices within the analysed area.

The current case study involves the thermal comfort modelling and design optimisation of a general learning area accommodation for a maximum of 135 people in Adelaide. The design incorporates automatic glass louvers located in the north side of the school (Refer Figure 9) to provide optimal airflow control. The initial design also suggested locating four thermal ventilation chimneys through the roof of a number of selected areas. The ventilation system in the building mainly relies on the buoyancy effect ventilation results from differences in air density and wind driven ventilation.

The aim of this study was to test and optimise the proposed natural ventilation system during the still wind condition.

A geometrically accurate 3-dimensional model of the proposed school was assembled in order to study the comfort conditions (temperature, pressure and air movement) inside the school. The developed CFD model incorporated solar loading, heating load from people, lights and appliance, external ambient calculations, shading and other analytical parameters for calm wind condition.

The CFD results of the base case design has shown that the proposed chimneys can exhaust hot air from the building but their height and opening size are not sufficient to reduce the indoor temperature to an acceptable level.

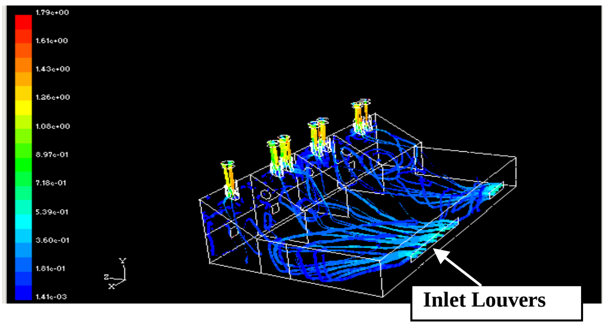
Based on the CFD results a number of design modification options were recommended to significantly improve the thermal comfort conditions for calm ambient wind condition. Design modifications included optimisation of inlet louvered, internal layout, number of chimneys, chimney location and dimensions.

Air flow path line for the final scenario for no ambient wind condition in a typical day in May is shown in Figure 9. One can see that the buoyancy forces commence to move the air by the stack effect. Heated air rises through the ventilated area and exit through the proposed thermal ventilation chimneys.

The CFD approach allowed to achieve an average air flow velocity and temperature of 0.18 m/s and 25.6 ^{o}C respectively during the no wind condition scenario.

The building is now operated and received a number of sustainability building awards.

Fig.9 Path line colored by velocity magnitude – Final Design Scenario



3.5 Case Study 5: Design Analysis of Single Sided Natural Ventilated Apartments

The use of CFD to improve natural ventilation in single sided ventilated apartments has gained an increasing interest during the last few years. Single sided ventilated apartments have openings on one façade and as a result those units may suffer lower indoor environmental air quality.

This case study focuses on improving the natural ventilation through the use of recessed facades and slots. Those building features combined with building balcony will help to create pressure differences at the various openings through the façade and increase airflow movement in single sided apartments.

ISBN: 978-960-474-398-8

Typical floor plan for a proposed development in Sydney is shown in Figure 10. The facade of units Type A and Type B were reassessed and the façade of unit Type C is provided with a central slot which has sliding doors from both bedroom and living room.

A 3D model of the proposed development and surrounding buildings are assembled for CFD modelling (Refer Figure 11). The CFD model is then integrated with the local weather data for the project site. In each analysed case a wind speed of 1.66 m/s was used at 10 m high. Based on actual wind data across 11 years, the average wind speed in the project site is higher than 1.7 m/s, 70% of the time.

The overall flow results on a colour coded velocity vector visualisation between 0 and 2 m/s for north east wind direction are shown in Figure 12.

Detailed results on a colour coded velocity vector visualisation between 0 and 0.4 m/s are shown in Figure 13. The analysed apartments appear to show reasonable airflow through the apartments with a maximum wind speed between 0.2 - 0.25 m/s.

The CFD approach can play a major role to optimise the design of single sided ventilated apartments and achieve natural ventilation criteria set by local governments in Australia.

Fig.10 Typical Floor Plan Showing the Proposed Recesses and Central Balcony Slot.

Fig.11 3D Geometry for CFD Modelling

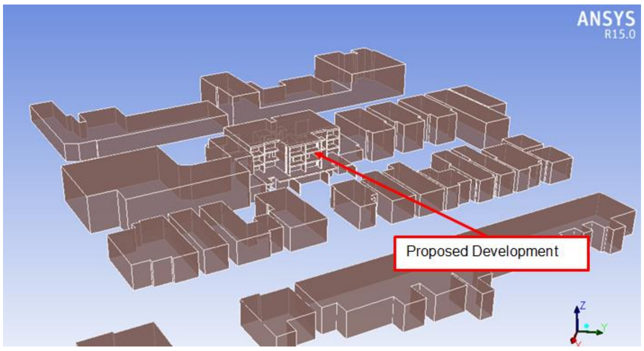


Fig. 12 Overall Northeast Flow (m/s)

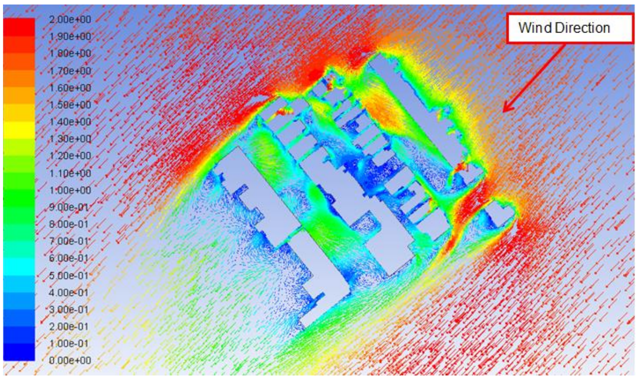
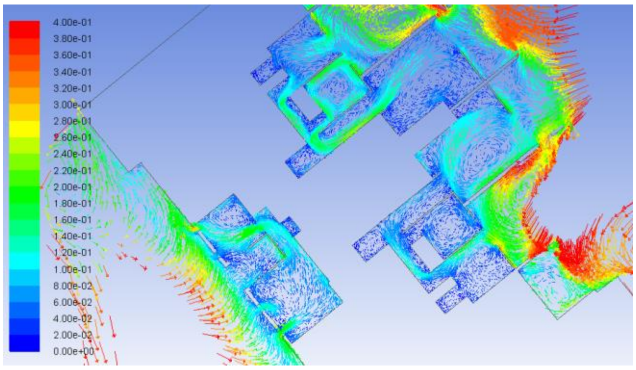


Fig.13 Velocity Vectors in Analysed Apartments



4 Conclusion

This paper introduces the role of Computational Fluid Dynamics (CFD) as a sophisticated thermo-fluid modelling method to optimise the design of buildings or solve building design problems. Case studies are provided for a number of topics including thermal comfort, combined internal-external air flow assessment, wind driven rain, pollutant dispersion study and natural ventilation design applications. The paper also discusses the application of CFD for entire city modelling. The use of CFD for entire city modelling will be a useful tool to help urban designers and environmental planners.

ISBN: 978-960-474-398-8

References:

[1] H. Poirazis, Double Skin Façades for Office Buildings, *Division of Energy and Building Design, Department of Construction and Architecture, Lund Institute of Technology, Lund University*, Report EBD-R--04/3, 2004.

[2] N. Al-Khalidy, Designing Butter Building with Computational Fluid Dynamics Analysis, *4th International Congress on Computational Engineering and Sciences, Las Vegas*, University of Nevada, (Reno, USA), 2012.

[3] B. Launder and D. Spalding, *Lectures in Mathematical Models of Turbulence, Academic Press*, London, England, 1972.

[4] S. Huler, *Defining the Wind: The Beaufort Scale, and How a 19th-Century Admiral Turned Science into Poetry*, Crown, 2004

[5] Fluent Inc, *Theory Manual*, 2002

[6] H. Shih, W. Liou, A. Shabbir, Z. Yang, and J. Zhu, A New, Eddy-Viscosity Model for High Reynolds Number Turbulent Flows - Model Development and Validation, *Computers Fluids*, Vol 24, No.3, 1995, pp. 227–238.

[7] P. Vandoormaal and G. D. Raithby, Enhancements of the SIMPLE Method for Predicting Incompressible Fluid Flows, *Numerical Heat Transfer*, Vol. 7, 1984, pp.147–163.

[8] N. Al-Khalidy, Shopping Sensation, Australian National Construction Review, (http://www.ancr.com.au/Pplaza.pdf), 2006, Page 49.

[9] N. Al-Khalidy, Computational Fluid Dynamics Simulations of Plume Dispersions in City Canyons, *Proceeding of the Fifth International Conference on Engineering Computational Technology*, Las Palmas de Gran Canaria, Spain, 2006.

[10] O. Fanger, *Thermal Comfort: Analysis and applications in environmental engineering*, McGraw-Hill, 1970

ISBN: 978-960-474-398-8