Wind Energy in the Built Environment: A Design Analysis Using CFD and Wind Tunnel Modelling Approach

by

Liliana Campos-Arriaga

Architect MSc.

Thesis Submitted to the University of Nottingham for the degree of

Doctor of Philosophy

May 2009

ABSTRACT

Renewable energies are a critical element for reducing greenhouse gases emissions and achieving a sustainable development. Until recently, building integration of renewable sources was focused on solar technologies. Nevertheless, building integrated wind turbines can and must be part of the solution to the global energy challenge.

This research investigated the potential of integrating small vertical wind turbines between medium-rise buildings. Wind velocities were measured around 7 fifteen-storey towers. The measurements were carried out for nine different configurations, using a boundary layer wind tunnel and computational fluid dynamics (CFD) simulations.

Computed and measured results showed reasonable agreement. The differences were more apparent at ground level. It was established that building orientation and the separation between buildings defines to a great extent the wind environment around buildings. It was found that a distance between buildings of 15 metres and an orientation of $\theta = 260^\circ$ produced the higher augmentation factors. This configuration produced up to 17,812kWh in a typical Nottingham UK year, using six vertical wind turbines of 2.5kW each.

Results suggested that the use of CFD as a visualisation tool is extremely useful at design stages in projects involving the integration of wind turbines. Nevertheless, the results of CFD simulations are highly dependent on the type of roughness modification applied to the wall functions, the choice of the turbulence model and the modelling of the inlet wind velocity profile.

Because servicing buildings accounts for around half of the UK's total energy consumption, the need to reduce the consumption of fossil fuels is central to good building design. That is why the architectural practice must respond professionally by delivering buildings that successfully integrate wind energy technologies, which can only be achieved if the designer actively engages with the environmental design principles and improves his understanding of building physics.

PUBLISHED WORK

- Campos Arriaga, L. and Riffat, S. **2006**. *Assessing the potential of building-integrated wind turbines in an urban Eco-Village.* Pp. 447-452 In: Proceedings of the 5th International Conference on Sustainable Energy Technologies. Vicenza, Italy. 30 Aug. - 1 Sep. 2006

- Campos Arriaga, L. **2006**. *Building integrated wind turbines.* Conference presented at: STaR City TC1: Sector D – Urban Planning Congress: Living in a Sustainable Community; STaR MC FP6, National Technical University of Athens. Athens, GR.

Acknowledgements

A scholarship from CONACYT, Mexico made this research possible, so did the positive intervention of many people in my life. I am most thankful for always having the support of my parents, brother and dear husband. Your love was truly what kept me going all these years.

I would like to thank Professor Riffat and Dr. Omer for their guidance and helpful comments. My deepest gratitude is extended to Dr. David Etheridge for his immense support and invaluable advice in wind tunnel modelling and data analysis.

Many thanks to all my friends for enduring countless thesis related conversations, particularly those of you that read my work. I am especially thankful for being part of the ‘Brotherhood of the Red Wine’ we shared incredible times together.

I am also very grateful to Professor Clito Afonso and to Dr. Guohui Gan for their comments to improve this work.

Finally, my love goes to those who are not here anymore but will be remembered forever, especially to Mª Antonieta Hoyos Campos.

Table of Contents

List of Figures

CHAPTER FIVE

CHAPTER SIX

CHAPTER SEVEN

List of Tables

List of Boxes

List of Symbols

Symbol	Description
A/D	analogue-digital
B	building passage width
BedZED	Beddington Zero (fossil) Energy Development
BUWTs	Building-Mounted / Integrated Wind Turbines
BWEA	British Wind Energy Association
CABE	Commission for Architecture and the Built Environment
CHP	combined heat and power
C_{Ks}	roughness constant
CO_2	carbon dioxide
CO_2-eq	carbon dioxide equivalent
C_p	power coefficient
CTA	constant temperature anemometer
DETR	Department for Environment, Food and Rural Affairs
DNC	declared net capacity
DTI	Department of Trade and Industry
EWEA	European Wind Energy Association
GHG	greenhouse gases
H	building height
HAWTs	Horizontal axis turbines
HS	stagnation point
IPCC	Intergovernmental Panel on Climate Change
k	turbulent kinetic energy

Symbol	Description
κ	von Karman constant
K	amplification factor
K_P	turbulent kinetic energy at the point P
K_s	physical roughness
P_{kin}	kinetic energy carried by the wind
R&D	research and development
RANS	Reynolds Averaged Navier Stokes equations
Re	Reynolds number
SBE	School of the Built Environment
SEV	Sherwood Energy Village
TRY	test reference year
u	wind velocity
u^*	friction velocity
UDF	User Defined Function
UDSs	User-Defined Scalars
UNFCCC	United Nations Framework Convention on Climate Change
UoN	University of Nottingham
U_{ref}	reference wind velocity
VAWTs	Vertical axis wind turbines
y	distance normal to the wall

Symbol	Description
y^+	non-dimensional distance from the wall to first node
y_P	distance from the wall to the centre point P
z_0	aerodynamic roughness length
ZED/LED	zero and low carbon emissions developments
ε	dissipation of turbulent kinetic energy
θ	building orientation
λ	tip speed ratio
μ	kinematic viscosity
ρ	Air density
u_τ	shear velocity
$\overline{q}$	dynamic pressure
μt	eddy viscosity

1 introduction

1.1 Opening remarks

This research works across interdisciplinary frameworks by experimentally exploring the use of wind tunnels and computational fluid dynamics (CFD) as environmental architectural design tools. The experimental and numerical findings are discussed in the ambiguous context of sustainable architectural design, against architectural projects that integrate wind turbines for electricity generation in urban environments. The primary research goal is to develop a conceptual framework for better understanding the role of the architect when environmental projects that include the modelling of wind flows in urban environments for building integrated wind turbines are undertaken.

This first chapter is divided in four main sections (Box 1.1). In section 1.2 the relevance of the elaborated study, contextualised in a global and local framework, is focused on the important issue of climate change and global warming. The role of the built environment on energy consumption is discussed, along with some ideas on how wind energy in urban environments can contribute to the reduction of carbon dioxide (CO_2) emissions. A review of the different approaches to the environmental building design is portrayed as a preamble to section 1.3, in which the scope of the research project is explained by presenting the research questions, hypotheses and objectives. Later on, in section 1.4, the thesis' structure is explained and a brief overview of the entire thesis is provided by presenting a short summary of each chapter. Finally, section 1.5 summarises the key points covered in this chapter and in 1.6 a list of the references used in this chapter is provided.

BOX 1.1 chapter one overview

1.2 The argument	An elaboration on climate change as one of the major challenges today, and the role of environmental architecture in incorporating renewable energy systems, such as building-integrated wind turbines, to reduce CO_2 emissions.
1.3 The scope	This study explores the role of simulation and experimental tools to investigate possible locations for building integrated wind turbines as part of environmental building design.
1.4 The structure	This thesis is divided in eight chapters within three sections: a)scope, context and methods; b) experimental work, results, and analysis; c) discussion, interpretation and final remarks.
1.5 The key points	Summarised in Box 1.3

1.2 The argument

1.2.1 Different climate, no change?

"In recent weeks there has been a meeting of 163 nations at the Climate Conference in Kyoto in Japan trying to hammer out a global strategy to manage climate change and reduce carbon emissions. This followed the Special Session of 70 Heads of State at the United Nations in New York, the environmental Earth Summit which was a follow up to the conference in Rio de Janeiro 5 years earlier."

Scott, 1998[1].

Kyoto protocol was signed on 1997 and since then, many other worldwide conventions have followed[I], some held specifically to deal with the subject of climate

[I] Including the Millennium Declaration and the Millennium Development Goals (2000), the UNFCCC conference (2002)[2] , the World Summit for Sustainable Development (2002), the international conference for renewable energies (2004), and G8 summits at Gleneagles(2005) and Heiligendamm (2007).

change and global warming and others dealt with it as part of a broader agenda. However, the UK government believes that climate change is the greatest long-term challenge facing the world today[3].

The UK[ii] contributes about 2% to global man-made emissions of CO_2, the most important gas contributor to climate change. The global emissions of CO_2 are currently estimated to range between 6.2 and 6.9 billion tonnes carbon per annum[4].

Is climate indeed changing at a faster rate than the policies and technologies do in order to mitigate it? What is the current scenario in the area of greenhouse gases (GHG) emissions in the UK and what is the role of wind energy in the built environment to mitigate CO_2 emissions? How much has been achieved so far, in relation to the use of renewable energy systems, since Kyoto?

According to Scott[1], the scarce advances achieved before and soon after Kyoto, were mainly because multi-national negotiations for the GHG reductions had only served to illustrate the complexity of the climate change scenario, in particular the difficulty of bringing developing countries on board while the industrial nations (especially USA) struggled to prove their interest in putting it on the agenda.

This situation has since, to some extent, changed; public opinion is less divided – largely because of the active role that media has played – there is no doubt now that climate change is caused by the emission of GHGs into the atmosphere and that the single most important GHG is CO_2, which is generated primarily from fossil fuel burning. World leaders are taking the issue of climate change and global warming

---

[ii] Further details: www.defra.gov.uk/environment/statistics/globatmos/index.htm

as central points on their negotiations – some even in their political agendas[III] – as it is reflected in the concluding documents of the recent G8 leaders conventions held at Gleneagles and Heiligendamm in 2005 and 2007, respectively.

At the Gleneagles Summit, the G8 leaders finally agreed that climate change was a serious and long-term challenge caused by human activity, and that urgent action was needed[3]. Two years later, the conclusions of G8 leaders on climate change according to the Chair's Summary[5] were:

> *"Combating climate change is one of the major challenges for mankind and it has the potential to seriously damage our natural environment and the global economy. […]We are convinced that urgent and concerted action is needed and accept our responsibility to show leadership in tackling climate change. […]We will consider seriously the decisions made by the European Union, Canada and Japan which include at least a halving of global emissions by 2050."*

At Heiligendamm, it was also concluded that in order to address the urgent challenge of climate change, it was vital that the major emitting countries agreed on a detailed contribution for a new global framework by the end of 2008 which would contribute to a global agreement under the United Nations Framework Convention on Climate Change (UNFCCC) by 2009.

Despite the fact that there has been a global increase in climate change literacy and that it has helped to raise awareness of the diversity and advantage on

[III] A clear example of the role that media plays on persuading public opinion on the subject of climate change and global warming was evident with the release of politician Al Gore's book and later film documentary: An inconvenient truth[6].

technological, cultural and behavioural choices on energy related issues, it is worthy to highlight that almost a decade after Kyoto, the annual total GHG emissions arising from the global energy supply sector continue to increase in spite of the greater deployment of low and zero carbon technologies, particularly those utilising renewable energy[7].

The Kyoto Protocol expires in 2012 and without the near-term introduction of supportive and effective policy actions by governments, energy related GHG emissions, mainly from the combustion of fossil fuels, are projected to rise around 50% from 26.1 $GtCO_2$-eq in 2004 to almost 40 $GtCO_2$-eq by 2030, according to the recent Intergovernmental Panel on Climate Change (IPCC) report and its findings on energy supply[7].

It is clear that it is no longer environmentally sustainable to continue to extract and combust the world's rich endowment of oil, coal, and natural gas at the current rate.

In the UK, projected climate changes could have a considerable impact on the thermal performance of the built environment and on measures implemented to improve such performance[8]. Fortunately, the UK has already taken significant steps to meet this challenge. The Government has implemented some policies, such as the Climate Change Levy and agreements, Renewables Obligation and the Energy Efficiency Commitment[3]. Those measures are indeed an obligation, if the UK is to meet the goal declared in the 2003 Energy White Paper of reducing carbon dioxide emissions by 60% by 2050.

According to the IPCC[7], the use of a wide range of available – and potential – low and zero carbon technologies together with improved power generation plant efficiency and fuel switching from coal to gas, could represent a total mitigation potential of 2.0 - 4.2 GtCO_2-eq /yr by the year 2030. That is for the electricity sector alone.

Moreover, according to the European Wind Energy Association (EWEA)[9], if the European Union does not increase research and development (R&D) funding to renewable energy, in particular to wind energy, at least to a par with that given to conventional technologies, it will fail to meet its targets for renewable energy and GHG emissions, and to reduce its dependency on energy imports.

The electricity, building and industry sectors are beginning to become more proactive and help governments make the transition happen. Sustainable energy systems emerging as a result of government, business and private interactions should not be selected on cost and GHG mitigation potential alone but also on their other co-benefits.

There is no single economic technical solution to reduce GHG emissions, but there is good mitigation potential available based on several zero or low carbon commercial options that are currently available or under research development.

The IPCC considers that the future choice of energy supply technologies will depend on the timing of successful developments for advanced nuclear, advanced coal and gas, and second generation renewable energy technologies. This stresses the importance to allocate sufficient resources to R&D on those technologies.

Although there is no silver bullet to combat climate change and global warming, renewable energy technologies[10] in general and in particular wind energy[9],[11] can significantly contribute to security of energy supply, reduce fuel imports and dependency, reduce GHG emissions and improve environmental protection.

One of the technologies being considered to reduce GHG emissions is micro-wind turbines designed to be mounted on new and existing homes for small-scale electricity generation. Wind energy is a significant and powerful resource. It is safe, clean, abundant and indigenous. Wind energy can and must be part of the solution to the global energy challenge.

However, there is little experience of the operation of such turbines mounted on domestic buildings in urban environments and hence little objective data about their actual performance in terms of power generation, service life and maintenance requirements. This has led to concerns that, in some environments, the installation of micro-wind turbines on housing could increase carbon emissions rather than reduce them.

In order to ascertain the reduction or increase in CO_2 emissions, studies that monitor the output of Building-Mounted/Integrated Wind Turbines (BUWTs) installations have to be carried out. These studies will need to collect data for at least one year to have an indication of the annual output, although the collection of data for longer periods would be statistically, more significant.

With enough data to process, these types of studies can give an indication of the energy balance of such turbines, by comparing estimates of the environmental impact of their manufacture, installation and maintenance with the impact saved by the electricity that the turbine could generate.

It is clear that the siting of a wind turbine is a fundamental element on the equation if the wind turbine project is to be carbon neutral. Therefore, studies -like this one- aiming to better understand the assessment of wind resource are of vital importance. Usually, for small wind turbine projects, the assessment of the wind resource are normally based on wind speed databases, which do not take account of the surface roughness effects of the buildings in an urban environment. The use of such data can give an over optimistic estimate of the power output. This study investigates CFD as a method for estimating the wind resource (chapter five), along with data obtained from a wind tunnel study (chapter four) and additionally explores the importance of the correct near wall treatment for roughness simulation.

In summary, although some has been achieved, it is clear that the world is not on course to achieve a sustainable energy future. According to the IPCC, the global energy supply will continue to be dominated by fossil fuels for several decades. To reduce the resultant GHG emissions will require a transition to zero and low carbon technologies.

This can happen over time as business opportunities and co-benefits are identified. However, to obtain more rapid technology deployment will require policy interventions concerning the complex and inter-related issues of security of energy supply, removal of structural advantages for fossil fuels, minimising related environmental impacts and achieving the goals for sustainable development.

1.2.2 Energy and the built environment

Servicing buildings accounts for around half of the UK's total energy consumption and the need to reduce the consumption of fossil fuels is central to good design[12].

Given the climate change and global warming scenario discussed in 1.2.1, it is clear that buildings should now be designed with the target of zero GHG emissions. Measures to reduce GHG emissions from buildings fall at least into one of the next three categories: reducing energy consumption and embodied energy in buildings, switching to low carbon fuels including a higher share of renewable energy and/or controlling the emissions of non-CO_2 GHG gases [13].

A key conclusion from the Fourth IPCC's Assessment Report[13] on residential and commercial buildings is that substantial reductions in CO_2 emissions from energy use in buildings can be achieved over the coming years, using mature technologies for energy efficiency that already exist widely and that have been successfully used. Moreover, Boardman et. al[14] are certain that the UK residential sector can deliver a 60% reduction in carbon emissions by 2050, in line with the targets outlined in UK Government's 2003 Energy White Paper by – among others – improving building fabric and using building-integrated technologies.

However, due to the long lifetime of buildings and their equipment, as well as the strong and numerous market barriers prevailing in this sector, many buildings do not apply these basic technologies to the level life-cycle cost minimisation would warrant.

While the IPCC report[13] acknowledges that there is a broad array of accessible and cost effective technologies and know-how that can abate GHG emissions in buildings to a significant extent, the report only includes passive solar design, high-efficiency lighting and appliances, highly efficient ventilation and cooling systems, solar water heaters, insulation materials and techniques, high-reflectivity building materials, and multiple glazing.

This opens many possibilities for building integrated renewable energy systems and reinforces the importance to further continue the research on wind energy generation in urban environments.

Moreover, according to the findings of the International conference for renewable energies held in Bonn[10] in 2004, an increased use of renewable energies in buildings offers significant economic potential.

Solar collectors are on top of the list of integration into the built environment amongst renewable energy technologies. Nevertheless, energy generation potential and technical feasibility of siting wind turbines in the built environment was thoroughly assessed in the final report to the Carbon Trust regarding the feasibility of Building-Mounted/Integrated Wind Turbines (BUWTs)[15] and it was concluded that wind energy could make a significant contribution to the energy requirements in the built environment and that a more detailed evaluation is justified.

However, as mentioned before, there are some drawbacks that must be taken into consideration when planning a BUWT project. Firstly, it is the concern on the energy balance during the turbine's life cycle[IV], this is, whether the small wind turbine can actually generate as much energy to offset the CO_2 emissions generated during its life cycle. Secondly, the visual impact of the wind turbines is an issue that should not be overlooked as social barriers have been also

[IV] A life cycle assessment of a wind turbine can be carried out by making some assumptions regarding its manufacture, transport and maintenance. A good example is the work published by the BRE[16].

documented to be of great importance for the deployment of renewable energy technologies[17].

While buildings have the largest share of cost-effective opportunities for GHG mitigation, achieving a lower carbon future will require very significant efforts to enhance programs and policies for energy efficiency in buildings and low carbon energy sources well beyond what is happening today[13].

A gap in sustainable architectural design

It has been discussed that the building sector is currently forward-looking in the challenge of transforming our energy systems to create lower carbon economies. But this is by no means a new trend, the building design professionals have considered environmental aspects on their designs many decades before the climate change and global warming challenges started to be evident and building regulations on energy savings were put into place.

In fact, the term bioclimatic was used for the first time in the early 1950s by Victor Olgyay and fully explained in his book 'Design with climate' in 1963[18]. He developed a bioclimatic chart, which relates climatic data to thermal comfort limits, to identify design strategies. After that, the scientific knowledge accumulated since the 1970s, with the definition of guidelines, simulation tools, case studies, among others, has improved and it has consolidated a technical field in energy efficiency[19] that has impacted the architectural practice in many aspects.

We can appreciate one of those impacts in the abundance of terms to define a design that is taking environmental concepts into consideration within the architectural discipline. These terms include: eco-construction, energy efficient design, bioclimatic architecture, green architecture, low energy design, ecological design, zero carbon emissions design, co-housing, environmental architecture and perhaps the widest concept of them all: sustainable architecture[20] [21],[20],[22],[23],[24].

Many authors have made attempts to tackle the ambiguous issue of sustainable architecture. Susan Hagan[25] accurately expresses:

> *"When applied to architecture, the term 'sustainable' currently refers to environmental sustainability. Swept up in the concern for environment, however, is an accompanying concern for social sustainability, as this implies public health and a fairer distribution of physical resources and physical risk. Economic sustainability, in the sense of value for money or return on investment, is also implicit within environmental sustainability, and increasingly easy to demonstrate with built examples."*

The diversity is not only found in the definitions of concepts. The architectural products and design approaches present a wide spectrum as well, from the traditional vernacular house to the ultimate high-tech high-rise all-gadgets-included building.

In one end of this spectrum we find mostly small size architectural practices and designers that rely on more qualitative methods to approach the environmental issues within the building design and those methods are often based on common sense and experience[19]. Although, the role of experience has been detected as a valuable tool in the design process[19],[26], in some cases the use of intuition and experience alone, can lead to the use of 'smart arrows'[V] drawn in a section that are supposed to indicate – and validate – that the environmental and indoor comfort conditions have been not only addressed but thoroughly studied.

At the other end of the spectrum of environmental design is the information produced by engineers, which provides with the theoretical background – the hard science – by taking all aspects of the building physics into consideration. In spite of the accuracy of the methods used and the complexity involved in explaining thermal, acoustic and wind behaviours on a numerical basis, most of the studies lack of the fundamental issue of translating those findings into feasible architectural proposals.

This is a shame, as ultimately the goal of those investigations is indeed to transform mainstream society, and it is only by binding the concepts to a tangible reality – in this case the building – that a reduction of energy consumption and a mitigation of GHG can be achieved.

But what we see in the middle section of the spectrum is a number of medium size architectural practices that use simulation tools to estimate and make predictions on building performance and energy systems. The majority of that work is produced as a result of a research process within the practice. The presentations of the firm's projects are usually accompanied by several colourful graphs produced with the simulation tools to complement the smart arrows in the sections, with little or none further explanation on how those graphs were obtained or how the simulation process itself was approached.

[V] The term smart is used here to describe the subjective and uninformed nature of those arrows to illustrate the best performance of a natural ventilation system, the thermal capacities of the building envelope and the good performance of a shading device to prevent overheating.

However, the inclusion of simulation tools into the design process in practice must involve a specialist consultant[VI]. The architect alone usually does not have the skills required to deal with the quantity and complexity of information needed, therefore, environmentally sound design usually requires the designer to tackle the design problem from a multidisciplinary point of view[19],[26].

Moreover, through designing and operating new buildings as complete systems, the largest savings in energy use can be achieved[VII]. Realising these savings require a global, interdisciplinary approach, known generally as integrated design process[27], that allows a rationalisation of all aspects of the project through a combination of traditional and innovative methods, involving architects, engineers, contractors and clients, with full consideration of opportunities for passively reducing building energy demands.

It is unquestionable that the building profession must contribute to social and environmental sustainability and now challenges architects beyond the limits of an autonomous brief[28] and encourages them to work across-disciplines. Some authors have raised the questions about the role of the architect within interdisciplinary teams[19],[26],[27],[29]. Amongst many roles that the architect can play, there is the central task of building design.

Szokolay[30] acknowledges that although design dominates the architectural ethos, it is much more than just the look of the product and that special importance must be

[VI] Augenbroe[31], elaborates further on the use of the simulation tools by the design team, suggesting that the design team needs to be more than a tool user, and become a central design manager, acting as a coordinating agent for domain experts.
[VII] According to the IPCC[13] report on residential and commercial buildings, 75% or higher.

given to the sustainability dimension, how the building works and how it uses resources namely: site, energy, materials and wastes.

Furthermore, sustainability objectives[VIII] should be included in the outline brief as they must pervade the whole design as part of a holistic process[32]. This implies a close collaboration with expert consultants, Battle and McCarthy[29], express their thoughts regarding this relationship:

> *"As engineers working with architects, we feel that the boundaries between our disciplines are fading"*

They suggest that what is often really missing in the discussion between the engineer and the architect is the fundamental issues associated with building physics, indispensable for the understanding of energy efficiency.

However, in order to initiate a discussion between architect and engineer regarding building physics they are required to build a new type of bridge, one that conciliates the different discourses, translates on the agreement and fulfilment of the expectations of all stakeholders and reduces the gap between environmental design and architectural quality.

In summary, each side of the spectrum is accompanied by an easily identifiable design approach, i.e. qualitative vs. quantitative.

[VIII] Further information is available from the following:
The Sustainable Development Comission at: www.sd-comission.gov.uk and www.sustainable-development.gov.uk www.dti.gov.uk/construction/sustain/bql/index

The qualitative design approach raises serious questions, since using simulation tools or smart arrows for client persuasion is a delicate subject, especially when one of the main barriers to overcome is the difficulty of introducing environmental issues for the risk – mainly financial – that innovative solutions in building design involve[33]. Client disappointment as a result of the failure to meet estimated energy savings, energy generation and thermal comfort; can lead to the disrepute of energy efficient and renewable technologies, in particular, and to mistrust on the environmental design approach overall.

On the other hand, the quantitative approach often fails in appealing to the architecture community which plays a fundamental role on the aesthetical integration of environmentally sound and innovative technologies. This method can increase the gap between research and design practice by alienating the non-specialist, increasing the possibility of valuable information and findings to be overlooked.

Nevertheless, a fresh approach to the problem – an approach that investigates the conflict between smart arrows and hard science – can contribute to the resolution of these issues; an integrated design process that puts together architectural and environmental issues at every stage.

To address this, the approach of this research was to work on a proposed architectural project in Nottingham, UK. The brief included the design of a building complex that integrated wind turbines[IX]. The site and the neighbouring surroundings were considered, and different building configurations were suggested, as usually

[IX] The site and design brief are thoroughly explained in chapter three.

done at a design stage[x]. From the literature review conducted, the methods to study wind flows around buildings were selected for this research. Wind patterns for the different building configurations were experimentally investigated with the use of a boundary layer wind tunnel and CFD simulations. The experimental and numerical results obtained, were then compared and discussed; from this, it was possible to draw some conclusions on the correct use of these tools and places to locate wind turbines were suggested. Finally, the CO_2 savings for the proposed configuration were calculated.

It is evident, that the design process becomes more complex because the designer needs to get involved and deal with more variables: knowledge from different disciplines, new modelling tools and new benchmarks. The method to achieve the design is, thus, never straightforward; and as Trebilcock[26] openly questions:

> *"Using [simulation] tools, benchmarks and indicators does not assure a sustainable building as a result by itself, it still needs an integration process that occurs somewhere: in the architect's brain? As a result of collaborative work?"*

1.3 The scope

The idea of doing a one-man research that aims to tackle the problem of building aerodynamics to integrate wind turbines and sustainable architectural design, covering in depth the requirements of both disciplines, might seem an unpromising

[x] For this research, nine different building configurations were proposed by changing parameters like building orientation and distance between buildings. These configurations are also explained in chapter three and are referred to as "cases", throughout this thesis.

and strenuous one. This is especially true when interdisciplinary work looks like the evident answer for such a problem.

Under this light it is only fair to enquire what the expectations of such a research are. This investigation is based on the grounds discussed on section 1.2.2 namely that, on one hand, the published quantitative research on building aerodynamics and more in particular on building integrated wind turbines that has been carried out with a strong engineering approach [34] ; [35],[36],[37], fails to deliver their conclusions into a suitable architectural solution. And that, on the other hand, the qualitative or descriptive research on the same topics lacks of the sufficient scientific background that can support further design decisions[38],[39],[40].

There are several research studies at both ends of the spectrum discussed above. Nevertheless, the assessment of wind regimes in urban areas for the location of building integrated wind turbines is still detected as a R&D priority[9],[15]. Hence, this investigation is targeting to the gap in the middle; by making an experimental and numerical assessment of the wind regime in urban environments and translating that findings into building design, in an attempt to link both approaches by working across disciplines, using different methodologies and by synthesising technical and theoretical topics that are often blurred on a multidisciplinary work.

1.3.1 Research questions

*"The main practical question is: **How** these noble **ideas** can be translated into **actions** at the level of everyday reality?"*

Szokolay, 2004[30]

The scope and the limitations of this study are defined by the following research questions, hypotheses and objectives. The chapters where they are addressed are also indicated.

- What is the role of the architect on a design team undertaking a sustainable design? (Chapter eight)
- Up to what extent does the architect need to gain knowledge of building physics and its simulation tools? (Chapter six)
- When designing a sustainable building, what is the potential of building integrated wind turbines for energy generation? (Chapter two)
- Which simulation tools and/or experimental procedures offer the best possibilities to investigate building aerodynamics for possible wind turbines integration? (Chapter three)
- What are the minimum standards to be observed for a reliable simulation on building aerodynamics? (Chapter six)
- How are the results obtained with the simulations translated into the architectural design? (Chapter seven)

1.3.2 Research hypotheses

- There is plenty of qualitative and quantitative research regarding building aerodynamics for integrating wind turbines into buildings that requires an effective juxtaposition to achieve a holistic sustainable building design (chapter two).
- Experimental tools like boundary layer wind tunnels (chapter four) along with CFD simulations (chapter six) are accepted methods to gather reliable information on the performance of building integrated wind turbines.
- CFD simulation techniques can be applied as a visualization and predictive tool during the design stage (chapter six).
- The use of an integrated design process that puts together architectural and environmental issues at every stage could lead to better building-integrated wind energy technologies (chapter seven).

1.3.3 Research objectives

- To discuss the different strategies, methods and approaches commonly used for environmental design in architecture, as well as their respective strengths and weaknesses (chapters one and three).
- To find out the present and potential role of wind turbines, in urban environments as part of a sustainable development design (chapter two).
- To identify the key factors that influence the performance of wind turbines in urban environments (chapter two).
- To evaluate the ability of wind tunnel (chapter four) and CFD simulations (chapter five) to accurately describe wind flow around buildings in urban environments in order to evaluate possible locations for building integrated wind turbines.

- To identify the minimum standards and requirements that an architect or environmental designer should observe for a reliable simulation of wind flows around buildings in urban environments using CFD as a design and visualisation tool (chapter six).
- To gather recommendations on the suitable siting of small wind turbines in the built environment (chapter seven).

1.4 The structure

At the beginning of each chapter, an informative box is presented; it provides a quick overview of the main aspects to be dealt within the chapter. Similarly, at the end of each chapter a summary box is included; it illustrates the key findings and concepts within each chapter. Additionally, a list of the references cited is provided at the end of every chapter. The table of contents and the lists of figures, tables, symbols and acronyms are presented at the beginning of the thesis, and they cover all the chapters of the thesis. Appendices are provided at the end of the thesis and they comprise the extended data used for the experimental procedures and useful but additional information found with the literature review.

This thesis is divided into three sections, as illustrated in Box 1.2.

BOX 1.2 thesis structure

S1 Context and Scope In which the relevance, structure and methods of the study are explained.	1	introduction
	2	literature review
	3	research approach
S2 Experimental In which quantitative data are analysed and discussed.	4	wind tunnel experiment
	5	CFD modelling
	6	comparison and discussion of results
S3 Final remarks In which numerical findings are related to architectural design and conclusion are drawn.	7	architectural integration
	8	conclusions and further work

First section – Context and scope

The first section includes the more theoretical chapters 1-3 (introduction, literature review and methodology). In this section, a framework is developed for understanding the context of climate change and the role that wind energy in urban environments play to mitigate it. Limitations and scope of the study are presented in this section as well.

In this first chapter, the thesis structure is presented. The argument and the relevance of the study are presented by discussing topics like contribution of the built environment to climate change and what has been the response of the architectural and design professionals to tackle that problem. The potential role of renewable energy systems and particularly of wind energy in urban environments as a means to reduce CO_2 emissions is introduced. Finally, the scope of the study is made by defining the research questions and the objectives for this research.

The second chapter continues the discussion of environmental design by providing an overview of the main issues on sustainable development and how wind energy can contribute to achieve it. The literature review conducted for this research is presented in this chapter, developing a broad understanding of wind engineering theory and wind turbines for the use in urban environments. It also presents a summary of the work that has been done to date in the field of building integrated wind turbines and how it connects with this research.

In the third chapter, the methodological approach adopted for this research is explained. It describes the common methods to study wind flows around buildings and provides a rationale of the chosen methods for this research. Additionally, a site analysis where the project located is made and the nature of the nine cases used for experimental purposes is fully explained and justified.

Second section - Experimental

The second section is the description and discussion of the experimental procedure. It contains the wind tunnel experimental chapter and computational fluid dynamics modelling chapter. In a following chapter, the results are presented and discussed.

This section starts with chapter four. It includes a quick synopsis of how wind tunnels have been previously used and the requirements that must to be taken into consideration for wind tunnel modelling are explained. A description on how the wind tunnel experiment was set up for this research is made, along with the explanation of the measuring and data acquisition equipment involved. Finally, the results of wind velocity measurements between the studied buildings for the nine cases are presented. These experimental results are used to suggest the location of small wind turbines.

Chapter five presents the modelling of wind flow around the studied building complex by using CFD techniques. The same nine cases modelled in the wind tunnel and described in chapter four were modelled using CFD commercial software FLUENT and are analysed in this chapter. A brief explanation on the use of CFD for modelling wind flow around buildings is given. Also, an explanation on how the CFD modelling was set up is given and, finally, the results of the CFD modelling are presented; from these results, suggestions on the location of small wind turbines are made.

The experimental procedure section finishes with chapter six. In this chapter, results of wind tunnel and CFD modelling are compared and discussed against a theoretical background. Recommendations on the selection of wind turbine types and their siting between or over the buildings are drawn from the experimental findings and are presented in this chapter.

Third section – Final remarks

Finally, the third section deals with the final remarks and conclusions; in it, some architectural examples of building integrated wind turbines are reviewed and the main findings and conclusions are presented.

Chapter seven is part of the last section of the thesis, where final remarks and conclusions are presented. In this chapter the connection between wind tunnel and CFD modelling to investigate possible location for wind turbines in buildings and architectural design is made by presenting the case with more potential for wind turbine integration. This case is used to calculate the energy produced by the wind turbine located at the higher velocity ratios and the CO_2 savings for the system.

The last chapter includes the final thoughts on this study. A summary of the findings is presented, along with some concluding thoughts on the lessons learned and recommendations for further work.

1.5 Chapter One - key concepts

BOX 1.3 chapter one key concepts

a) On climate change

- Combating climate change is one of the major challenges for mankind and it has the potential to seriously damage our natural environment and the global economy.
- UK has established a goal of reducing CO2 emissions by 60% by 2050.
- There is good mitigation potential available based on several zero or low carbon commercial options.

b) On energy and buildings

- Servicing buildings accounts for around half of the UK's total energy consumption and the need to reduce the consumption of fossil fuels is central to good design.
- Measures to reduce CO2 emissions from buildings include a higher share of renewable energy as part of a holistic environmental design.
- Wind energy could make a significant contribution to the energy requirements in the built environment.
- The building design professionals are considering environmental aspects on their designs, but with very different approaches and there is a gap to be bridged in order to bind the concepts to a tangible reality.
- The architecture community plays a fundamental role on the aesthetical integration of environmentally sound and innovative technologies, in particular building integrated wind turbines.

c) On the scope of this study

- This investigation is an attempt to promote a link between the different environmental design approaches by working across disciplines, using different methodologies and by synthesising technical and theoretical concepts.

1.6 Chapter One - reference list

1. Scott, A. (1998) *Dimensions of sustainability : architecture form, technology, environment, culture*, London ;, New York : E & FN Spon.

2. UNFCCC (2003) Report of the conference of the parties. New Delhi, India. *Eighth session Proceedings.* FCCC/CP/2002-7. 28pp. Available at: www.UNFCCC.org.

3. Secretary of State for the Environment Food and Rural Affairs. UK. (2006) *Climate Change. The UK Programme 2006.* 202pp.

4. DEFRA and National Statistics (2005) The environment in your pocket 2005. *Key facts and figures on the environment of the United Kingdom.* 72 pp.

5. Anonymous (2007) G8 Summit 2007. Heiligendamm, Germany. *Chair's summary.* 8pp. Available at: <http://www.g-8.de>. Accessed 18/06/07

6. Gore, A. (2006) An inconvenient truth. The planetary emergency of global warming and what we can do about it. 325 pp Great Britain: Bloomsbury Publishing Plc.

7. IPCC (2007) Fourth Assessment Report, Working Group III. *Chapter 4 Energy Supply. International Panel for Climate Change.* Final Draft. 112pp. Available on line at: <www.mnp.nl/ipcc/index.html>. Accessed: 20/06/07

8. Gaterell, M.R. and McEvoy, M.E. (2005) The impact of climate change uncertainties on the performance of energy efficiency measures applied to dwellings. *Energy and Buildings* **37** (9):982-995.

9. EWEA (2005) Prioritising wind energy research. *Strategic Research Agenda of the Wind Energy Sector 2005-2020.* European Wind Energy Association. 60 pp

10. Anonymous (2004) Renewables 2004. International conference for renewable energies. Bonn, Germany. *Conference Report Outcomes & Documentation – Political Declaration / International Action Programme / Policy Recommendations for Renewable Energies.* 54pp. Available at: <http://www.renewables2004.de/pdf/conference_report.pdf>. Accessed 12/07/05

11. GWEC (2005) Global Wind 2005 Report. *Global Wind Energy Council* 52 pp. Available at:<http://www.gwec.net/fileadmin/documents/Publications/Global_WindPower_05_Report.pdf> Accessed: 07/06/06

12. CABE (2006) Better public building. *Commission for Architecture and the Built Environment.* 24 pp

13. IPCC (2007) Fourth Assessment Report, Working Group III. *Chapter 6 Residential and Commercial Buildings. International Panel for Climate Change.* Final Draft. 91pp. Available on line at: <www.mnp.nl/ipcc/index.html>. Accessed: 20/06/07

14. Boardman, B., Darby, S., Killip, G., Hinnells, M., Jardine, C.N., Palmer, J. and Sinden, G. (2005) *40% house*, Oxford : Environmental Change Institute, University of Oxford.

15. Dutton, A.G., Halliday, J.A. and Blanch, M.J. (2005) The feasibility of Building-Mounted/Integrated Wind Turbines (BUWTs): Achieving their potential for carbon emission reductions. Final report to the the Carbon Trust (2002-07-028-1-6). 109 pp.

16. Phillips, R., Blackmore, P., Blackmore, J., Clift, M., Aguiló-Rullin, A. and Pester, S. (2007) Micro-wind Turbines in Urban Environments: An assessment. *Building Research Establishment* **BRE FB17**, 47 pp

17. Wilkins Gill and Royal Institute of International Affairs Sustainable Development Programme (2002) Technology transfer for renewable energy : overcoming barriers in developing countries. London ; London : Royal Institute of International Affairs and Earthscan. 1853837539. Leeds ; Newcastle ; Oxford ; Trinity College Dublin.

18. Olgyay, V. (1963) *Design with climate. Bioclimatic approach to architecture regionalism.*, New Jersey: Princeton University.

19. Maciel, A.A., Ford, B. and Lamberts, R. (2007) Main influences on the design philosophy and knowledge basis to bioclimatic integration into architectural design--The example of best practices. *Building and Environment* **42** (10):3762-3773.

20. Porter, T. (2004) ARCHISPEAK. An illustrated guide to architectural terms. First edn, 218 pp. London; New York: Spoon press.

21. Hu, D. and Wang, R. (1998) Exploring eco-construction for local sustainability: An eco-village case study in China. *Ecological Engineering* **11** (1-4):167-176.

22. Cunningham, D. and Goodbun, J. (2006) Marx, architecture and modernity. *The Journal of Architecture* **11** (2):169-185.

23. Haraldsson, H. and Sevensson, M.G.E. (2000) Is Ecological living sustainable? - a case study from two Swedish villages in South Sweden. *Proceedings of the 18th International Conference of the System Dynamics Society*

24. Kirby, A. (2003) Redefining social and environmental relations at the ecovillage at Ithaca: A case study. *Journal of Environmental Psychology* **23** (3):323-332.

25. Hagan, S. (2001) *Taking shape : a new contract between architecture and nature*, Oxford ;, Boston : Architectural Press.

26. Trebilcock, M. (2007) The integration of environmental sustainability issues in the design process of contemporary architectural practice. *PhD. Dissertation. University of Nottingham, UK.* 337pp

27. Gauzin-Müller, D. and Favet, N. (2002) Sustainable architecture and urbanism : concepts, technologies, examples. Basel ; Berlin ; Boston, MA : Birkhäuser : Momenta. 3764366591. Edinburgh ; Leeds ; Sheffield.

28. Rogers, R. (1997) Cities for a small planet. Gumuchdjian, P., (Ed.) 180pp Great Britain: Faber and Faber Ltd.

29. McCarthy, C. and Battle, G. (1998) The Need and Drive for Sustainable Architecture and Urban Design. In: Scott, A., (Ed.) *Dimensions of sustainability : architecture form, technology, environment, culture*, pp. 55-63. London ;, New York : E & FN Spon.]

30. Szokolay, S. (2004) *Introduction to architectural science: The basis of sustainable design*, Oxford : Architectural Press.

31. Augenbroe, G. (2004) Trends in building simulation. In: Malkawi, A.M. and Augenbroe, G., (Eds.) *Advanced Building Simulation*, pp. 4-24. New York, UK: Spon Press]

32. Eley, J. (2003) Creating excellent buildings. A guide for clients. CABE, AMA and DLC, (Eds.) First edn, London:

33. Intrachooto, S. and Horayangkura, V. (2007) Energy efficient innovation: Overcoming financial barriers. *Building and Environment* **42** (2):599-604.

34. Guirguis, N.M., Hanna, G.B., Kotkata, M.F. and Gad, I.A. (1998) An investigation of building/wind interaction. *Renewable Energy* **15** (1-4):383-386.

35. Lawson, T.V. (2001) *Building Aerodynamics*, London : Imperial College Press.

36. Mertens, S. (2006) *Wind energy in the built environment*. Multi-Science. Essex, UK. 169 p.

37. van Bussel, G. and Mertens, S. (2005) Small wind turbines for the built environment. N´aprstek, J. and Fischer, C. (eds). *The Fourth European & African Conference on Wind Engineering* Prague: ITAM AS CR.

38. Bill Dunster architects ZEDfactory Ltd. (2006) From A to ZED. Realising Zero Energy (fossil) Developments. Second edn, CD-ROM format. 186pp. UK: The housing corporation. Innovation & Best Practice Program (IGP).

39. Blanch M.J. (2002) Wind Energy Technologies for use in the Built Environment. *Wind Engineering* **26** (3):125-143.

40. Smith, P.F. (2003) *Sustainability at the cutting edge : emerging technologies for low energy buildings*, Jordan Hill, Oxford : Architectural Press.

2 literature review

2.1 Introduction

This chapter elaborates on the important role of the architect/designer to achieve sustainable development and sustainable communities. The concept of zero and low carbon emissions developments (ZED/LED) is introduced and the use of renewable energy systems, in particular wind energy is detected as a key element for reducing CO_2 emissions. Examples of ZED and LED are presented along with the latest developments in building integrated wind turbines.

2.1.1 Chapter overview

This chapter is divided in four main sections (Box 2.1). Section 2.2 establishes the background for the use of wind turbines in the built environment by examining examples of low carbon emissions developments. Next, in section 2.3 the role of renewables and particularly of wind energy as part of a sustainable development agenda is discussed. Later, sections 2.4 and 2.5 provide the basic concepts of wind energy and wind turbines. Differences between large scale wind turbines, building mounted and building integrated wind turbines (BUWTs) are pointed out. Finally, the key points of the chapter are summarised and presented in section 2.6.

BOX 2.1 chapter two overview

2.2 The background: Sustainable development	The link between green building design and sustainable urban design is discussed. Examples of low carbon emissions developments are shown.
2.3 The rol of renewables	Where the benefits of renewables, barriers and incentives for their implementation in building design are explored.
2.4 The nature of wind	A review of the general characteristics of wind flows around buildings and how to convert the energy of wind into electric power.
2.5 The basics of wind turbines	A brief explanation of the functioning of wind turbines and the characteristics for their use in urban environments.
2.6 The key points	Summarised in Box 2.3

2.2 Sustainable development

If indeed cities only reflect the values, commitment and resolve of the societies which they contain, it is no surprise then that most recent transformations of cities reflect society's commitment to the pursuit of personal wealth.

Rogers, R. (1997)[1]

In the previous chapter the ambiguous concept of sustainable architecture was introduced. It was explained that it usually refers to environmental design in architecture. Similarly, the concept of sustainable development, in its broad sense, stresses the idea of ensuring future generations to have the same range of options and resources that we have today[2], namely; quality of water, food, energy, non-renewable resources, housing, communications and transportation and the principal: resources to maintain desired environmental standards and quality of life.

In practice, however, the term is used in the limited sense of including environmental policies and indicators in the planning of cities and urban developments. Subsequently, urban planning, design methods and indicators should consider the regional context from the start to achieve sustainability.

Sustainability for cities also implies that buildings and landscapes are designed to be resource neutral, by providing systems services to reduce their net environmental impact[3]. In this sense, it is clear that energy services are fundamental requirements needed to achieve sustainable development. It is expected that the development and diffusion of renewable energy resources and technologies will help realise important economic, environmental, and social objectives in the early decades of the 21st century. Renewable energies are a critical element for achieving sustainable development[4].

The design of the building itself is then of critical importance in achieving urban sustainability. In the next sections, the relation between building design and urban design will be briefly discussed.

2.2.1 The stand alone ‘green’ building

There are some examples of self-sufficient buildings that are sometimes referred to as the engineered ideal, but that are too often at odds with the sustainable development design ideal of a self-sufficient town or region. The fact that these buildings can generate as much energy as they consume allows them to be – literally– stand alone. Most of the architectural practices that produced the examples of BUWTs, illustrated in chapter seven, portray to different degrees how the proposed building is interacting with its surroundings. Even so, information regarding the urban context is missing during the simulations (if any) of the wind regime.

As said by Dunham-Jones[5], buildings that stand alone, also stand apart from the city, people and transport. A vast amount of the contemporary buildings regarded as models of self-sufficiency have to be driven to. Stand alone buildings tend to locate in either remote sites and or on low-density ex-urban fringes where they can control access to sunlight, wind and soil.

It is expected that a building design that relies exclusively on technical solutions, to making it more energy efficient and/or energy sufficient but gives little attention to an urban design fix, is likely to offset those valuable gains by an inefficient lower density land use patterns and an increased need for transportation to and from the building. This is why; the cases modelled in this thesis considered nine different building layouts and are based in a real urban location. Additionally, the surrounding buildings were modelled both during wind tunnel experiments and CFD simulations.

2.2.2 Zero and low carbon emissions developments

It is crucial to better integrate building and urban design, starting at a community scale, if the profession of architecture wish to truly contribute to a society that is more sustainable economically, socially and environmentally.

A good reference guide for designers to achieve sustainable communities is the General Information Report 53[6] from the Department for Environment, Food and Rural Affairs (DETR). It represents one vision of the factors likely to contribute to sustainability at a community scale, with ideas on the layout, density and transport considerations in settlement design. A development that promotes these concepts in a community is often referred to as zero or low carbon emissions development (ZED / LED).

A desirable feature of a ZED / LED[i] is that, on balance, it should not contribute to any of the greenhouse gases to the atmosphere, especially carbon dioxide (CO_2). To achieve this, and according to the DETR[6], a sustainable community should feature among others:

- Local generation of energy by using wind, solar and biomass
- Interconnection of work, housing, community and leisure facilities

Both of the above were considered on the cases modelled in this thesis. Nevertheless, as it will be discussed in the next sections, one of the most important issues among zero and low carbon emission developments is the energy generation. It is because renewable and energy efficient technologies play a significant role in ZEDs and LEDs that this study makes emphasis on the characterisation of the wind regime in urban environments to integrate small wind turbines in buildings.

The architects' interest in getting involved in ZED and LED is reinforced by the recognition by the UK government that an appropriate uptake of a mix of renewables and energy efficiency measures installed in new build and refurbished buildings, that

---

[i] More information on sustainable communities can be found in the text: 'Sustainable communities: building for the future'[7], here, the UK government underlines the importance of raising the quality of life through increasing prosperity, employment, better public, education and health services. More importantly it relates housing with sustainable development and stresses the point that communities are more than just houses.

respond to the market and local resource conditions is needed[8]. This has resulted in activities led by the local and regional government, including schemes such as Solar for London, (which provides grants for the installation of solar thermal hot water systems), and others, that support zero and low carbon emissions developments such as Sherwood Energy Village (SEV)[II] and Beddington Zero (fossil) Energy Development (BedZED)[III].

Sherwood Energy Village is situated in Ollerton, Nottinghamshire. SEV has won the Enterprising Britain award in 2005 and the Royal Town Planning Institute's Silver Jubilee Cup in 2008. The development has 196 dwellings planned that range from single dwelling bungalows and apartments through to terraced, semi-detached and detached housing, making a total of eleven different housing types architecturally designed to achieve Eco-Homes rate Excellent. SEV opened in October 2006 the business centre or E-Centre. The E-Centre building design includes green roofs, rain water harvesting systems and solar panels and a wind turbine that power lampposts.

Award winning[IV] BedZED[9] is located in Wallington, Surrey. It was designed by Bill Dunster architects and developed by The Peabody Trust and BioRegional Development Group. The scheme in South London is a mixed use development solar urban village that provides 82 affordable dwellings in a mixture of flats, maisonettes and town houses and approximately 2,500m² of workspace/office and community facilities. The 165 hectare site, that was originally a brownfield site, was organised into a layout that obtains maximum benefits from both passive and active

---

[II] More information regarding SEV is available at: http://www.sev.org.uk
[III] More information rearding BedZED and ZED standards is available at: http://www.zedfactory.com/
[IV] Among others, BedZED project was selected as finalist in the world habitat awards (2001) and in the Bremen partnership award (2004). Also in 2004, it was rated as Excellent in BRE's EcoHomes environmental assessment award.

solar gain, by arranging all the dwellings facing south in a series of terraces to minimise heat loss. The energy strategy includes the use of wind driven ventilation with heat recovery, solar heating, rain water collection and a combined heat and power (CHP) plant.

BedZED integrates renewables by using 1,138 PV panels that have been integrated into the building fabric and provide 109kW at peak output. The PV cells have a three-fold function: energy generation, solar shading and external cladding. The rest of the community energy requirements are met with a 125kW dried woodchip fuelled CHP station. All the buildings are wind driven ventilated using one of the most recognisable ZED products; the wind cowls that work with a passive stack principle (Fig. 2.1). Even though, BedZED has been a model development in that it exceeded building regulations, at this time, there are not wind turbines for energy generation in this development.

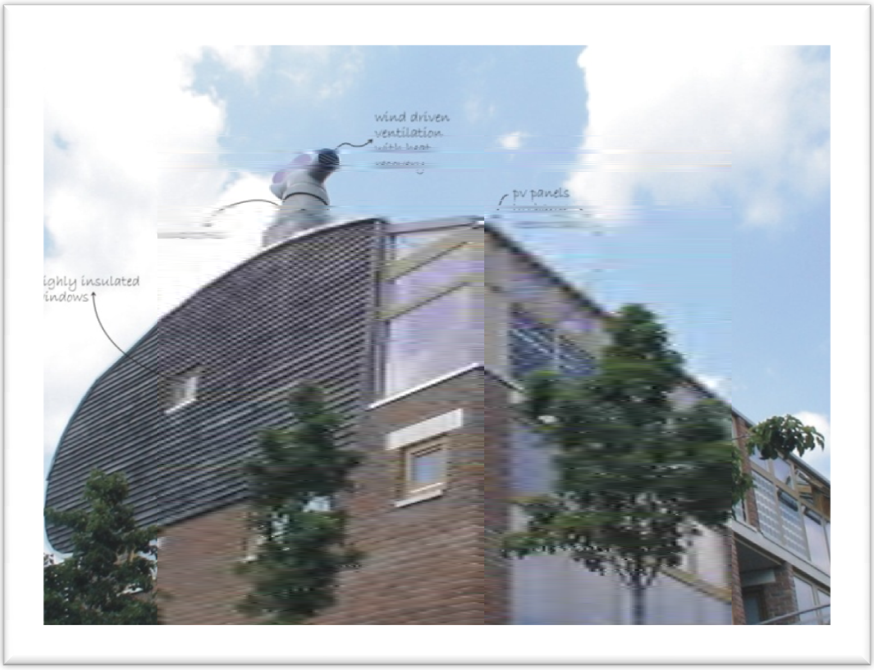


FIG. 2.1. BedZED exterior view. Passive and active design strategies.

These are only two examples of urban ZEDs / LEDs, among a number of other examples of zero and low carbon emissions developments throughout the UK. All of which include bioclimatic concepts such as building orientation and passive solar heating. Nevertheless, not many of the developments include wind energy generation, in this sense Hockerton Housing Project in Southwell, standout with the use of two stand alone wind turbines, each rated at 5kW.

This suggests that even though there is an increased awareness regarding the importance of sustainable development and the role that renewable energy systems play to achieve a ZED / LED; there are still some issues that need to be resolved in order to foster the integration of wind energy into buildings.

2.3 Renewable and energy efficient technologies

"Embracing change carries uncertainty and risk. The power to transform and change both ourselves and the world defines our modern condition. The speed of technological change and above all the speed and breadth of its dissemination provides modern society with its greatest potential power. The challenge we face is to move from a system that exploits technological development for pure profit to one that has sustainable objectives"

Nobel Prize, Octavio Paz, 1991[10]

Although concerns with science and technology have been voiced regarding the various hazards and perceptions of the risk associated with modern technology[V], that have deteriorated the belief that scientific and technological advances are linked to progress, there has been a noticeable transformation in culture on recognising that society creates its environment as much as it adapts to it[11]. Furthermore, there is an increasing tendency to associate building-integrated renewable energy technologies as statements of new dimensions of progress: a manifestation of modernism in architecture[VI].

These somehow contradictory ideas of modernity and progress have lead to some groups to be in opposition to the use of renewable energy systems, both for large scale and building electricity generation; this is especially the case for wind energy.

Additionally, there are some other barriers that hold back the deployment of renewable energy technologies e.g. risk and uncertainty are great barriers to the incorporation of new technologies in architecture[VII]. Other major groups of barriers in architectural projects include financial barriers, technical barriers, psychological barriers, the need to satisfy the client, incompetent professional and conservative building standard/codes[12]. These barriers partially explain why introducing environmentally responsible and innovative solutions in buildings is difficult.

[V] Examples of such concerns include, genetically modified food, computer based surveillance, genetic discrimination, but perhaps there is no better example of how technology has changed the environment like global warming[11].
[VI] Further references on modernity can be found in the work of Octavio Paz[10] and in David Cunningham and Jon Goodbun[13].
[VII] Additionally, for the domestic sector DTI[8] has identify the following barriers in the implementation of building integrated renewable systems: landlord/tenant dilemma, insurance risk, information gap on grid connection and public acceptance.

Nevertheless, considerable efforts have been made to disseminate the important role of renewable energy technologies in contributing towards carbon emissions reductions, thus making evident that environmentally oriented innovations, particularly in the building sector, represent in fact a step forward to a more just and sustainable society. An example of those efforts took place in June 2004 where ministers and Government Representatives from 154 countries gathered in Bonn, Germany, for the International Conference for Renewable Energies. They acknowledged the role of renewable energies and enhanced energy efficiency technologies, to significantly contribute towards a sustainable development worldwide[4].

Moreover, an added benefit of renewable energy sources is that they are usually widely disbursed compared with fossil fuels that are concentrated at individual locations and require distribution. Renewable energy thus, can either be used in a distributed manner or concentrated to meet the higher energy demands of cities and industries.

Renewable energy supply technologies particularly solar, wind, geothermal and biomass are currently small overall contributors to global heat and electricity supply, but are the most rapidly increasing, albeit from a low base. Costs, as well as social and environmental barriers, are restricting this growth. Therefore increased rates of deployment may need supportive government policies and measures[14].

Fortunately, in the UK, promotion of renewable energy systems at a microgeneration scale to achieve ZEDs and LEDs includes a series of support measures, for example, providing grants towards the cost of installing various renewable and energy efficient technologies, including small scale wind turbines through the Low Carbon Buildings Programme[VIII].

The programme, seeks to promote a more holistic approach to reducing carbon emissions from buildings by the use of microgeneration technologies, with a focus on building integrated technologies. In England, more than £880,000 has been granted through the programme for small wind turbines, in a total of 448 committed house hold projects[IX], with 11% granted to the East Midlands in 55 committed projects[15].

Compared with other renewable energy technologies, wind energy is the closest to being competitive with fossil-based systems; this is another important factor encouraging the deployment of urban wind power. Wind generation is a good choice of renewable energy as it is much less expensive in terms of installed cost per kilowatt than PV and therefore makes it an attractive proposition as a building integrated power source[16].

This explains why micro-scale wind turbines in the UK are an emerging technology, driven by advances in device design, increasing energy prices and the financial incentives offered to aid their uptake in buildings[17].

Moreover, the Department of Trade and Industry (DTI)[8] estimates that by 2050, up to 30-40% of UK's electricity generation could be produced by small and microgeneration technologies, including 6% from small wind energy generation. The

---

[VIII] Launched on 1 April 2006, BERRs Low Carbon Buildings Programme Phase 1 will run over three years and replaces the previous DTI Clear Skies and Solar PV grant programmes[15].
[IX] The programme has different schemes for small, medium and large grant applications. Stream 1 is for small applications and it is divided into household and communities grants. More information can be found at: http://www.lowcarbonbuildings.org.uk

UK's housing sector is responsible for around 28% of the UK's CO_2 emissions; hence the 25 million homes in the UK, as well as schools, businesses and other public and private sector buildings can have an important part to play in tackling climate change by generating their own power.

Nevertheless, the first stage of any wind energy project is the available resource base. It is essential to make an assessment to determine the average wind speed available on the site and accordingly the amount of energy that can be generated. To do so, a clear understanding of the wind regime in urban areas, especially around buildings is needed. In the following sections key concepts of the wind characteristics and how they affect the siting of wind turbines in urban areas will be briefly outlined.

2.4 Wind characteristics

Wind is air in motion relative to the earth; it is created by differences in atmospheric pressure that in turn are caused by differences in temperature. Wind moves from areas with high atmospheric pressures to areas where the pressure is lower. It is, in general, three dimensional, with both horizontal and vertical components.

In the layer closest to the Earth, the wind is influenced by friction against the surface. At a certain height this influence is negligible; this undisturbed wind is called the geostrophic wind[18]. The distance from the ground to the geostrophic winds varies depending on weather conditions and surface roughness.

At lower heights, the surface roughness heavily influences the wind. At this height, wind's movement is affected by a friction force acting in the opposite direction. This results in a decrease of wind speed and changes in wind direction, this phenomenon is called wind shear[18].

Due to friction, the wind speed always vanishes at the ground[X]. How rapidly the velocity increases with height depends on the friction against the Earth's surface. On an open plain with low friction, the wind will not be retarded very much and the increase with height will not be very big. Over a surface with many obstacles like a city, the wind will be more retarded so the wind speed will increase more with height. This relation between wind velocity and height is called the wind profile or wind gradient [18]. The profile of wind speed in particular must be taken into account when tall buildings or wind projects are being designed.

The friction against the surface affects the wind, when the wind moves from the sea to land the turbulence increases, so the wind speed close to the ground will decrease. An internal boundary layer (Figure 2.2) is formed between the turbulent wind and the smooth wind at higher levels. The height of the boundary layer will stabilise after a while and remain constant until the surface roughness changes again.

[X] Approximately 50% of the transition in wind speed due to frictional effects takes place in the first 2 m above the surface[20], it is important to keep this in mind for dispersion and pedestrian comfort studies.

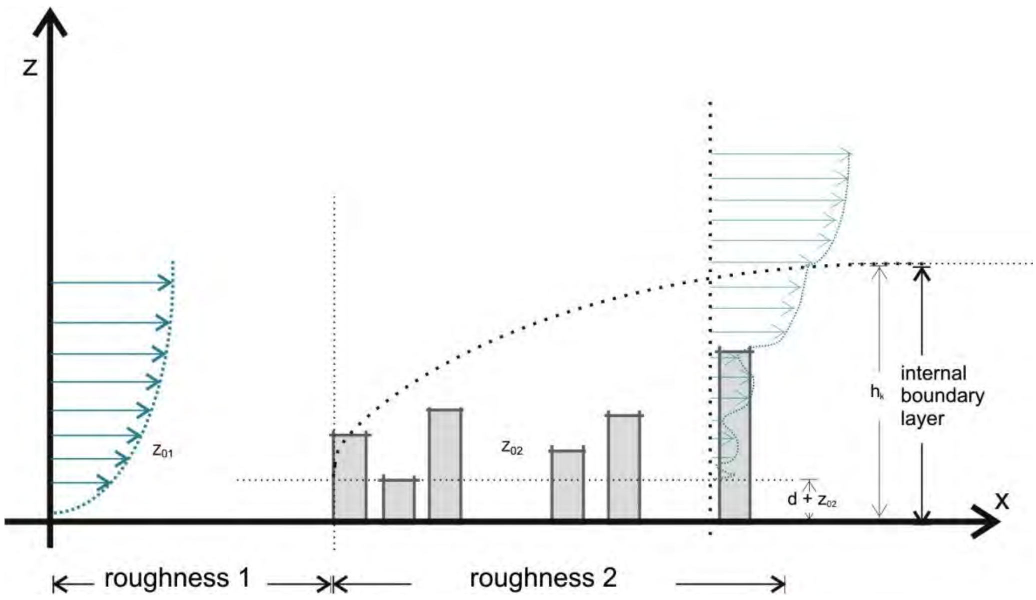


FIG. 2.2. Internal boundary layer. (Adapted from Mertens[19]).

These changes in terrain's roughness are classified into different roughness classes, and are characterized by the aerodynamic roughness parameter z_0, also called the aerodynamic roughness length[21]. This length has nothing to do with the length of the grass or height of buildings; it is a mathematical factor used in the algorithms for calculations of how the terrain influences the wind speed[18], it is therefore, a theoretical height that must be determined from the wind speed profile. For the purposes of design assessment, there are six different roughness categories or classes[18] that define the character of the site, from zero to five, with category zero representing large expanses of water.

Wind is also influenced by different kinds of obstacles, when the air hits an obstacle, air whirls or waves are formed, the air will not move parallel to the ground – what is called laminar wind – it will move in different directions around the prevailing wind direction. The length of those waves varies and the whirls are broken down to successively smaller whirls. When wind is measured, these waves and whirls appear as short variations of wind speed; this receives the name of turbulence.

The porosity of the obstacle also influences turbulence generation; wind hitting a wall will produce strong turbulence behind it compared with wind passing through a hedge or fence. According to a simple rule of the thumb, illustrated in Figure 2.3, an obstacle of height H creates turbulence of 2H, starting at a distance of 2H in front of the obstacle and continuing 20H times behind it[18]. It is clear then that urban environments represent zones of high turbulent flows and as a result the use of wind turbines in the built environment is inherently more complicated than in rural or open spaces.

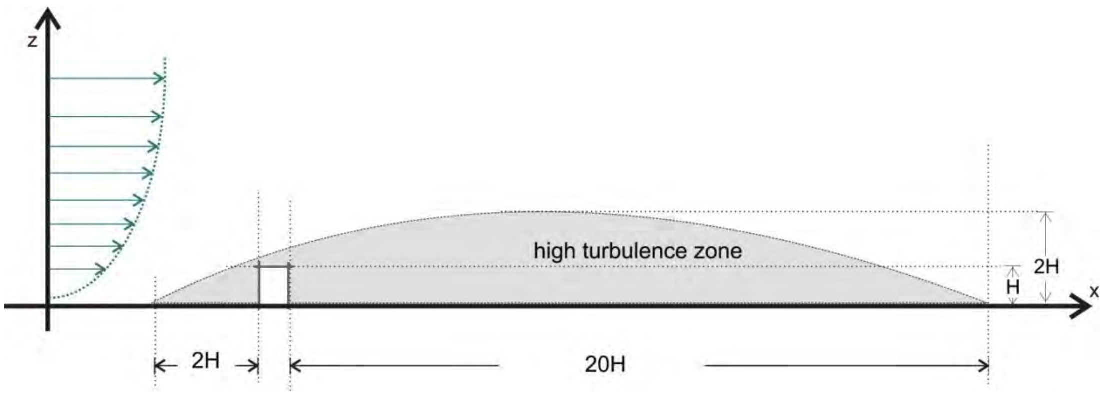


FIG. 2.3. Turbulence from obstacles. (Adapted from Wizelius[18]).

Differences in air temperature can also create turbulence and reduce the wind speed. Air temperature at ground level is warmer than at higher levels, at the same time, temperature decreases somewhat rapidly with height causing warm air to rise upwards. As a result, the horizontal wind will meet air that is rising – moving in a vertical direction – creating turbulence. As it will be further discussed, turbulence is of special importance when selecting the type of wind turbine for building integration.

2.4.1 Wind flows around buildings

The study of flow fields around buildings is important because the combination of building shape, height and distance to other buildings affect the direction and intensity of wind flows and can potentially deteriorate the wind conditions in public access ways or recreational areas, especially at pedestrian level. Additionally, the understanding of how wind flows behave around a building is of major importance when assessing the best location for wind turbines integration.

The building is a three dimensional object that in aerodynamic terms is often referred to as a bluff body. The wind flows around the building and the incident freestream flow is a turbulent boundary layer with mean velocity increasing with height and turbulence intensity decreasing with height. There are two different mechanisms, called pressure fields, which cause high induced wind speeds at ground level.

In the first pressure field, the wind coming upstream causes a pressure distribution on the windward façade of a building of height H, the wind flows generated as a consequence are depicted in Figure 2.4. The local dynamic pressure increases with height and induces flow vertically down the windward façade below the stagnation point (H_S). As a rule of thumb, for sharp edged buildings of approximately square proportions, H_s is located at 0.85H[XI]. Below H_s, the flow rolls up into a standing vortex system at the base of the building causing high wind speeds in this region.

[XI] Studies showed that on sharp edged buildings in a boundary layer with negligible displacement height, the location of H_S does not significantly vary when modifying the size of the building[19].

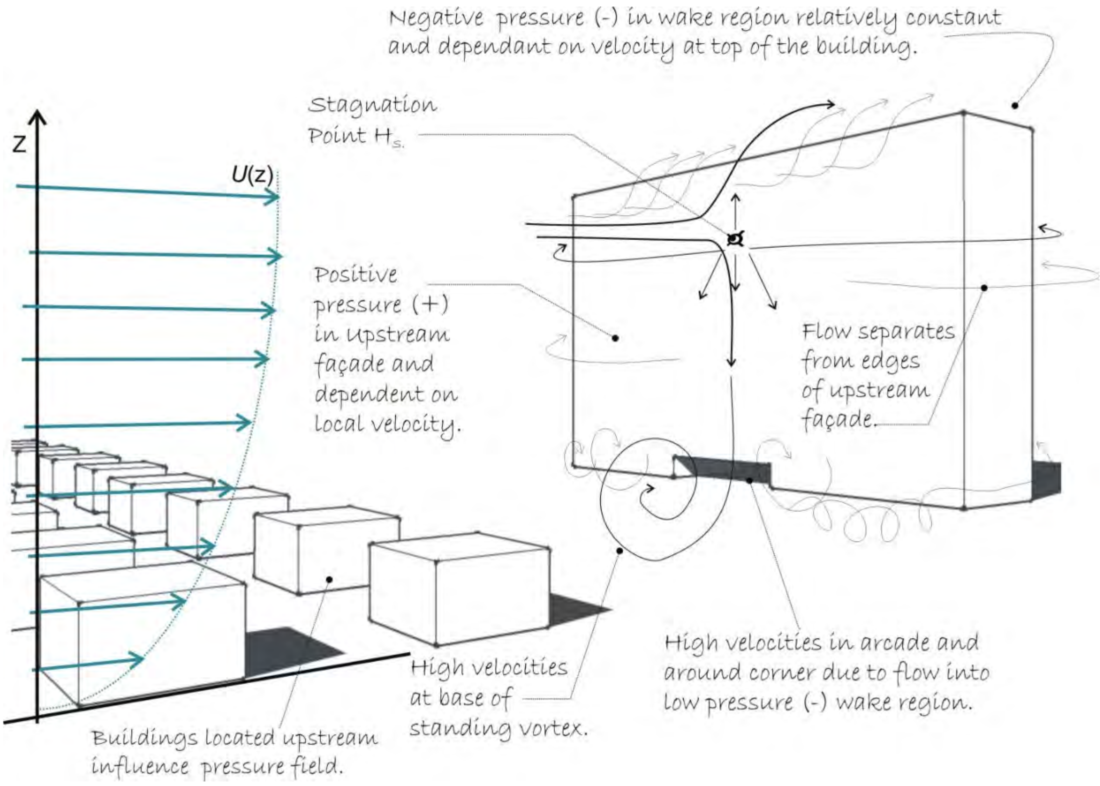


FIG. 2.4. Wind flow around a building. (Adapted from Aynsley[22]).

Buildings with circular platform promote lateral flow and do not produce strong vertical flows. Conversely, rectangular and concave buildings produce strong vertical flows with consequent high wind conditions in the standing vortex system[22]. Buildings located upstream have a critical influence on this pressure field. Under certain conditions, upstream building configuration can augment the vortex in front of the building, increasing the high wind speeds at pedestrian level.

The second pressure field is caused by the pressure difference between the low pressure wake regions (leeward and side façades) and the relatively high pressure regions at the base on the windward façade. The low pressure on the leeward side, or wake pressure, cannot be easily modified and is dependent on the velocity along the top free boundary, that is, the free-stream velocity at the top of the building. In other words, the taller the building, the lower the wake pressure and the higher the velocities which are induced through arcades and around corners.

Ideally, those high velocities should be avoided at pedestrian level to prevent uncomforting sensations, but they can be exploited to harness the power carried in the wind and transform it into electric power.

2.4.2 The power of wind

The wind has kinetic energy[XII] and this can be turned into electric power using wind turbines. The energy content of the wind at a specific height at a site is usually specified in kWh per square metre per year; in other words, the energy contained in the wind that passes through a vertical area of one square metre during one year (kWh/m^2/year). Sometimes, the same measure of wind is expressed as power density instead and it is expressed in W·m^{-2}. Multiplying a given value of power density by 8760 hours and dividing it by 1000 one can obtain the energy content in kWh/m^2/year.

The power of the wind is proportional to the cube of the wind speed, therefore if the wind speed doubles, the power increases eight times.

The power carried by the wind P with velocity u is expressed by the following equation:

$$P = \frac{1}{2} m \times u^2 \qquad \text{(Eqn. 2.1)}$$

[XII] It is worth to differentiate the concept of power from the concept of energy. Energy is the power multiplied by the time the power is used and is usually expressed in kWh, in contrast with power that is expressed in kW.

Where m stands for mass flow rate with density ρ, and is flowing through an area A with a velocity u:

(**Eqn 2.2**) $$m = \rho \times A \times u$$

Hence substituting equation **2.2** in equation **2.1**, the power of wind is:

(**Eqn 2.3**) $$P = \frac{1}{2}\rho \times A \times u^3$$

The density ρ of air varies with the air pressure and temperature. The standard values used for density are typically at sea level (pressure equal to 1 bar) and at a temperature of 9°C, which gives a value of ρ = 1.25kg.m^{-3}. The power of wind per square metre is then:

(**Eqn 2.4**) $$P = \frac{1}{2} \times 1.25 \times u^3 = 0.625 \times u^3$$

Where P = power in Watts or J.s^{-1} and u^3= wind velocity. The annual energy content of the wind per square metre is then:

(**Eqn 2.5**) $$E = \frac{0.625 \times u^3 \times 8760}{1000} = \text{kWh/m}^2\text{/year}$$

2.4.3 The wind resource

According to the European Wind Energy Association (EWEA)[23], the UK has the best wind resource in Europe, an asset that has the potential to provide a considerable proportion of the UK energy market in years to come. Nevertheless, this does not mean that every wind project will produce satisfactory outputs. In order to design buildings that harness wind power it is not sufficient to understand how a wind turbine operates from a technical point of view. It is just as important to be able to estimate the wind resources at a specific site: the energy content of the wind should always be the basis for a wind power project.

For example, suppose that the wind speed at the design site is assumed to always be 5m.s^{-1}; the annual energy content of the wind can be calculated with equation 2.5, and it would be:

$$E = \frac{0.625 \times 5^3 \times 8760}{1000} = 864\ \text{kWh/m}^2\text{/year}$$

This is a simplistic approach that does not reflect the real life wind patterns, the wind speed and direction change continuously during the day, in different seasons and even between years.

In order to calculate the power density at a specific site it is necessary to calculate the average power density. This is done by averaging the cube of the recorded wind velocities. Yet the average wind speed can only partly describe the potential of a site, because the energy content of the wind does not depend linearly on the wind speed and it is thus necessary to know the different wind velocities that occur at the site and their duration, this is called the frequency distribution of wind speeds.

To obtain better estimates of the power density, the calculations of the energy content of the wind during one year using the frequency distribution are carried out using the cubes of the wind velocities, multiply these by the frequency and then adding these values. The result of this calculation is then substituted in the value of *u* in equation 2.6[XIII].

(Eqn 2.6) $$E = \frac{0.625 \times v \times 8760}{1000} = \text{kWh/m}^2\text{/year}$$

Alternatively, a cube factor can be added when using equation 2.5 to calculate the energy content during a year if the mean speed is known but not the frequency distribution. A cube factor value of 1.91 can be used for places in mid latitudes, the USA and most parts of Europe[18]. This factor depends on the frequency distribution of the wind and for places with trade or seasonal winds other values should be used. The most effective way to produce acceptable and detailed information about the energy content of the wind at the design site is to carry out a series of readings. This is carried out by installing wind measurement equipment that record wind speed and direction data[XIV], preferably at the hub height of the turbine intended for the site.

Nevertheless, wind speed and frequency distribution vary significantly during different years; this is why examining long term averages provide less variations in the data, which is better from a statistical point of view. It is often however, not

[XIII] Note that equations 2.5 and 2.6 differ on the value of wind velocity, one being cube (u^3) and the other one linear (v) respectively.
[XIV] Wind velocities are measured using and anemometer and wind directions are registered by a wind vane.

practical to measure wind conditions over periods of five years or more in the specific site where the wind turbines would be sited.

This is why one approach is to take the nearest meteorological station that has long term average records as a reference site. The station must be representative for the regional wind profile at the design location. Usually, a correlation is valid only between sites having the same shape of wind speed distribution[XV].

Alternatively, commercially available computer-based prediction tools[XVI] can be used to give a slightly more refined indication of site wind speed, like the computer model of the UK wind energy resource developed by the DTI[24].

It is clear then that electricity production obtained from a wind potential with a given wind speed and wind turbine type, varies a lot with the wind speed distribution around the mean value. Wind speed distributions are commonly used to indicate the annual available wind energy. Therefore, it is essential to know the distribution of wind speeds on the project site[XVII].

For this study, wind data for Nottingham, UK. were obtained as a Test Reference Year (TRY) file[26]. The Test Reference Year consists of hourly data for twelve typical months, selected from approximately 20-year data sets (typically 1983-2004), and

---

[XV] The frequency distribution of the wind has proved to fit quite well to a probability distribution called the Weibull distribution and to the Rayleigh distribution, which is a special case of the former[25].
[XVI] There are several different computer programs for wind power applications, based on the wind atlas method, like the Danish WAsP and EMD or the British WindFarm and WindFarmer[18]. Nevertheless, the developer of WaSP has expressed his concerns regarding the software being applied for physical and statistical estimates of wind speeds in the built environment; stating that given the complexity of the flow patterns, he would not expect good results[27].
[XVII] As general indication, in the UK, it is possible to obtain an assessment of the average wind speed at a particular site, by using the UK wind speed database, available at www.bwea.com.

smoothed to provide a composite, but continuous, 1-year sequence of data. These data show the probability of maximum gust velocity likely to occur at a height of 10m and lasting for 3 seconds.

In order to manage these data, the TRY file was exported to ECOTECT V.5.2 software to be later analysed with the Weather Tool of the same software.

Table 2.1 shows the frequency distribution for Nottingham, UK. It can be appreciated that the prevailing winds approach from South-West and West orientations (202.5° <θ <270°). Figure 2.5 illustrates the frequency distribution of wind speeds for those orientations (SW, SSW, WSW and W). The most frequent wind speeds are in the range of 3 to 5.5m.s^{-1}; 3.9% of the time the wind speed is 2.8m.s^{-1} and the mean wind speed coming from those orientations is 4.4m.s^{-1}.

Using this frequency distribution of wind speeds, the energy content during a year for Nottingham, UK can be calculated with equation **2.6**, resulting in a power density of 97.14 W.m^{-2}, or a total energy content of 851 kWh/m^{2}/year.

How efficiently this power of the wind is utilized – i.e. converted to electricity – largely depends on the type of wind turbine to be used and its chosen siting location. Therefore, it is crucial to have a broad understanding of wind turbine basics.

Table 2.1. Frequency distribution of wind speed for different orientations in Nottingham, UK.

Orientation	wind Speed					wind Frequency
	2.77 m/s	5.55 m/s	8.33 m/s	11.11 m/s	13.88 m/s	(Hrs) %
N	1.8%	1.6%	0.6%	0.2%	0.0%	**4.2%**
NNE	1.2%	0.7%	0.4%	0.1%	0.0%	**2.4%**
NE	1.4%	0.8%	0.4%	0.1%	0.0%	**2.7%**
ENE	1.3%	1.4%	0.5%	0.2%	0.0%	**3.4%**
E	1.6%	1.3%	0.4%	0.1%	0.0%	**3.4%**
ESE	1.5%	1.5%	0.6%	0.1%	0.0%	**3.7%**
SE	1.3%	1.5%	0.6%	0.0%	0.0%	**3.4%**
SSE	2.2%	2.4%	1.2%	0.1%	0.0%	**5.9%**
S	2.4%	2.9%	1.5%	0.2%	0.0%	**7.0%**
SSW	3.9%	4.0%	1.7%	0.4%	0.0%	**10.0%**
SW	4.9%	5.6%	1.9%	0.4%	0.1%	**12.9%**
WSW	5.9%	5.3%	1.8%	0.5%	0.1%	**13.6%**
W	5.0%	4.0%	1.6%	0.4%	0.1%	**11.1%**
WNW	3.0%	2.3%	0.9%	0.3%	0.0%	**6.5%**
NW	2.1%	1.6%	0.6%	0.1%	0.0%	**4.4%**
NNW	2.0%	1.4%	0.7%	0.2%	0.0%	**4.3%**
	41.5%	38.3%	15.4%	3.4%	0.3%	

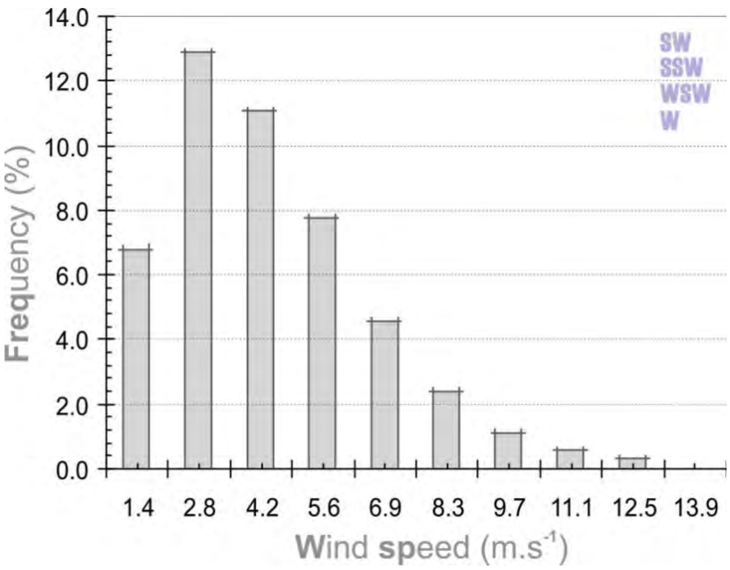


FIG. 2.5. Nottingham frequency distribution of wind speed for orientations 202.5° <θ <270°.

2.5 Wind turbine basics

Wind turbines harness kinetic energy in the wind and transform it into other forms of energy: electric power, or mechanical work[28]. There are many designs and configurations of wind turbines but, whatever their shape and size, they can generally be grouped into two types (Figures 2.6 and 2.7):

- Vertical axis wind turbines (VAWTs).
- Horizontal axis turbines (HAWTs).

The most common type of wind turbine is the horizontal axis type. They receive this name because the axis of rotation of the blades is in a horizontal position. HAWTs for electricity generation generally have either two or three blades and are sometimes referred to as low-solidity[XVIII]. In the UK, these are by far the most common kind of wind turbines – although not necessarily the best type for all urban locations.

FIG. 2.6 Horizontal axis wind turbine (HAWT)

[XVIII] Multi-bladed HAWTs are used mainly for water pumping in farms and because of the large number of blades they have what appears to be a virtually solid disc, hence, they are called high-solidity devices[29].

FIG. 2.7 Vertical axis wind turbine (VAWT)

The VAWTs have a vertical axis of rotation of the blades and a shaft is in a vertical position. One advantage of these types of machines is that they do not have to be faced in any particular direction with respect to the wind. These turbines will work equally effective irrespective of wind direction, unlike their horizontal counterparts that make use of a yawing system to track variations in wind direction and then reposition the rotor when the wind direction changes. Another benefit of this type of turbines is that the generator and gearbox can be installed at ground level, making them easy to service and repair.

VAWTs can also be classified depending on whether they use drag or lift forces to harness wind energy. Drag forces are those forces experienced by an object in an air stream that are in line with the direction of the air stream. Lift forces are those forces experienced by an object in an air stream that are perpendicular to the direction of the air stream[29]. The magnitude of the drag and lift forces depends on the shape of the object, its orientation to the direction of the air stream and the velocity of the air stream.

The most common drag and lift VAWT are:

- SavoniusType[XIX]: uses drag forces to create rotation of the shaft.
- Darrieus Type: uses lift forces to create the rotation of the shaft.

The main difference between the two systems is the **tip speed ratio (λ).** Tip speed ratio is a non-dimensional ratio, obtained by dividing the tangential velocity of a rotor at the tip of the blades called tip speed, by the undisturbed wind velocity upstream of the rotor. The tip speed ratio is useful when comparing wind turbines of different characteristics. A high solidity device has a lower **λ,** while a low solidity device has a higher **λ** because the blades need to travel much faster to virtually fill up the rotor area. Optimum tip speed ratios[XX] of low solidity wind turbines range between 6 and 20[29].

Besides the tip speed ratio, there are other characteristics that differentiate drag and lift systems:

Drag design

The wind hits the face of the turbine blades and **pushes** it out of the way causing the turbine to spin. Drag design wind turbines have lower aerodynamic efficiency and they require a large amount of material (high solidity) for their construction.

[XIX] The lift force is actually applied also to the drag devices, but to a lesser degree, otherwise the tip speed ratio would not exceed one[18].
[XX] The optimum tip speed ratio will depend on the width and number of blades and it is when the wind turbine operates with its best efficiency i.e. the velocity of its blades tips is a particular multiple of the wind velocity[29].

Lift design

The blades have an aerofoil section and the **difference in velocity** of the wind flowing over the top of the blade from the wind flowing over the bottom, gives rise to lift forces that create the spinning motion. Lift design wind turbines have a high aerodynamic efficiency and they will run smoothly in high turbulence winds. They are low solidity, what means that they require less material to manufacture.

According to the lift and drag characteristics, it can be concluded that a lift type VAWT offers more advantages for using in urban environments.

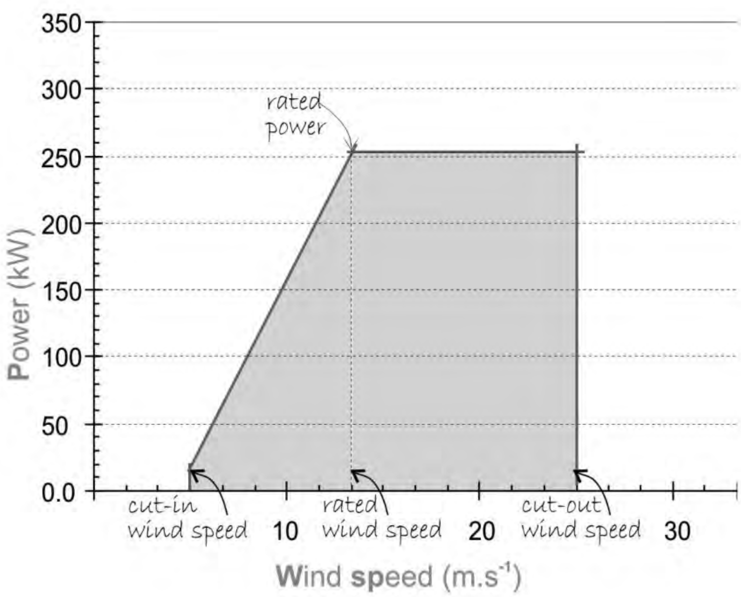
2.5.1 Installed capacity and power

The power output of a wind turbine varies with wind speed and every turbine has a characteristic wind speed power curve. The power curve will primarily determine how much energy can be produced by a particular turbine on a given site under given wind conditions. Box 2.2 shows a typical wind turbine wind speed power curve. The power curve of a wind turbine can also be shown as a table or a bar chart and it is available from the manufactures.

Therefore, to make an exact calculation of how much a wind turbine will produce at a given site, two things have to be known: the power curve of the wind turbine; and the frequency distribution of the wind speed at the hub height at the selected site (See Figure 2.5).

For each wind speed within the operating range of the turbine, the energy produced at that wind speed can be obtained by multiplying the number of hours of its duration by the corresponding turbine power at this wind speed, provided by the turbine's wind speed power curve. These data can then be used to plot a wind energy distribution curve. The total energy produced is then calculated by adding the energy produced at all wind speeds within the operating range of the turbine.

BOX 2.2 typical wind turbine speed power curve



Rated power

Maximum power produced by a wind turbine.

Rated wind speed

Wind speed at which a wind turbine reaches its rated power.

Cut-in wind speed

Lowest wind speed at which a wind turbine will operate.

Cut-out wind speed

Or shut down wind speed is the maximum wind speed at which a wind turbine will operate.

The installed capacity, also called rated capacity is the maximum steady power that an electricity generator can produce. It is usually expressed in kilowatts or megawatts. The installed capacity of a wind turbine tells us its potential output. However it is more relevant to look at the historical or expected annual electricity yield as this is the electricity that we will actually use[30].

The declared net capacity (DNC) is a measure of the equivalent capacity of base-loaded plant (coal gas or nuclear power plant) that would generate the same average annual energy output as the renewable energy plant. It is therefore, used to provide a rough comparison of the output of different energy technologies against a conventional plant. The DNC for wind is calculated by multiplying the net installed capacity (the capacity allowing for on-site losses) by 0.43.

Wind turbines have been developed with a range of power ratings from a few kilowatts up to multi-megawatt capacity. Typically, according to its size, a wind turbine can be classified as:

- Large scale stand alone wind turbines >100kW
- Small scale roof mounted wind turbines <10kW
- Building integrated wind turbines
- Small to medium scale stand alone wind turbines

Commercially available large scale stand alone turbines are now rated at around 600kW[XXI]. Wind technology is well established, with global installed capacity of 94 GW that will save about 122 million tons of CO_2 every year[31]. Current research and development work is concerned mostly with improvements in reliability, cost and noise reduction, and performance. Existing designs aim at a machine lifetime of about 25 years.

However, this rated power, is only theoretical and in practice the value will be a lot less. A value known as the power coefficient (Cp) is the ratio of the actual power

---

[XXI] There are bigger prototypes like the German Enercon E-126, rated at 6 MW. The tower is 138 metres high and the diameter of the rotor is 126 metres[32].

output compared to the theoretical available. The value of Cp is very unlikely to exceed the value of 0.593.

Because wind is a very unpredictable resource, the electricity that can be produced is affected due to its intermittent nature. At other times it can be very strong and causes vast amounts of damage to structures, this is why most wind turbines are designed to have a cut-in and cut-out speed in the range of ~4m/s to ~ 25m/s.

When the wind passes through the rotor of a turbine, a very strong turbulence is created. This whirling wind on the lee side of the rotor is called wind wake and influence wind speed up to a distance of 10 rotor or more behind the turbine[18].

2.5.2 Building mounted and building integrated wind turbines (BUWTs)

It has been noted that modern wind turbines have developed according to the three-bladed HAWT principle, operating with a high tip speed, typically around $60m.s^{-1}$. They usually operate in well-exposed wind conditions, away from populated areas or urban areas[XXII]. Their dimensions have grown well beyond 100m in height and to rotor diameters of similar dimensions.

The rated capacity of both HAWTs and VAWTs roof mounted turbines are in the 2.5kW to 6kW range. The construction and operation of wind turbines for use in the built environment however have to be developed using a different set of design

[XXII] Nevertheless, there are increasing numbers of examples of large scale stand alone wind turbines being implemented in the urban environment. See for example the 600kW turbine installed in the Sainsburys distribution centre in East Kilbride, Scotland: http://www.scotland.gov.uk/pages/news/2001/09/SE4063.aspx starting points. The design of such urban wind turbines according to van Bussel[33] has to be defined by the following requirements:

- Good performance in complex winds
- Safe operation in the urban environment
- Low noise level
- Simple, rugged design
- Minimised maintenance
- Aesthetic appearance

The reason why a number of research projects on building integrated wind turbines have been published recently is easy to understand according to the information presented in the previous sections. Arguably, the most relevant work on the integration of wind turbines in buildings is the report partially funded by the Carbon Trust that explores the feasibility of building mounted and building integrated wind turbines (BUTWs)[27]. The report uses the term BUTWs to define wind turbines that are close to or on buildings. Nevertheless, it differentiates between the building integrated wind turbines and building mounted wind turbines. Defining the former as turbines capable of working close to buildings and where possible any augmentation that the building causes to the local wind flow and that can or cannot be integrated within the building design. Building mounted wind turbines are defined as those turbines that are physically linked to the building structure, mainly rooftop mounting.

According to the Carbon Trust report, interest from architects in fitting wind turbines to buildings has grown in recent years. Although well-known practices are currently leading on these kind of projects, there are also some new, innovative architectural practices that are including BUWTs in their conceptual designs.

One project that features roof mounted wind turbines integrated into the building's structure is the COR project by the firm OPPENheim architecture + design[XXIII]. The project comprises several HAWTs located at different heights all over the four façades (Figure 2.8), this decision could be the designer's way of overcoming the shortfall of the HAWT not being able to harness wind from changing directions.

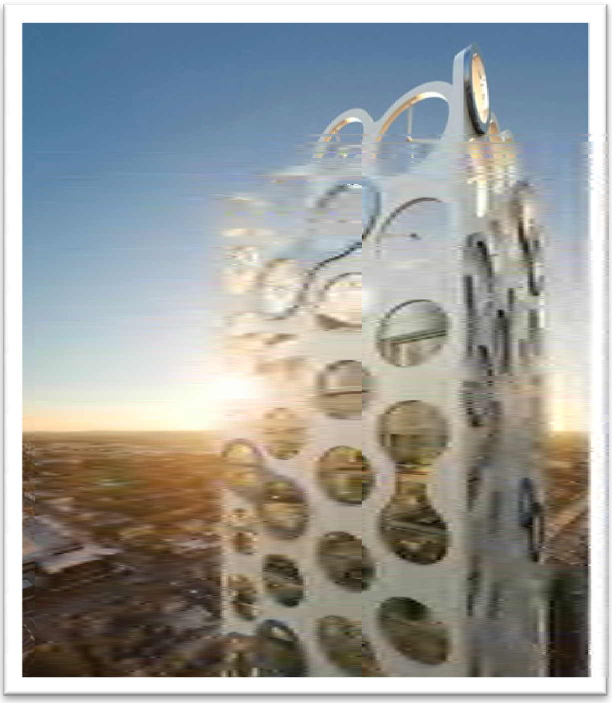


FIG. 2.8 COR's exterior view.

Nevertheless, the annual wind rose, provided on the firm's site analysis, clearly shows prevailing winds from the E and SE directions; making the decision of siting wind turbines on the four façades of the building not financially sensible. Moreover, as it will be discussed on section 6.3, in a sharp-edged building with flat roof the flow separates at those sharp edges. Because of the separation at the upwind roof edge,

[XXIII] More information available at: www.oppenoffice.com the velocity vectors outside the recirculation region are not parallel to the roof. In the case of the COR project, it is expected that the rotors would be operating in skewed flow, increasing the probability of failure.

One of the retrofitted projects is the Boston Logan Airport Building (Figure 2.9). The wind turbine model is the AVX1000, also known as Architectural Wind[XXIV]. The manufacturer states that it has been specifically engineered to take advantage of the acceleration effect of wind as it passes over the building parapet.

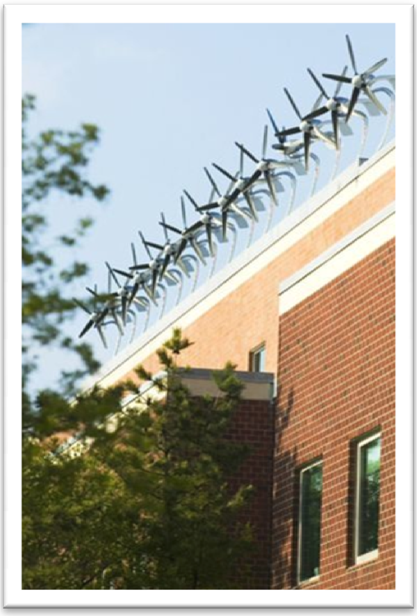


FIG. 2.9 Boston Logan Airport Building.

Even if it seems that because of its angled rotor, the Architectural Wind overcomes the issue of working on skewed flow, the performance of the turbine may vary because they are designed for installation on a concrete tilt-up and the skew angle will vary depending on many factors[34]: the position of the roof, the roughness of the

[XXIV] More information available at: http://www.avinc.com/cleanenergy.asp upwind area, the size of the building, upwind edge rounding and the yaw of the free stream wind.

The Carbon Trust report provides a comprehensive introduction to wind turbines and the requirements for assessing the wind resource in urban areas. It also includes a brief review of the most relevant designs of small wind turbines for urban use (building mounted) and some designs of turbines that include ducting devices that in theory are capable of augmenting power by accelerating wind speed (building integrated).

Probably the most popular of the recent examples of the building integrated designs using the funnelling concept are the Bahrain WTC[XXV] twin towers (Figure 2.10). On March 2007 the successful installation of the three HAWTs was announced, making the Bahrain twin towers the first architectural project in the world to integrate wind turbines for energy generation.

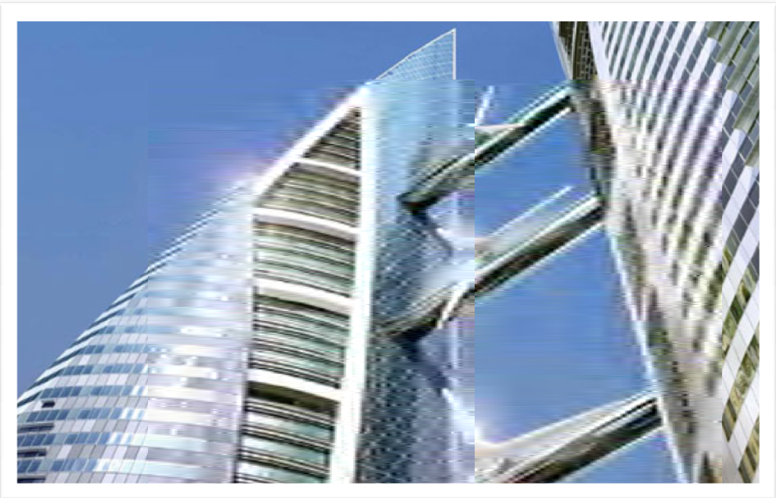


FIG. 2.10 Bahrain WTC Twin Towers. Exterior view.

[XXV] More information available at: http://bahrainwtc.com/news35.htm

Finally, an attractive concept of building integration wind turbines and aerodynamic geometry is SOM's Pearl River Tower[XXVI] illustrated in Figure 2.11. The tower's curvilinear form funnels air through two inlets in the façade where two VAWTs are sited.

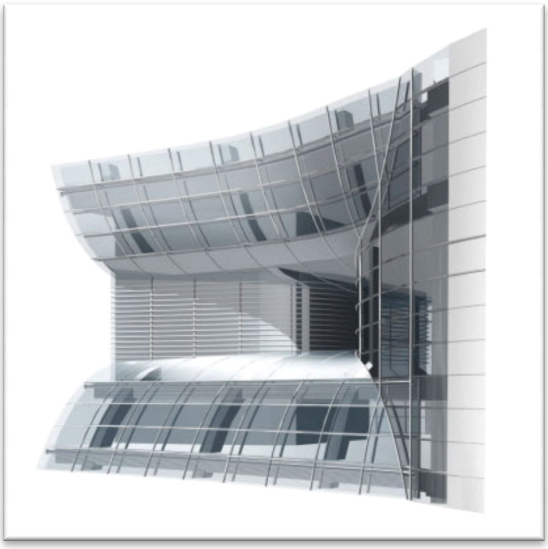


FIG. 2.11 Pearl River Tower and augmentation inlets.

Two key distinctive features of this design can be pointed out: it is not dependant on having two tower buildings to funnel wind and the wind turbines are not visually intrusive. As a designer, this can be regarded as an ideal way of integrating renewable energy systems to building design.

[XXVI] More information available at: http://www.som.com/content.cfm/pearl_river_tower

It is clear that the choice of the wind turbine will be highly dependent on the site and the type of project. Table 2.2 summarises some of the wind turbine models that have been especially designed for the use in the built environment. Where possible, the rated capacity and the rated wind velocity were included.

Table 2.2 Wind turbines designed for the built environment.

Company / Research Centre	BUWT model	Type of Wind Turbine	Rated capacity	Development date	COUNTRY
BRE (WT), Garrad Hassan & Partners	Wind Dam WD1 system	VAWT	N/A	2004	UK
Ecofys	Neoga & Hera	VAWT	N/A	2002	Netherlands
XCO2	Quiet Revolution	VAWT	4.2 kW@ $11 m.s^{-1}$	2005	UK
TU Delft, CORE International, S. Mertens	Turby	VAWT	2.5 kW @ $14 m.s^{-1}$	2004	Netherlands
Aeromag	Lakota	HAWT	1 kW	2004	Canada
NGUp	Wind Wall	VAWT	3 kW	2004	Netherlands
Proven	WT 600 and WT2500	HAWT	0.6 and 15kW	N/A	UK
Aerotecture, University of Illinois, B. Becker	510 V and 520 V Aeroturbines	VAWT/HAWT	1 and 1.8 kW	2006	USA
WES	WES5 Tulipo	HAWT	2.5 @ $10 m.s^{-1}$	N/A	Netherlands

Table 2.3 summarises some of the wind turbine designs for urban environments that include augmentation devices[XXVII].

[XXVII] N/A is used in tables 2.3 and 2.4 in the cases where the specific information was not available.

Table 2.3 Wind turbines for the built environment with augmentation devices.

AUTHORS	Company / Research Centre	Project Name	Development date	COUNTRY
Derek Taylor, Godfrey Boyle	Altechnica and The Open University (wtt)	Aeolian Roof	2004	UK
N/A	FreeGEN	Combined Augmented Technology Turbine (CATT)	2004	UK
Neil Campbell	BDSP (CFD), Imperial College (wtt), University of Stuggart	WEB Concentrator	2002	UK, Germany
Mike Blanch, Geoff Dutton, Jim Halliday	CCLRC	Field-testing for WEB project	2002	UK
Garrad Hassan	BRE (wtt)	Wind Dam WD1 system	2004	UK
Stanley Marquiss	Wind Electric Inc.	Ducted Fan Wind Turbine (DFWT)	2004	USA

Problems associated with the implementation of BUWTs

According to the Carbon Trust[27], BUWTs are likely to fail for several reasons, including not being robust enough to cope with the turbulence and wind shear that are characteristic of urban environments. In addition to that, having lower than expected performance is also identified as a common drawback when developing wind energy projects.

Additionally, safety measures must be addressed in key points such as[XXVIII]:

Noise: The selection of wind turbines for urban environments has to comply with low noise levels. Careful placement of turbines is also required (avoid places of unfavourable wind conditions, avoid sensitive locations i.e. areas where noise levels are required to be low).

[XXVIII] Any commercial turbine fitted to a building or next to it has to be certified to compliant with BS EN 16400 and be CE marked.

Structurally transmitted vibration: Turbines must provide special mountings to absorb or isolate vibration.

When proper consideration is given to these points, wind energy systems can offer a secure, equitable, affordable and sustainable energy supply, which is vital to future prosperity. Nevertheless, it has to be kept in mind that the main goal of all energy projects is to provide energy services that improve quality of life (e.g. health, life expectancy, and comfort) and productivity.

The concept of distributed energy generation, which means that energy conversion units are situated close to energy consumers and large units are substituted by smaller ones, is closely related with the use of BUWT, because it is an efficient, reliable and environmentally friendly alternative to the traditional energy system. In terms of technology, distributed energy generation especially aims at the utilisation of local sources of energy[35].

Nevertheless, in order to properly make assessments of the wind resource at a specific site for the successful siting of wind turbines, especially in the built environment, the use of appropriate tools and techniques is needed. The relevant techniques used for the evaluation of wind conditions around buildings will be discussed in next chapter.

2.6 Chapter two – key concepts

In this chapter, it has been established that wind energy is one of the most efficient renewable energy sources, and that due the nature of wind, it is nonetheless, variable compared to other sources of energy. It was also recognized that it is more sensitive to variations in topography and weather patterns. Nevertheless, wind energy can be harvested effectively if the wind turbine is sited in a windy area, - as the power of wind is proportional to the cube of wind speed- and a careful choice of the type of wind turbine that matches with the wind pattern of the site is properly made. Consequently, a thorough knowledge on the wind speed characteristics at a site of interest is very important before planning to harness the wind energy.

To sum up, (Box. 2.3) to successfully implement wind energy in the built environment three major issues have to be addressed: Wind resource assessment and wind characterisation around buildings, structural integration of wind turbines with buildings and special urban wind turbine design requirements[33].

BOX 2.3 chapter two key concepts

a2 On zero and low carbon developments

- ZED/LEDs include bioclimatic concepts such as building orientation and passive solar heating. An important feature of a ZED / LED is that it should not contribute to any of the greenhouse gases to the atmosphere. Nevertheless, not many of the developments include wind energy generation.

b2 On renewable and energy efficient technologies

- Renewable energies are a critical element for achieving sustainable development.
- In the UK, there are support measures for promoting microgeneration using renewable energy systems, including small scale wind turbines.
- Wind energy is a good choice of renewable energy, with low costs per installed kW and it is a potentially attractive building integrated power source.

c2 On wind characteristics and wind turbines

- The friction against the surface affects the wind creating an internal boundary layer.
- Urban environments represent zones of high turbulent flows and as a result the use of wind turbines in the cities is more complicated than in rural or open spaces.
- The power of the wind is proportional to the cube of the wind speed, therefore if the wind speed doubles, the power increases eight times.
- Wind speed distributions are essential to indicate the annual available wind energy.
- Lift type VAWTs offer more advantages for using in urban environments.
- Building mounted wind turbines and ducted wind turbines have been specially designed for using in urban environments.

2.7 Chapter two – reference list

1. Rogers, R. (1997) Cities for a small planet. Gumuchdjian, P., (Ed.) 180pp Great Britain: Faber and Faber Ltd.

2. Appleton, A.F. (2006) Sustainability: A practitioner's reflection: Sustainable Cities. *Technology in Society* **28** (1-2):3-18.

3. Birkeland, J. (2002) *Design for sustainability. A sourcebook of integrated eco-logical solutions.* UK: Earthscan.

4. Anonymous (2004) Renewables 2004. International conference for renewable energies. Bonn, Germany. *Conference Report Outcomes & Documentation – Political Declaration / International Action Programme / Policy Recommendations for Renewable Energies.* 54pp. Available at: <http://www.renewables2004.de/pdf/conference_report.pdf>. Accessed 12/07/05

5. Dunham-Jones, E. (1998) Comment: Questioning the Stand-Alone Building. In: Scott, A., (Ed.) *Dimensions of sustainability : architecture form, technology, environment, culture*, pp. 93-98. London ;, New York : E & FN Spon]

6. Department of the Environment, T.a.t.R.D. (1996) Building a sustainable future. Homes for an autonomous community. *Energy Efficiency Best Practice Programme.* General Information Report 53. 24pp.

7. ODPM (2005) Sustainable communities: building for the future. Office of the Deputy Prime Minister. 60pp

8. DTI (2006) Our Energy Challenge: Power from the people. *Microgeneration Strategy.* Department of Trade and Industry. DTI/Pub 8243-1k/03/06/NP. URN 06/993.

9. Bill Dunster architects ZEDfactory Ltd. (2006) From A to ZED. Realising Zero Energy (fossil) Developments. Second edn, CD-ROM format. 186pp. UK: The housing corporation. Innovation & Best Practice Program (IGP).

10. Paz, O. (1991) La búsqueda del presente (In search of the present). Nobel Lecture, 1990. *In*: Frängsmyr, T. (Editor) 1991. *The Nobel Prizes 1990*. Stockholm, Sweden. [online] URL: http://nobelprize.org/nobel_prizes/literature/laureates/1990/paz-lecture-e.html.

11. Hess, D.J. (2006) *Pathways in science and industry: activism, innovation and the environment in an era of globalization.*, USA: MIT Press.

12. Intrachooto, S. and Horayangkura, V. (2007) Energy efficient innovation: Overcoming financial barriers. *Building and Environment* **42** (2):599-604.

13. Cunningham, D. and Goodbun, J. (2006) Marx, architecture and modernity. *The Journal of Architecture* **11** (2):169-185.

14. IPCC (2007) Fourth Assessment Report, Working Group III. *Chapter 4 Energy Supply. International Panel for Climate Change.* Final Draft. 112pp. Available on line at: <www.mnp.nl/ipcc/index.html>. Accessed: 20/06/07

15. BERR (2008) Department for Business Enterprise & Regulatory Reform. Low Carbon Buildings Programme. <http://www.lowcarbonbuildings.org.uk> Accessed 7-2/08.

16. Smith, P.F. (2003) *Sustainability at the cutting edge : emerging technologies for low energy buildings*, Jordan Hill, Oxford : Architectural Press.

17. Bahaj, A.S., Myers, L. and James, P.A.B. (2007) Urban energy generation: Influence of micro-wind turbine output on electricity consumption in buildings. *Energy and Buildings* **39** (2):154-165.

18. Wizelius, T. (2007) *Developing wind power projects. Theory and practice.* UK: Earthscan.

19. Mertens, S. (2006) *Wind energy in the built environment*. Multi-Science. Essex, UK. 169 p.

20. Panel on the Assessment of Wind Engineering Issues (1993) *Wind and the Built Environment: U.S. Needs in Wind Engineering and Hazard Mitigation*, Washington, DC, USA: National Academies Press.

21. Cook, N.J. (1985) *The designer's guide to wind loading of building structures*, Garston, Watford : Building Research Establishment, Dept. of the Environment ; London ; Boston : Butterworths.

22. Aynsley, R.M., Melbourne, W.H. and Vickery, B.J. (1977) *Architectural aerodynamics*, London : Applied Science Publishers.

23. EWEA (2008) European Wind Energy Association. <http://www.ewea.org> Accessed 6-2/08.

24. DTI and ETSU (1999) Assessing the potential of wind energy projects. *Notes for developers.* Department of Trade and Industry. 43 pp. Available at: <http://www.dti.gov.uk/files/file17833.pdf>

25. Quaschning, V. (2005) *Understanding renewable energy systems.*, Toronto, Canada: Earthscan.

26. CIBSE/Met Office (2003) Test Reference Year data for Nottingham, UK. Nottingham TRY/DSY Hourly Data Set (CD-ROM)

27. Dutton, A.G., Halliday, J.A. and Blanch, M.J. (2005) The feasibility of Building-Mounted/Integrated Wind Turbines (BUWTs): Achieving their potential for carbon emission reductions. Final report to the the Carbon Trust (2002-07-028-1-6). 109 pp.

28. Gipe, P. (1999) *Wind energy basics : a guide to small and micro wind systems*, White River Junction, Vt. : Chelsea Green Pub. Co.

29. Boyle, G. (2002) *Renewable Energy. Power for a sustainable future.* UK: Oxford University Press & The Open University.

30. DTI (2001) Wind energy fact sheets. Department of Trade and Industry. Available at: <http://www.dti.gov.uk/renewable/index.htlm

31. GWEC (2008) Global Wind Energy Council. <http://www.gwec.net> Accessed 6-2/08.

32. ENERCON (2008) <http://www.enercon.de> Accessed 6-2/08.

33. van Bussel, G. and Mertens, S. (2005) Small wind turbines for the built environment. N´aprstek, J. and Fischer, C. (eds). *The Fourth European & African Conference on Wind Engineering* Prague: ITAM AS CR.

34. Mertens, S., van Kuik, G. and van Bussel, G. (2003-) Performance of an H-Darrieus in the Skewed Flow on a Roof. *Journal of Solar Energy Engineering* **125** (4):433-440.

35. Alanne, K. and Saari, A. (2006) Distributed energy generation and sustainable development. *Renewable and Sustainable Energy Reviews* **10** (6):539-558.

3 research approach

3.1 Introduction

In this chapter the rationale for the nine cases utilised for this study is given and the common methods for studying wind flows around buildings are briefly outlined. Lastly, the selection of the experimental methods is justified.

3.1.1 Chapter overview

This chapter is divided in four main sections (Box 3.1). The first section provides with an introduction on the selection of the project for this research and briefly describes its urban context. The second section (3.2) describes and justifies the selection of cases the nine cases used for this study.

BOX 3.1 chapter three overview

3.1 The site	The background of the research project and the description of the site where it is located.
3.2 The studied cases	The explanation and the rationale of the nine cases utilised in this study.
3.3 The methods	The description of the common methods used for studying wind flows around buildings and the justification for the ones used in this study.
3.4 The key points	Summarised in Box 3.2

In section (3.3) the common methods for studying wind flows around buildings are briefly outlined and the rationale for the selected methods for this study is provided. Additionally, some conclusions and key points are presented in section 3.4.

3.1.2 The studied building complex

Following the completion of the University of Nottingham's Jubilee Campus located in Nottingham, UK; many ideas were put forward for the development of the 6Ha brown field area next to it (52°57'5.41"N, 1°10'58.59"O), all of them were to include sustainability issues. Figure 3.1 shows an aerial view of the Jubilee Campus and to its right the plot of land used for design purposes in this study.

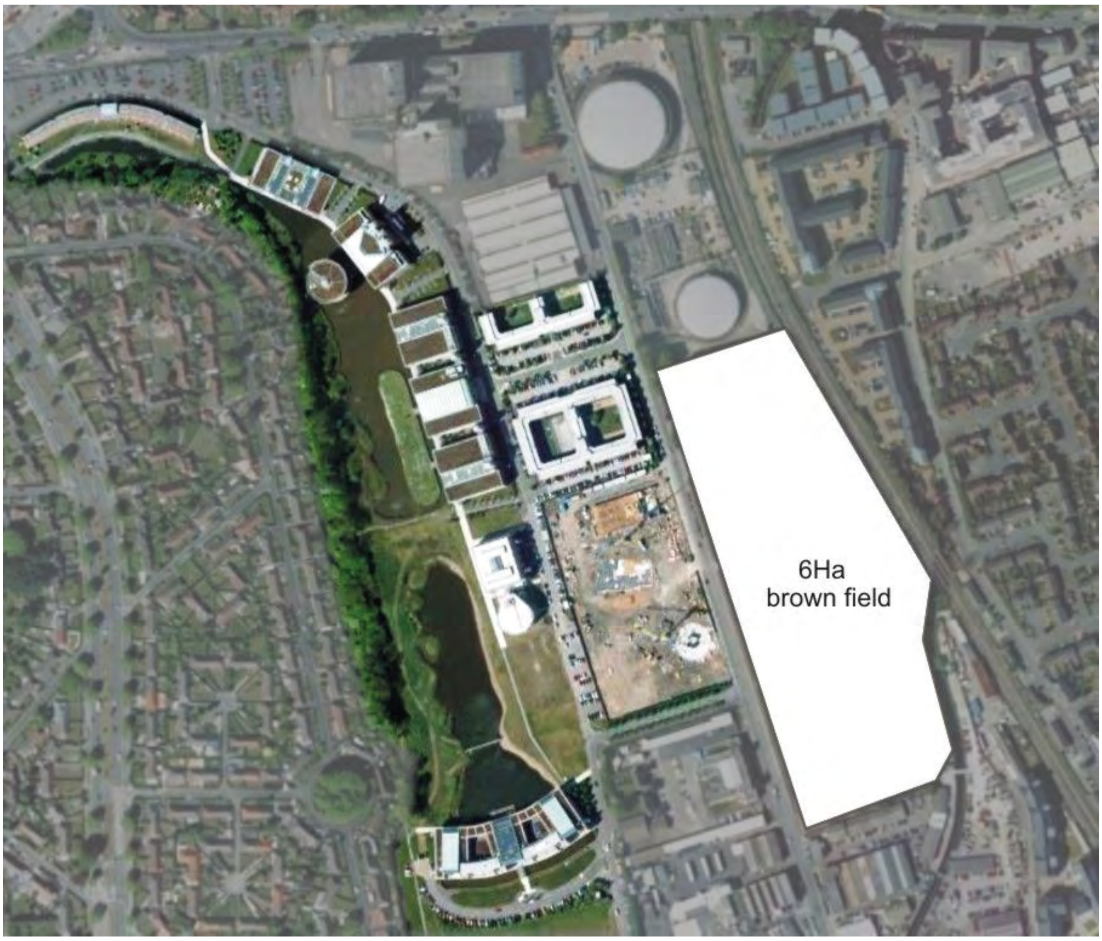


Fig. 3.1 Aerial view of Jubilee Campus and the adjacent 6Ha brown field site used for this study.

One of those ideas was to create a mixed use development that allowed students and staff for working and living on the site. The fundamental concept in the design brief was to achieve an autonomous energy community with zero carbon emissions by integrating energy efficient and renewable energy technologies as well as many other sustainability concepts.

A number of masterplans for the site were assessed by the relevant authorities at the University of Nottingham. Significant importance was to be given to the wind and solar resources as an integral part of a wider environmental strategy.

The selected plan consisted on 40 detached houses, 8 four-storey buildings for offices and classrooms and 7 fifteen-storey towers for student accommodation. The selected building complex used for the wind tunnel and computational modelling in this study was the group of seven fifteen-storey towers.

3.2 Description of the nine study cases

Nine layout configurations are proposed for this study. They follow the design objectives of harnessing the prevailing winds to enhance the use of wind energy and to maximise the use of solar energy. Parallel building configurations were discarded at early stages, as those configurations are more likely to cause over-shading[I] and uncomfortable wind environments at pedestrian level.

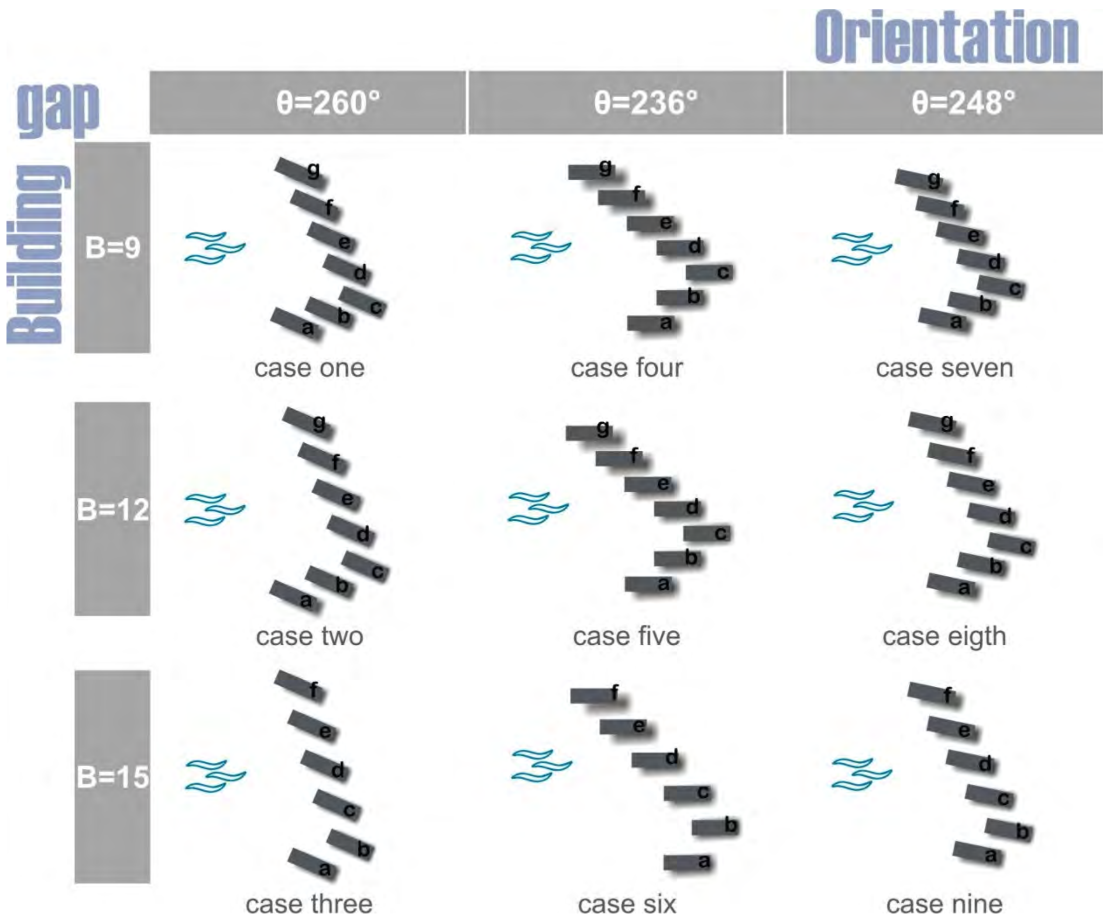
It has been addressed that building orientation, building shape and the separation between buildings define to a great extent the wind environment around buildings. In order to investigate possible locations for small wind turbines between the 7

[I] Studies before and during the Second World War on the relation between density and daylight showed that the corridor type street was one of the least efficient from daylight point of view. It was found that by rotating every other building through 90°, so that adjacent buildings were at right angles, there was a marked improvement in daylighting for the same density. This suggested the open plan layout in which buildings would be taller, drawn back from the street and concentrated in the central parts of sites. Such considerations, lead to the present situation where small groups of towers dominate the urban scene. See for example Wise et al[1].

studied buildings, two parameters were changed: building orientation (θ) and the separation between buildings or street canyon (B).

The combination of three different orientations (θ) and three different building gaps (B) produced the 9 cases examined in this study as illustrated in Table 3.1.

Table 3.1 Studied cases.



The choice of the studied variables was constrained to the sustainable design brief, which included energy generation from solar and wind sources. The relevance of a good choice on the site layout is acknowledged by many authors[2],[3],[4],[5]. It is regarded as the most important factor affecting the duration of sunlight in buildings and having a substantial impact on solar and wind design. Therefore, even though this study only deals with wind energy generation, the selection or orientations and distances between buildings was made in a way that all the options guaranteed optimum use of solar and wind resources.

The variables tested on this study were conceived as part of the iterative design process at early masterplanning stages, where the proposals establish the principles of land use for the area, including densities, orientation, grids and blocks. It is only by this iterative process that the results obtained with the simulations can be translated into the architectural design, answering the research questions formulated in chapter one.

According to CABE[6] it is necessary to present these proposals – nine cases in this study - as three-dimensional models, showing the key urban design principles[II]. Figure 3.2 illustrates the 3D modelling of the studied building complex and the surrounding buildings. The proposed buildings are sharp-edged for two reasons; firstly, they correspond to the vast majority of existing buildings and secondly, they represent a greater opportunity for retrofitting small wind turbines, giving a wider applicability to the findings of this research.

[II] Those urban design principles include the proposed massing, height, densities, orientation, grids and blocks, (without architectural or style details).

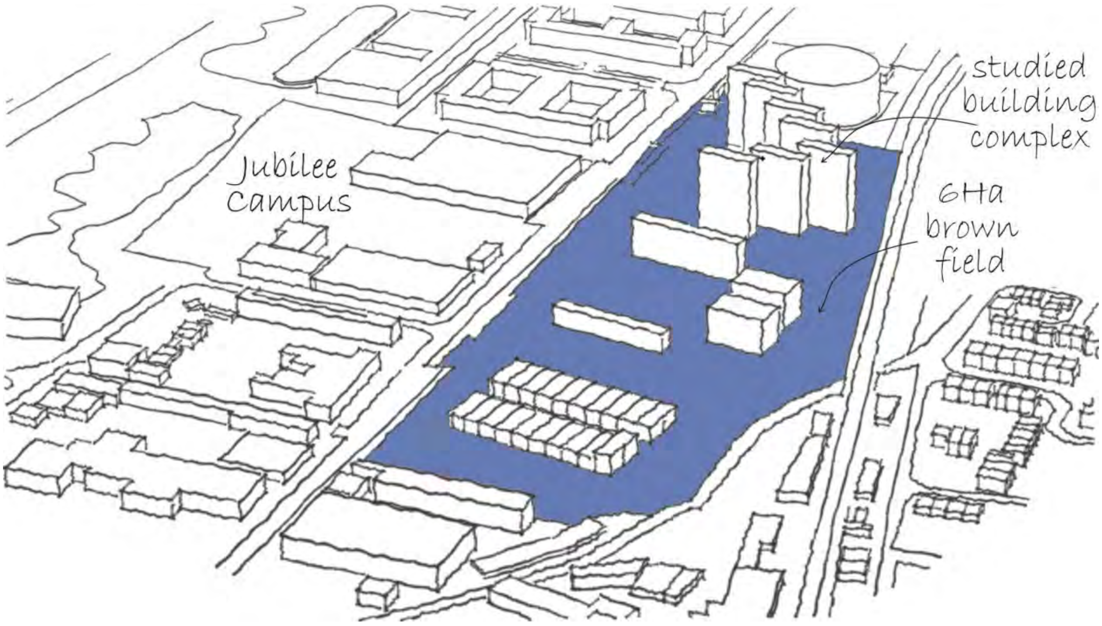


Fig. 3.2 Three-dimensional model of Jubilee Campus and the adjacent 6Ha masterplan.

3.2.1 The selection of the orientation (θ) variables

The key to optimising the solar potential of the site is to orientate buildings broadly to the south. This tends to result in an East -West street pattern. It is possible to move up to 30° away from due south and yet have 90-95% of the maximum output of a PV module or a solar collector[7]. Ideal building orientation for solar gain is to stay within 15-20° of due south.

Following these recommendations, the intention of the first orientation is to achieve maximum incident solar radiation in winter whilst avoiding overheating during summer months. To obtain this orientation a calculation was made using ECOTECT V5.2 software. The weather data for Nottingham site was used. According to the calculation, orientation (θ= 260˚) is ideal to achieve maximum incident solar radiation in winter and it will also avoid overheating in summer months.

The second orientation of θ= 236˚, was selected so that the buildings' long façades were parallel to the prevailing winds. Wind data for Nottingham, UK were obtained as a TRY file[8] and then exported to ECOTECT V.5.2 software to be later analysed with the Weather Tool of the same software.

According to the frequency distribution of wind speeds for Nottingham, UK presented in chapter two[III], it can be appreciated that the prevailing winds approach from South-West and West orientations (202.5° <θ <270°). In this way, the selection of the second orientation as θ= 236˚ locates the buildings in a good position to harness the prevailing winds.

The third orientation of θ= 248˚, corresponds to the midpoint between θ= 260˚ orientation (better for solar radiation collection) and θ= 236˚ orientation (better for wind collection).

The three orientations are in the prevailing winds range of 202.5° <θ <270°. At the beginning of this study, it was expected that the interaction between the orientations, the building separation and the variation in height would produce variations on the wind conditions that would favour some locations more than others for small wind turbines between the buildings.

3.2.2 The selection of the street canyon (B) variables

Along with the orientation, the distance between buildings was to be varied to enhance wind speed at heights suitable for siting wind turbines, whilst avoiding

---

[III] Figure 2.5, page 55

uncomfortable wind speeds at pedestrian level. At the same time, the layout has to allow inclusive access to and through the site. The distance between buildings had to be such that allowed for the collection of solar radiation without overheating or causing over-shading.

Rules of the thumb regarding solar access are that passages of 7 to 8 metres between three stories buildings will produce a 60 – 70% loss of total annual solar radiation, while streets of 13 to 14 metres will generate a 30 – 40% loss of total annual solar radiation[7].

Nevertheless, strict adherence to solar access can serve to space buildings further and further apart – lowering densities and weakening street enclosure. Figure 3.3 shows the shadow study for cases one, four and seven. The images on the left side correspond to June 21st at 11am and images on the right side correspond to a simulation of solar access for December 21st at the same hour.

In masterplanning terms, the distance between buildings (B) determines the access to buildings by pedestrians, cycles, car movement corridors, streets, trails, etc. the pedestrian realm should provide of a sidewalk on each side of the street, often accompanied by shade trees and places of rest.

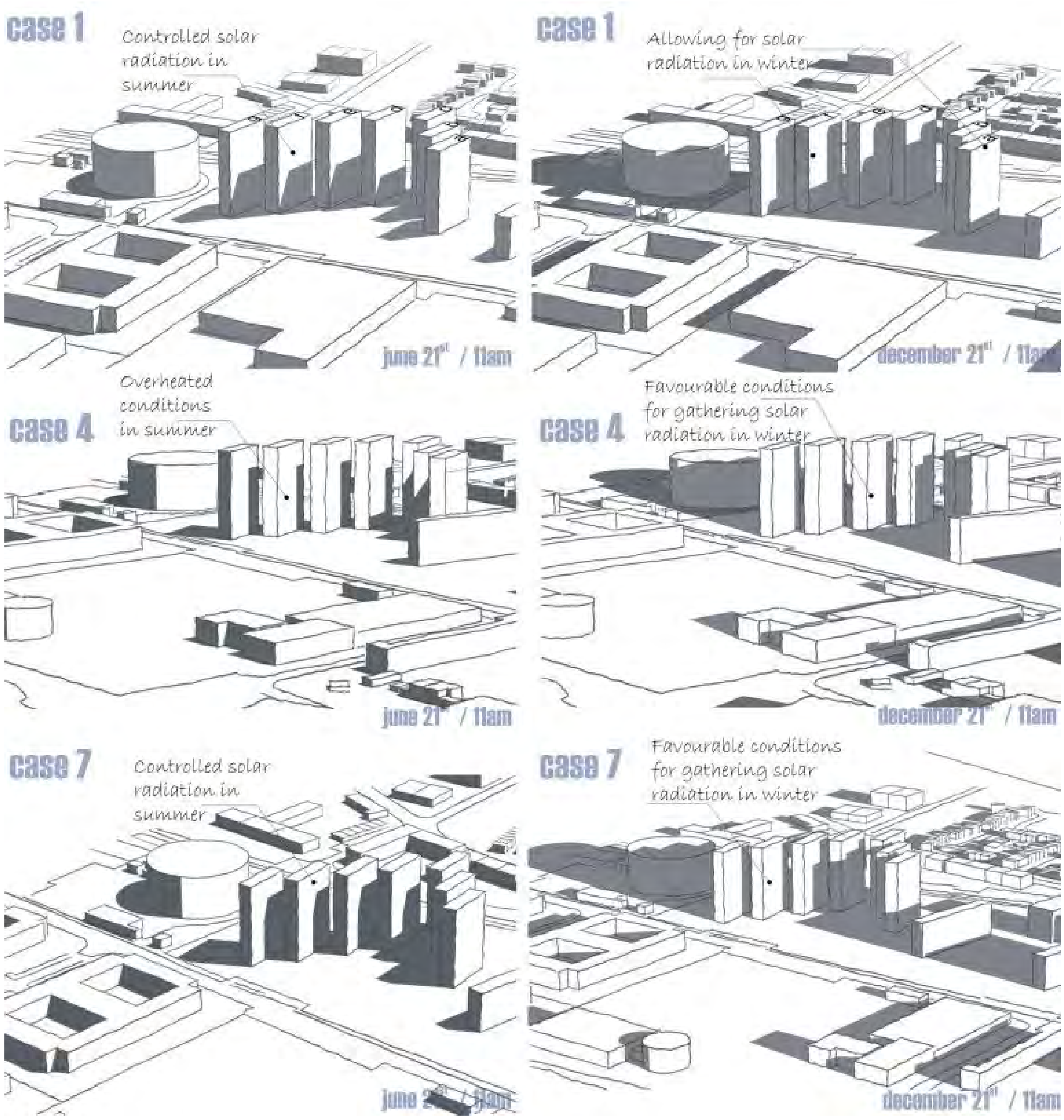


Fig. 3.3 Solar access in Summer and Winter for cases one, four and seven.

The selection of the first distance (B=9) was made so the buildings were closer together as possible and expecting to enhance wind velocity between the buildings. This distance will also allow for a generous shared-use pathway. These pathways usually accommodate two-way travel and are constructed with up to 5 metres in width (to facilitate passing and mixing of modes) and up to 2 metres in width (for bicycle parking[9]).

The second distance of B=12 metres would also allow for pedestrian and bicycle transit and additionally it can accommodate for a shared green space.

The third choice of the distance variable of B=15 metres, is considered as a comfortable space between buildings that allows for pedestrian, bicycle and car transit, whilst still providing with sufficient enclosure to the site. In terms of wind energy generation, the maximum separation between buildings is to be so, that the interaction between buildings cannot be neglected and provided with the opportunity to locate small wind turbines between buildings.

3.3 Methods for studying wind flows around buildings

The study of airflows in and around buildings is required to perform various studies such as heat and mass transfer analyses, thermal comfort assessments, indoor air quality studies, system control analyses and contaminant dispersal predictions, among others. The use of research tools to gain understanding of airflows and building aerodynamics is therefore of great importance.

The research tools used to understand problems of building aerodynamics can be divided into three great groups[10]:

- Mathematical models
- Experimental (both full scale and scale models)
- Simulation tools or Computational Fluid Dynamics (CFD) calculations

All of them have specific advantages and drawbacks that define the suitability of the tool for a certain analysis.

3.3.1 Mathematical models and experimental methods

Mathematical models are based on the Navier-Stokes equation[IV], which describes the flow behaviour. There are a number of flows that can be approximated with a special case of the Navier-Stoke equation (the Euler Equation), in which the flow is regarded as homogeneous and inviscid. Nevertheless, this method is difficult to use, extremely time consuming and requires a thorough knowledge of fluid dynamics.

Experimental methods include full-scale measurements and scale-modelling techniques (i.e. wind tunnel experiments).

Full-scale measurements of the wind approaching over variable terrain and flowing through urban environments are perhaps the most accurate way to obtain information of the wind environment on a particular site. Besides, these measurements are required to validate data obtained from scale-modelling techniques. The major drawback of this approach is that this is a very difficult and expensive experiment to carry out even once, yet along, many times. The Building Research Station in the UK carried out extensive investigation on full-scale experimentation and wind tunnel data validation during the 60's[V]. Their full-scale experiment at Aylesbury is described in detail by Lawson[11].

Wind tunnels have been widely utilised for the study of wind flows around buildings and the data obtained from the experiments is – under certain circumstances – considered reliable. It is less expensive than the full scale measurements and can

---

[IV] For further information on fluid dynamics see Batchelor[12] and for the application to building aerodynamics see Lawson[13].

[V] Amongst them were: Royer House and the GPO Tower in London and a group of terrace houses at Aylesbury.

allow for the visualisation of streamlines. It also has the advantage that it can be used in early design stages, and by testing different model configurations, the user can take an informed design decision.

3.3.2 Simulation tools or CFD calculations

Building simulation tools have become part of the many computer applications for the building design industry. Moreover, simulation tools can provide with a better understanding for the consequences of design decisions[14].

Within this thesis, the term Computational Fluid Dynamics (CFD) is used to describe a variety of techniques used to solve and analyse problems in fluid dynamics with the use of computers and special CFD codes or software. The usefulness of CFD software is a direct function of its reliability and affordability.

CFD calculations are used to verify the results obtained with mathematical models and wind tunnel tests. Although the underlying physical equations and the solution techniques vary, they commonly involve the replacement of the governing flow equations with a discrete representation and the numerical solution of these approximate equations using a computer. This discretisation process means that in all cases the solutions obtained are approximate[15].

CFD simulations are not straightforward to perform and their results still have to prove their extended acceptability. Furthermore, fluid flow processes are physically complex and in certain cases the governing equations are only an approximate representation of the true physical processes. This level of uncertainty and the overall quality and reliability of CFD simulations have been extensively discussed by Casey and Wintergerste[VI] and many other authors [16],[17],[18],[19],[20].

In addition to the source of errors and uncertainty that are introduced by the numerical model, the CFD user can also introduce errors and uncertainties, making the process of performing a CFD calculation even more complex.

The deficiencies or inaccuracies of CFD simulations can be related to a wide variety of errors and uncertainties. The definitions used in this thesis follow closely the definitions given in the AIAA[21] guide, as described in chapter five.

The main advantage of CFD modelling is that it is less expensive than full scale measurements and wind tunnel experiments. Some major drawbacks are that, in order to produce reliable simulations, sensible knowledge of fluid dynamics is required; proficiency in the use of the specific and thoroughly validated CFD code is needed along with related computational skills.

3.3.3 Selection of the research tools

This thesis deals with an hypothetical set of buildings therefore, wind tunnel (chapter four) and Computational Fluid Dynamics CFD (chapter five) techniques were selected as the research tools for modelling wind environmental conditions around a variety of building configurations.

[VI] Casey, M. and Wintergerste, T. (Eds) (2000) Best Practice Guidelines, ERCOFTAC Special Interest Group on Quality and Trust in Industrial CFD, ERCOFTAC Triomflaan 43, B-1160, Brussels.

The purpose of the tests was to examine wind flow conditions, specifically wind velocity around the nine proposed building configurations. The tests were done in order to investigate possible locations for small vertical wind turbines. The examined configurations included three variations in the distance between buildings (street canyon) hereafter referred with the letter B, and three variations in building orientation (incidence angle), hereafter referred as θ.

The CFD technique was applied as a predictive tool. The numerical findings generated with the CFD simulations were discussed in chapter six against the experimental data obtained with the wind tunnel. The purpose was to be able to draw some conclusions on the siting of small vertical wind turbines near the buildings and its consequences on building design.

Due to the different nature of the selected tools, their particular methods are briefly explained in the corresponding chapter, i.e. chapter four includes the theory and methods used for wind tunnel experiments; similarly, chapter five describes the considerations that should be observed when performing a CFD simulation.

3.4 Chapter three – key concepts

The interaction of the building orientation, height and form, along with the local landscape and prevailing wind conditions, changes significantly the wind environment making it unique and complex. This is why it is best to carry out the studies of wind environment at master-planning stage; where factors potentially affecting a particular area of interest can still be changed without affecting the overall scheme.

To accurately study wind conditions around buildings a distinctive scale modelling process is required. Traditionally it has been carried out in boundary layer wind tunnels. These wind tunnels, however, are not commonly available for architects and engineers, mainly because of the high costs associated with the construction, operation and maintenance of these facilities. It is, therefore, difficult for designers to justify the costs and time involved in a wind tunnel test, especially at design stage, when they are most likely to benefit from them in order to make informed design decisions.

CFD techniques present an alternative to wind tunnel tests that have become increasingly popular for simulating wind conditions around buildings. The advantages of using CFD are that they can provide with much more information in less time and for a fraction of the cost than the equivalent in wind tunnel tests. Clearly, this leads to a substantial reduction of time and money during the design process.

BOX 3.2 chapter three key concepts

On the research approach used in this study

- A group of seven fifteen-storey buildings was analysed in this study.
- In order to investigate possible locations for small wind turbines, a number of layouts were suggested and tested.
- Two parameters were varied: Building orientation (θ) and the gap between buildings (B).
- Using three different building orientations (θ) and three different distances between buildings (B) a total of nine cases were produced to be tested.
- The selected methods to analyse wind regime between buildings were boundary layer wind tunnel and computational fluid dyamics, presented in chapters four and five respectively.

3.5 Chapter three – reference list

1. Wise, A., Sexton, D. and Richards, D. (1971) Effects Due to Groups of Buildings [and Discussion]. *Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences (1934-1990)* **269** (1199):469-485.
2.
2. Littlefair, P. (1998-) Passive solar urban design: ensuring the penetration of solar energy into the city. **2** (3):303-326.

3. CABE (2002) By design. Better places to live. *Commission for Architecture and the Built Environment.* A companion guide to PPG3. 108 pp

4. CABE and BioRegional (2008) What makes an eco-town? *Commission for Architecture and the Built Environment.* 32 pp

5. Wizelius, T. (2007) *Developing wind power projects. Theory and practice.* UK: Earthscan.

6. CABE (2008) Creating successful masterplans. A guide for clients. *Commission for Architecture and the Built Environment.* 159pp. Available on line at: http://www.cabe.org.uk/

7. English Partnerships and Housing Corporation. (2000) Urban Design Compendium. 125 pp London: Llewelyn-Davies.

8. CIBSE/Met Office (2003) Test Reference Year data for Nottingham, UK. Nottingham TRY/DSY Hourly Data Set (CD-ROM)

9. Walters, D. (2007) Designing Community. *Charrettes, masterplans and form-based Codes.* 283 pp London: Architectural Press. 978-0-7506-6925-2.

10. Mertens, S. (2006) *Wind energy in the built environment*. Multi-Science. Essex, UK. 169 p.

11. Lawson, T.V. (1971) Landscape Effects with Particular Reference to Urban Situations. *Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences (1934-1990).* **269** (1199):493-501.

12. Batchelor, G.K. (1988) *An introduction to Fluid Dynamics* , England: Cambridge University Press.

13. Lawson, T.V. (2001) *Building Aerodynamics*, London : Imperial College Press.

14. Augenbroe, G. (2004) Trends in building simulation. In: Malkawi, A.M. and Augenbroe, G., (Eds.) *Advanced Building Simulation*, pp. 4-24. New York, UK: Spon Press]

15. WS Atkins Consultants. Best practice guidelines for marine applications of Computational Fluid Dynamics. Marnet-CFD. 84 pp. Available at: <https://pronet.wsatkins.co.uk/marnet/publications/bpg.pdf> Accessed: 17/07/07

16. Sørensen, D.N. and Nielsen, P.V. (2003) Quality control of computational fluid dynamics in indoor environments. *Indoor Air* **13** (1):2-17.

17. COST (2007) Best practice guideline for the CFD simulation of flows in the urban environment. Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B., (Eds.) COST Action 732. Quality assurance and improvement of micro-scale meteorological models. 52 pp. Hamburg, Germany:

18. Schatzmann, M. and Britter, R. (2005) Quality assurance and improvement of micro-scale meteorological models. *COST 732 report. European Science Foundation.* 6 pp.

19. de Wit, S. (2004) Uncertainty in building simulaton. In: Malkawi, A.M. and Augenbroe, G., (Eds.) *Advanced Building Simulation*, pp. 25-59. New York, UK : Spon Press]

20. Chen, Q. and Zhai, Z. (2004) The use of Computational Fluid Dynamics tools for indoor environmental design. In: Malkawi, A.M. and Augenbroe, G., (Eds.) *Advanced Building Simulation*, pp. 119-140. New York, UK: Spon Press]

21. AIAA (1988) AIAA guide for the verification and validation of computational fluid dynamics simulations. *American Institute of Aeronautics and Astronautics.* AIAA G-077-1998.

4 wind tunnel experiment

4.1 Introduction

This chapter presents the wind tunnel experiments, which were conducted for nine different building configurations, to investigate the influence of building orientation (θ) and street width (B) on wind velocity at different heights using hot wire anemometers, which were previously calibrated.

4.1.1 Chapter overview

This chapter is divided in four main sections (Box 4.1). The first section describes how the use of wind tunnels for building aerodynamics has evolved over the years. It also provides a general overview of the relevant theories and methodologies related to the use of wind tunnel modelling and some practical issues that need to be taken into account when using wind tunnel scale modelling techniques for the experimental measuring of wind speed in urban environments.

BOX 4.1 chapter four overview

4.1 The background	An overview on past uses of wind tunnels is provided, along with the theory and requirements for a sucessful wind tunnel scale modelling.
4.2 The experiment	A detailed description of the measurement and data acquisition equipment used and how the scale model was set up and instrumented for measuring wind velocities.
4.3 The results	The measured effects of builing orientation, height and street cayon on wind velocity are presented and possible locations for small wind turbines are suggested.
4.4 The key points	Summarised in Box 4.2

The second section is a description of the wind tunnel and the measurement and data acquisition equipment utilised for this research. In addition, a detailed explanation of the employed scale models and how they were instrumented for wind velocity measurement is presented.

The results of the wind tunnel experiment are presented in the last section of the chapter along with some conclusions. These results will be presented in chapter six and will be discussed against those found with the CFD simulations described in chapter five.

4.1.2 Historical use of Wind tunnels

Building aerodynamics is an experimental area that has emerged of aircraft aerodynamics and wind tunnels are used to produce a variety of studies regarding wind flow. Nevertheless, there are many differences regarding the use of wind tunnels between those disciplines.

A large amount of development in wind tunnel techniques occurred during the Second World War due to the need for faster fighter aircrafts. Post war use of wind tunnels became slightly redundant and, as a result, they began to use it in other fields, including the built environment.

The wind tunnel techniques developed by aircraft engineers that solved their problems satisfactorily, had to be adapted to meet the new demands of building aerodynamics.

According to Lawson[1], the fundamental differences that demanded the development of new wind tunnel techniques are:

- The description of the wind around buildings is very complex, compared to the still flows around aircrafts.
- Wind around buildings can approach from any direction and has shear and turbulence that can only be described in statistical terms.
- The shapes in the built environment are bluff shapes which cause the air to separate as it flows around them, rather than smooth sleek shapes, like wings and fuselages, to which the flow is attached.

Before the importance of shear was appreciated, aircraft type wind tunnels, which do not contain shear, were used and the data acquired from them were not accurate for the study of flows around buildings.

Despite those barriers, wind tunnels have been used to study the fluid flow dynamics of scale buildings since the early 1930's as reported by Davenport[2] and Cook[3]. Additionally, a great amount of research on building aerodynamics was conducted in the UK for wind tunnel studies at the Building Research Station during the 60's by Sexton[4], Wise[5] and Lilly-White; among others, and at the department of Aeronautical Engineering at the University of Bristol by Lawson[6].

Since then, boundary layer wind tunnels have been used to investigate different effects of wind on buildings and structures. Among them there are studies for:

- Local exterior pressures on walls, cladding and roof components[7],[8],[9] where mean and fluctuating local exterior pressures on walls, cladding and roof components are studied.
- Area and overall wind loads[10],[11] to test wind loads on specific tributary areas, using pressure models and spatial or time averaging of simultaneously acting local pressures.
- High frequency force balance [12]. High-frequency base balance tests are commonly used to evaluate loads associated with the fundamental sway and torsional modes of tall buildings.
- Section model tests are conducted using dynamically mounted models. It is common for the study of bridges but also for other structures.
- Aerolastic studies[13] provide information on the wind-induced responses of buildings, structures and objects that move relative to the wind flow.
- Pedestrian winds[14] studies use scaled static models to investigate character of flow around buildings and structures and to measure local wind speeds and directions required for environmental assessments.
- Air quality / dispersion of pollutants[15],[16],[17]. These tests evaluate the dispersion of pollutants and determine the resulting air quality around buildings and in urban areas.
- Terrain and topographic studies[18] are conducted to assess the wind energy potential of particular sites and correlations of flows at different location and heights.
- Natural ventilation/ indoor air quality[19],[20],[21]. This kind of studies estimate wind induced interior pressures including fluctuations in the presence of significant openings.

These and other numerous similar studies have led to a general recognition that boundary layer wind tunnels and building scale models are valuable tools to provide quantitative analysis of airflows around buildings.

4.1.3 Theory and development of wind tunnel scale modelling

Wind tunnel scale modelling requirements

Successful test programs begin with a well designed model. The model should allow for easy change of parts and provide for means of precisely aligning the model in the tunnel and change the required parameters. Besides these considerations, when carrying out the wind tunnel experiment, deciding the model scale is usually one of the first issues that need to be resolved. In order to investigate wind flows around buildings in urban environments, a model of the specific group of buildings to be studied and the adjacent buildings have to be reproduced with adequate detail. The amount of detail required in the models will be discussed later in this chapter.

In addition to the specific building complex and adjacent buildings, Aynsley et al[22] suggest that it is essential to model all major buildings for at least 800m, approximately, the equivalent of eight blocks. Nevertheless, Lawson[23], indicates that it would be ideal if five streets in all directions from the building complex of interest were modelled, but as this can rarely be achieved in practice, a minimum of two streets should be modelled. Ultimately, the linear scale for the model is usually determined by the extent to be modelled and the size of the working section of the wind tunnel.

In order to directly measure wind speed of a scale model in the wind tunnel and relate this to the full scale building there are a number of factors that must be considered, being similarity the most important issue among these. Similarity in scaling can be achieved in different grades [22]:

- Geometric similarity, where ratios of linear dimensions are equal
- Dynamic similarity, where the ratios of forces are equal
- Kinematic similarity, where particle paths are geometrically similar

Consequently, mean velocity and turbulence intensity profiles must be correctly reproduced in order to accurately model the turbulent flow conditions induced by buildings[22].

Similarity requirements

Geometric similarity

The geometric scale of the model of a building should be chosen to maintain, as closely as possible, equality of model and prototype (full scale) ratios of overall building dimensions[24]. It should be noted that this does not mean that the model building needs to fully model every architectural detail of the prototype building, but it must include the features that will affect its airflow characteristics, such as canopies, overhangs, and arcades.

Appropriate scaling ratios have to be considered in a way that the model size does not cause a blockage of the airflow through the tunnel. It is good practice for wind tunnel models to have a windward façade visible area of less than 5% of the overall wind tunnel cross sectional area to ensure that the airflow around the model is not affected. A greater value will require blockage corrections[23] or validation of the data must be ascertained through additional tests at a smaller scale[24].

The effect of models with relative dimensions larger than this would force the simulated wind to be squeezed between the building and wind tunnel walls in an unrealistic manner. This puts the scale limits for the model between 1:150 and 1:500 on an average wind tunnel, although scales of 1:1000 can be tolerated under certain circumstances[23]. Providing model scale has been appropriately considered, it is possible to achieve geometric similarity.

Dynamic similarity

In order to achieve dynamic similarity, it is very important that the fluid flow properties in the model are representative of those in the full scale building. A powerful tool to describe the characteristics of wind flow around buildings and structures is the dimensional analysis, therefore it is commonly used in engineering design; it allows formulating a relation between sets of dimensionless groups of variables which number less than the dependent variables[22].

If the values of the non-dimensional groups are the same in the model and full scale systems, then both systems will be dynamically similar[1]. In building aerodynamics, the dimensionless coefficients most commonly used are the pressure coefficient, the Jensen number (J_e), Strouhal number (S_t) and the Reynolds number (R_e)[i] among

[i] See for example, Aynsley et. al.[22] for a thorough explanation on dimensional analysis and the use of dimensionless coefficients.

others. The use of dimensionless coefficients allows simplifying data representation, enabling important parameters to be rapidly identified [25].

Probably the most important of those dimensionless coefficients is the Reynolds number. This is because with the same fluid at full scale and scaled model, the R_e number changes proportionally to the characteristic velocity times the characteristic size. This means that for a scaled version of a building, the characteristic size can be a factor 200 smaller (1:200). In order to maintain the same R_e number, the velocity should then be 200 times higher, this is unfeasible to achieve in any wind tunnel simulation. The R_e number at the scaled model will therefore differ from the full scale model. This R_e change can only be ignored if the measured properties are R_e-independent. Such a R_e - independent property is for instance C_d (Drag coefficient) of some bluff bodies[26].

This is the reason why it is concluded that the effects of scaling should be carefully considered in any wind tunnel experiment, because they can prevent a direct translation to the full scale properties.

Therefore, if the boundary conditions are shown to be similar, then the fluid flows in those systems can be assumed as similar. The boundary conditions that should be considered include the relative velocities, physical boundaries (i.e. geometric similarity of the studied building complex and surroundings), temperatures and turbulence characteristics of the flow systems[27].

4.2 Experimental procedure

4.2.1 Wind tunnel specifications

The open jet boundary layer wind tunnel at the School of the Built Environment (SBE), in the University of Nottingham (UoN), was used to carry out the wind tunnel experiments described in this chapter. The SBE wind tunnel has been previously used for scale model studies of natural ventilation[28],[20], indoor air quality and pollutant dispersal from buildings[19].

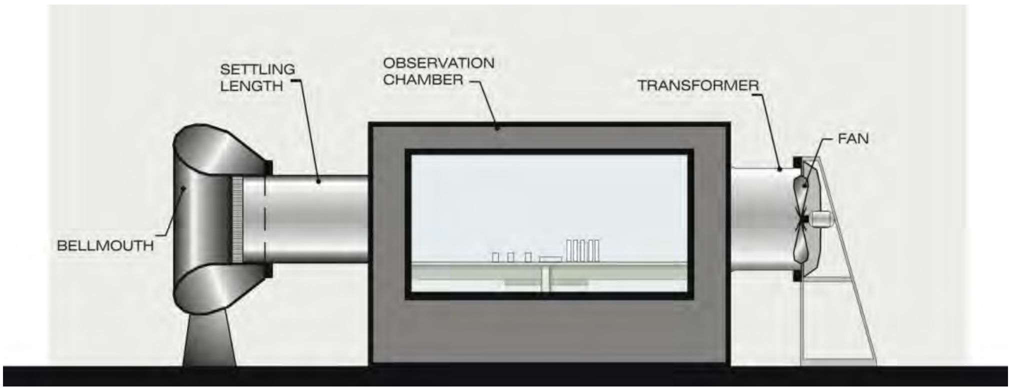


Fig. 4.1 Schematic section wind tunnel at SBE.

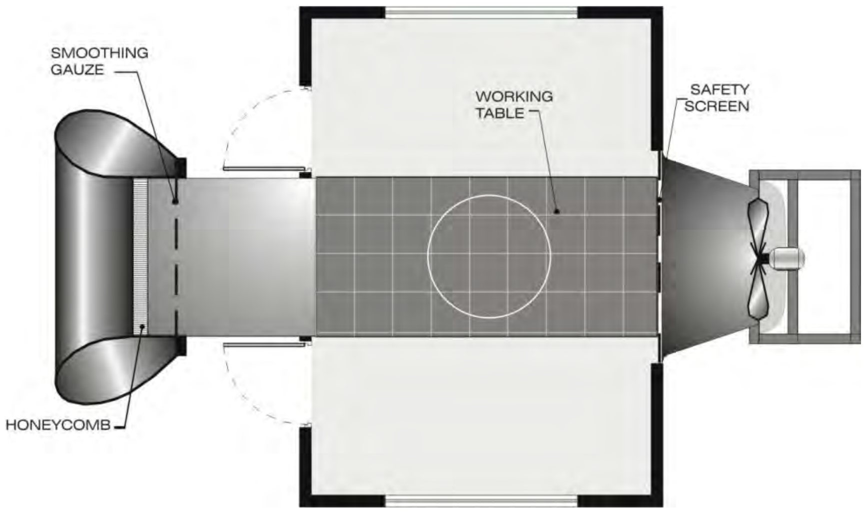


Fig. 4.2 Schematic plan wind tunnel at SBE.

Figure 4.1 shows a schematic section of the SBE wind tunnel and principal components. Similarly, in Figure 4.2, a plan view of the SBE wind-tunnel is illustrated. Open jet wind tunnels are less sensitive to flow blockage than wind tunnels with closed test sections, and this is an open jet tunnel based on the system described and utilised by Sexton[4].

In the SBE wind tunnel, air flow is drawn into the wind tunnel through the bell-mouth entrance by a variable speed axial fan of 0.90 m diameter, capable of producing airflows within a range of approximately 0.5 to 4.5 $m.s^{-1}$

The incoming air is straightened so it flows parallel to the working section by a section of 12.5 mm width honeycomb cells. A gauze section is then used to smooth the airflow further before passing into the settling chamber. The settling chamber is a short square duct, 0.75 m in length, used to reduce turbulence and disturbances caused by the honeycomb and gauze sections.

At the end of the settling chamber lies a fence on the floor of the tunnel, above the fence there are a number of horizontal aluminium slats which allow modifying the velocity profile of the incoming airflow to produce an approximate boundary layer profile.

The air flow is then drawn across the working section which is 2.25 m in length, 1 m wide and 0.75 m high, without significant velocity variations[19]. This working section is located centrally inside an observation chamber in which the measuring and data acquisition equipment is also placed.

The transformer section, where the fan is mounted, acts as a funnel and converts the airflow from the working section to the fan mountings. The opening of the transformer is slightly larger than the working section as this allows for the expansion of the airflow across the working section.

4.2.2 Measurement and data acquisition equipment

Velocity forms part of the major variables requiring measurement in wind-tunnel investigations. Hot wire anemometers are among the most common instruments to perform velocity measurements.

Hot wire anemometers come in two different types, constant temperature and constant current. In this experiment, a standard hot-wire probe with Constant Temperature Anemometer (CTA) designed for measurements of velocity and turbulence in gas flows at moderate and low turbulence, was used to measure the wind velocity.

The measuring and data acquisition equipment constitutes a measuring chain and it is illustrated on Figure 4.3. This equipment consisted of a single-sensor miniature wire probe with probe support, cabling and vertical support, a CTA, a signal conditioner, a connector box, an analogue-digital (A/D) board converter and a computer.

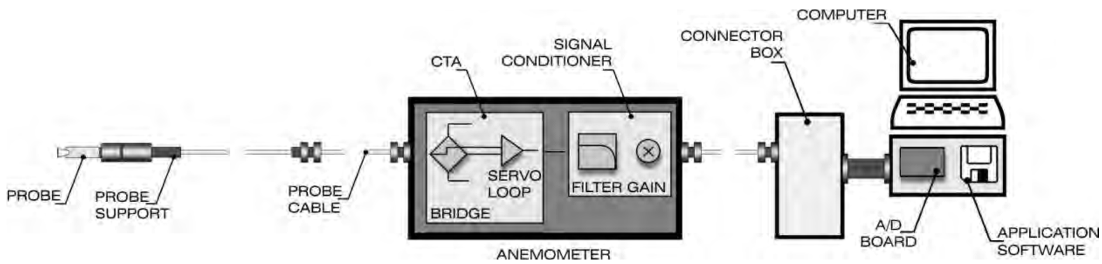


Fig. 4.3 Measuring and data acquisition equipment scheme.

The sensor used is the Dantec model 55P16[29]. This single-sensor miniature wire probe consists of a thin Pt-plated tungsten wire, with a diameter of 5 μm and a length of 1.25 mm suspended between two prongs. The wire is heated and maintained at a temperature of about 200 °C. As the wind blows the wire is cooled and the current is increased to maintain the constant temperature. The wind speed is related to the voltage output via 'King's Law' convective heat transfer equation. These wires have a very high frequency response >20 kHz and are therefore very sensitive to small fluctuations in the wind velocity. Accuracy is typically >1%. The sensor is built into one with its support and includes 1 m of coax cable with a bayonet Neill-Concelman (BNC) connector.

A stainless steel graduated vertical support was designed and constructed to hold the probe at specific heights.

The anemometer used is the Dantec Mini-CTA 54T30, a dedicated single-channel anemometer built into a small box. It contains a constant temperature anemometer circuit and a signal conditioner, which allows amplification and bandwidth filtering of the output signal.

The CTA signal is acquired via the Computer Boards' BNC-16SE connector box which incorporates a 16 front panel mounted BNC's allowing connection of up to 16 single-ended analogue inputs. The BNC-16SE was connected to the A/D converter board using the P5 out 32/64 channel and a C100FF-X 1-50 pin cable.

The A/D board installed in the computer is the Measuring Computing PCI-DAS1000. This board is a multifunction analog and digital I/O board designed for the PCI bus.

The signal was then saved as data-series in a computer using ComputerBoards DAS Wizard software that runs as a Microsoft Excel add-in.

4.2.3 Scale Model Instrumentation

Nine hypothetical configurations of buildings as thoroughly described in chapter three, were tested, with the main investigation centred on the building complex consisting of seven towers of 50 m height (H= 50). The plan layout of buildings used for this investigation is shown in Figure 4.4.

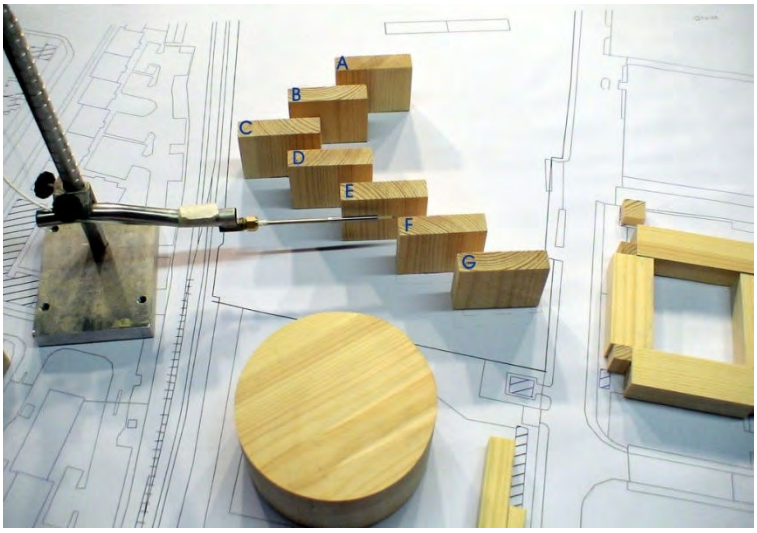


Fig. 4.4 Model and layout plan as positioned on wind tunnel working table.

A model of the cluster of buildings of interest A, B, C, D, E, F and G and its surroundings was made out of solid wood scale 1:500. Figure 4.5 illustrates the model positioned on the wind tunnel working table.

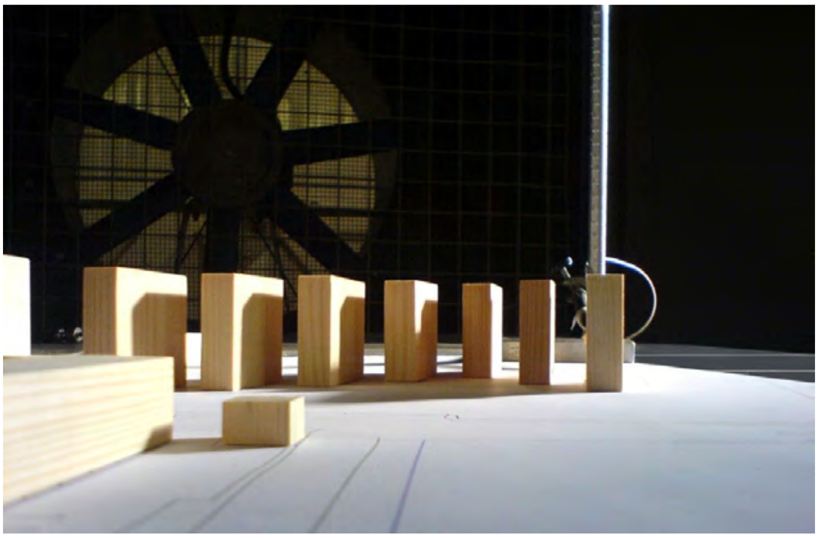


Fig. 4.5 Model on the wind tunnel working section.

In order to be able to easily modify the passage width between buildings (B) and the building orientation (θ), three master plans were plotted at the same scale of the model and attached to the wind-tunnel working section.

- In the first plan that was plotted, the first three cases were illustrated. The studied building complex and adjacent buildings in the first orientation θ= 260˚ with the respective passage widths: B= 9 m, B= 12 m, B= 15 m were included in the plan. Figure 4.6 shows case one (θ= 260˚, B= 9 m) with the studied building complex coloured in yellow.

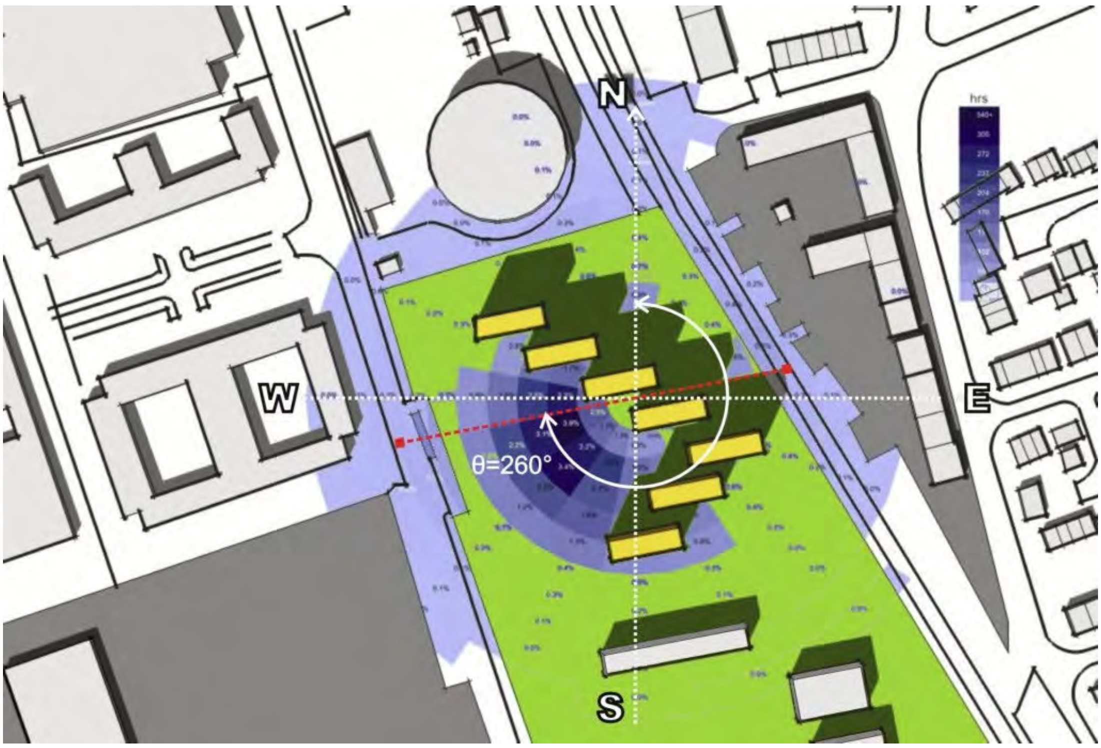


Fig. 4.6 Building complex and surroundings (θ= 260˚, B= 9 m).

- The second plotted plan, showed cases four, five and six, the studied building complex in the second orientation; where the long façades are parallel to the prevailing winds, θ= 236˚ and the gap between buildings: B=

9m, B= 12 m, B= 15 m. Figure 4.7 shows the studied building complex in case four (θ= 236˚, B= 9 m) and neighbouring buildings.

- The last three cases were plotted on a different plan, where the studied building complex was orientated at an angle of incidence θ= 248˚ and the passage between buildings was varied at the same distances as in previous cases. Figure 4.8 shows the studied building complex in case seven (θ= 248˚, B= 9 m) and adjacent buildings.

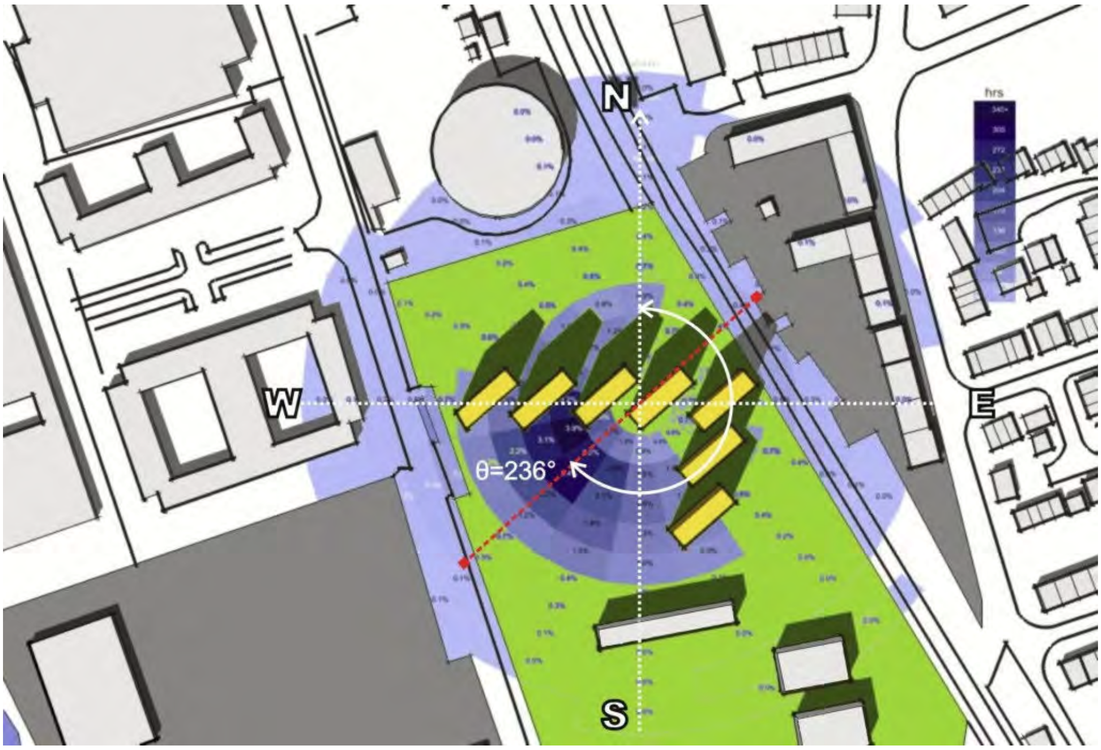


Fig. 4.7 Building complex and surroundings (θ= 236˚, B= 9 m).

Fig. 4.8 Building complex and surroundings (θ= 248˚, B= 9 m).

Depending on the building orientation (θ), the total model windward façade visible area varies from 2.5% for θ= 260˚ to a 2% for θ= 236˚ of the overall wind tunnel cross sectional area. The wooden blocks were then firmly fixed on top of the plotted plans and rotated and separated according to each case.

For the nine cases the tests were carried out as a series of velocity measurements at five points between buildings at different heights (h_1-h_5) as illustrated on Figure 4.9. The idea behind the selection of these measuring points was to investigate the variation of wind velocity for possible locations for small vertical wind turbines.

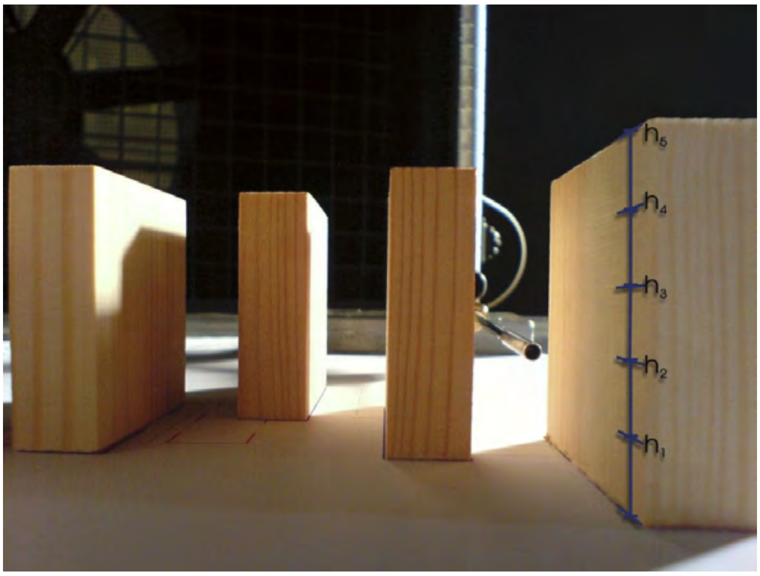


Fig. 4.9 Reference points for sensor location.

Measurements of atmospheric pressure and temperature were recorded prior to every run. An aneroid barometer was used for measuring atmospheric pressure in mmHg. Temperature was measured in °C using a mercury thermometer.

During the wind velocity measurements, the probe was mounted in the flow with the same orientation it had during the calibration.

4.2.4 Hot-wire calibration

Probes with identical sensors have similarly shaped calibration curves[30]. It is possible to create an individual transfer function on the basis of the output voltage from the probe at two or multiple known velocities ranging from the minimum to maximum velocities. This creates a unicurve transfer function for the specific probe type.

Directional calibration is not required for single hot wire sensors; therefore, prior to instrumenting the wooden model, only velocity calibration was carried out. In order to calibrate the hot wire, a pitot-static tube was used to measure the reference wind velocity (U_{ref}) of the incoming air at the centre of the working section.

The sensor probe was then exposed with the wire perpendicular to the flow and the prongs parallel with the flow to a set of known velocities to determine the transfer function. Figure 4.10 shows the calibration curve fitted to a fourth-degree polynomial. The hot wire calibration factors obtained are: -66.748772, 88.686611, -17.3449 and 5.061203.

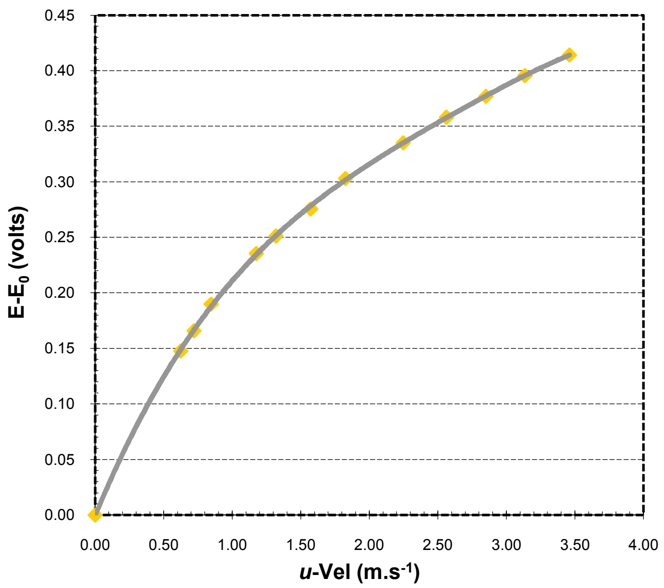


Fig. 4.10 Hot-wire multi point calibration curve with fitted 4th order polynomial.

4.2.5 Velocity Profile

As described before, the velocity profile at the SBE wind tunnel was produced by a series of aluminium slats and measured with a pitot-static tube located at the centre of the settling chamber (0.50 X 0.35 cms). The pitot-static tube measured the reference wind velocity (U_{ref}) of the incoming air. A series of velocity measurements (U) were taken at different heights with the hot-wire probe, being the results of the velocity profile shown as a ratio of these velocities to those measured at $U_{ref.}$

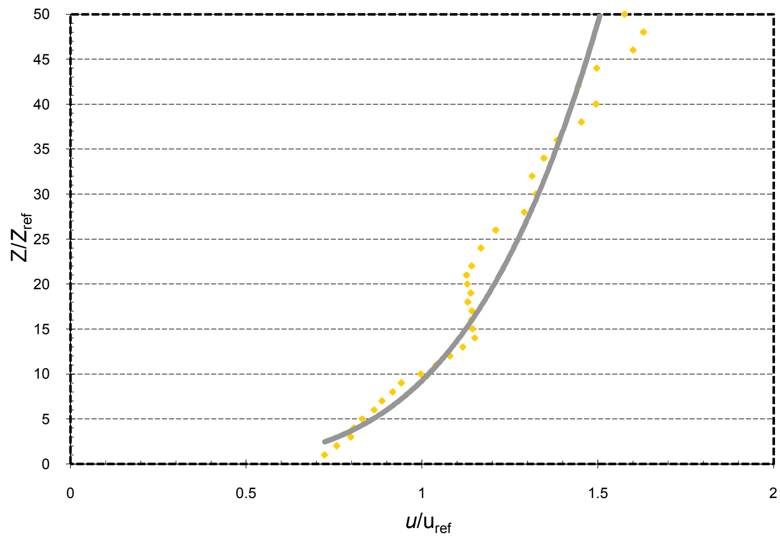


Fig. 4.11 Measured mean velocity profile fitted to a power law.

Figure 4.11 shows the measured points as a curve for the ratio U/ U_{ref} to a height of $50Z_{ref}$, which fitted to a power law curve. This indicates that the incident flow reasonably simulated a natural wind over an urban terrain with a stable wind profile characterized by the exponent α= 0.23 according to the power law of velocity profile:

(**Eqn. 4.1**)
$$\frac{U}{U_{ref}} = K\left(\frac{z}{Z_{max}}\right)^{\alpha}$$

4.2.6 Wind velocity measurements

The wind velocity was measured at five different heights between buildings with a CTA hot wire. The signal was sent to a spreadsheet using analog input tasks. The parameters for those tasks were generated using ComputerBoards DAS Wizard software that runs as a Microsoft Excel add-in.

Tasks were made with 1048 samples per channel and a scan rate of 30Hz. Air density (ρ) was calculated by converting atmospheric pressure (P_{atm}) and temperature (T) readings, taken prior to every run, to Pa and K, respectively and then using the equation of state for a perfect gas (Eqn. 4.2), where $\overline{R}$ is the specific gas constant for air.

(Eqn. 4.2)
$$\rho = \frac{P_{atm}}{\overline{R} \times T}$$

Wind velocity u was calculated converting the input voltage of each of the 1048 samples using the hotwire calibration factors applying the following equation:

(Eqn. 4.3)
$$u = CF_1\left(E_1 - \overline{E}_0\right)^4 + CF_2\left(E_1 - \overline{E}_0\right)^3 + CF_3\left(E_1 - \overline{E}_0\right)^2 + CF_4\left(E_1 - \overline{E}_0\right)$$

Where:

- CF_1, CF_2,…CF_4 are the hotwire calibration factors,
- E_1 is the sample input voltage
- And $\overline{E}_0$ is the mean input voltage at zero run

Wind velocities values of each sample were then averaged. Mean velocity values were divided by the reference velocity (U_{ref}) obtained using the equation:

(Eqn. 4.4)
$$U_{ref} = \sqrt{\frac{2\overline{q}}{\rho}}$$

Where, $\overline{q}$, is the dynamic pressure of the air as the result of its motion, measured at the centre of the working section and ρ is the air density calculated with Equation 4.2.

4.3 Results

These sets of experiments were relatively modest, where only mean velocity measurements were made at five different heights with a CTA hot wire. Tests were performed at a single wind velocity and the results are presented as ratios of wind velocity at those points to a reference wind velocity.

In order to be able to translate those ratios to full-scale, it is crucial to count with an appropriate statistical description of the local wind climate. For this study, wind data was acquired as a TRY file as explained in section 2.3.1. The idea is to determine from this velocity, the velocity of the wind at the measured points between the buildings, which is what the architect needs to know in order to make informed decisions on the building design.

Figure 4.12 (a-e) shows five surface graphs that illustrate the ratio u/U_{ref} at each of the five reference points (h_1-h_5) between the seven buildings for the nine cases.

As a general remark, as can be noticed in Figure 4.12 for all the reference points the ratio in between buildings F-G in cases three (θ= 260˚, B= 15 m), six (θ= 236˚, B= 15 m) and nine (θ= 248˚, B= 15 m), is equal to zero. This is because those cases represent the low density urban arrangements[ll], where the seventh building (G) was taken out of the layout plan as described in chapter three.

The ratio u/U_{ref} at these five heights ranges from 0.1 to 1.1. Velocity magnitude increases with height as expected but in different percentages at each measured point.

The velocity ratio at a normalized h_1 (z/H = 0.2) varies from 0.1 to 0.7, while at h_5 ratios show a wider range for the different cases from 0.1 to 1.1.

The location that showed less variation of velocity at different heights was between buildings A-B for incidence angle θ= 260˚ and B= 9 m (case one) where only an 8% increase in velocity ratios was recorded.

Table 4.1 summarises the points where measured u/U_{ref} ratios are optimum for small vertical wind turbine integration. The highest velocity ratio was measured between buildings D-E for incidence angle θ= 260˚ and B= 15 m (case three) at h_5 (z/H = 1). Other locations of high velocity ratios were recorded at h_3 (z/H = 0.6) between buildings D-E for incidence angle θ= 260˚ and B= 15 m and at h_5 (z/H = 1) between buildings D-E and E-F for incidence angle θ= 236˚ at B= 9 m and between buildings E-F for incidence angle θ= 248˚ at B= 9 m and B= 12 m.

[ll] Where density is measured by the ratio of area occupied by the buildings to the area of the site.

Depending on the passage width to building height ratio (B/H), the flow in the street canyons can be classified as skimming flow (B/H = 0 to 1.2); wake interference flow (B/H = 1.2 to 5) or an isolated roughness flow (B/H > 5:0) as originally proposed by Nakamura and Oke[31] and further adopted by Chang and Meroney[16] and Grosso[32] in their reports.

Flow in the street canyons between the seven buildings in all cases in the present study can then be regarded as skimming. This issue will be further discussed using CFD as a visualization tool to corroborate that the flow is indeed, characterized by stable vortices created in the space between buildings and to assess whether the flow appears to be skim on the crest of the elements.

It has been suggested by many authors[33],[34],[6],[1] that comparisons of wind tunnel data and full scale data are particularly important to assure that experimental techniques provide representative information. In this study, such comparisons were carried out with the CFD simulations presented in chapter five, and further discussed in chapter six; as this research deals with hypothetical building configurations.

Table 4.1 Measured velocity ratios suitable for wind turbine integration.

Velocity Ratio (u/U_{ref})	Case Number	Orientation (θ)	Gap (B) (m)	Height (z/H)	Buildings	Schematic Plan
0.9 - 1.0	1	260°	9	1.0	F-G	
0.9 - 1.0	1	260°	9	0.8	F-G	
0.9 - 1.0	2	260°	12	1.0	E-F	
0.9 - 1.0	2	260°	12	1.0	F-G	
0.9 - 1.0	2	260°	12	0.8	E-F	
0.9 - 1.0	2	260°	12	0.8	F-G	
0.9 - 1.0	2	260°	12	0.6	E-F	
1.0 - 1.1	3	260°	15	1.0	D-E	
0.9 - 1.0	3	260°	15	1.0	E-F	
1.0 - 1.1	3	260°	15	0.8	D-E	
1.0 - 1.1	3	260°	15	0.6	D-E	
0.9 - 1.0	3	260°	15	0.4	D-E	
0.9 - 1.0	4	236°	9	1.0	D-E	
0.9 - 1.0	4	236°	9	1.0	E-F	
0.9 - 1.0	4	236°	9	1.0	F-G	
0.9 - 1.0	4	236°	9	0.8	D-E	
0.9 - 1.0	7	248°	9	1.0	E-F	
0.9 - 1.0	7	248°	9	1.0	F-G	
0.9 - 1.0	7	248°	9	0.8	D-E	
0.9 - 1.0	7	248°	9	0.8	F-G	
0.9 - 1.0	7	248°	9	0.6	E-F	
0.9 - 1.0	7	248°	9	0.6	F-G	
0.9 - 1.0	8	248°	12	1.0	D-E	
0.9 - 1.0	8	248°	12	1.0	E-F	
0.9 - 1.0	8	248°	12	1.0	F-G	
0.9 - 1.0	8	248°	12	0.8	D-E	
0.9 - 1.0	8	248°	12	0.8	E-F	
0.9 - 1.0	8	248°	12	0.8	F-G	
0.9 - 1.0	9	248°	15	1.0	E-F	

• possible wind turbine location

4.4 Chapter four - key concepts and conclusions

Although results will be further discussed in chapter six, it is sensible to bring in some ideas at this point regarding the reliability of the wind tunnel data obtained. This aspect must be addressed and should include considerations of both the accuracy of the overall simulation and the accuracy and repeatability of the measurements[24].

Unless these questions are carefully considered there will be an inherent risk that possible insufficiencies and inaccuracies will not be noticed and subsequent important decisions may be made based on faulty or insufficient data.

In this research careful attention was placed to achieve geometric similarity and observe blockage considerations that avoid distortion effects of the flow. At the same time it was necessary to produce a model of an appropriate scale that did not make it too difficult to model details and to study the wind velocity around them. Additionally, and in agreement with Farrell and Iyengar findings[35], the selection of the model scale was intended to represent the surrounding buildings adjacent to the studied building complex. It was possible to achieve the above requirements for this study as the model scale 1:500 effectively reproduced the studied building complex and three blocks in all directions were modelled (above the minimum stated by Lawson[23]) without exceeding the 2.5% of the overall wind tunnel cross sectional area.

Even though simulation experiments can be best carried out in long test section wind tunnels[36], there is evidence that the SBE wind tunnel vertical distributions of the mean wind velocity can be reasonably well simulated[19].

In these experiments, the locations that showed less potential to integrate small wind turbines were between buildings A-B and B-C in any of the tested cases. The locations that showed greatest potential to incorporate small wind turbines were between buildings D-E in case three, (θ= 260˚ and B= 15 m) at h_3, h_4 and h_5.

The key points addressed in this chapter are summarised in Box 4.2. It can be concluded that the important parameters which have to be modelled in the wind tunnel for the accurate study of wind flows around buildings and that the architect and building designer should be aware of including: a mean velocity profile roughly similar to the atmospheric boundary layer, turbulence and the correct sizing of the scale model.

BOX 4.2 chapter four key concepts

On the use of wind tunnels

- Boundary layer wind tunnels and building scale models are valuable tools to provide quantitative analysis of airflows around buildings
- The important parameters which have to be modelled in the wind tunnel for the accurate study of wind flows around buildings include: a mean velocity profile roughly similar to the atmospheric boundary layer, turbulence and the correct sizing of the scale model.

On the experiments

- The wind tunnel experiments conducted and described in this thesis were carried out in the boundary layer wind tunnel at the School of the Built Environment (SBE), at the University of Nottingham. The tests were done for nine different building configurations to investigate the influence of building orientation and street width on wind velocity at different heights between buildings using a CTA hot-wire.
- The incident flow reasonably simulated a natural wind over an urban terrain with a stable wind profile characterized by the exponent α= 0.23 according to the power law of velocity profile.

On the results

- Flow in the street canyons between the seven buildings in all cases in this study can be regarded as skimming flow.
- The locations that showed less potential to integrate small vertical wind turbines were between buildings A-B and B-C in any of the measured cases.
- The highest velocity ratio was measured between buildings D-E for orientation θ=260° and B= 15 m at h_5 (z/H = 1).

4.5 Chapter four – reference list

1. Lawson, T.V. (2001) *Building Aerodynamics*, London : Imperial College Press.

2. Davenport, A.G. (2002) Past, present and future of wind engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **90** (12-15):1371-1380.

3. Cook, N.J. (1985) *The designer's guide to wind loading of building structures*, Garston, Watford : Building Research Establishment, Dept. of the Environment ; London ; Boston : Butterworths.

4. Sexton, D.E. (1968) A simple wind tunnel for studying air-flow around buildings. In: Building Research Station (now Building Research Establishment) Current Paper CP69/68, (Ed.) *Reprinted from: Architect and Building News. June 1968. pp. 983-985*, Garston, U.K.:

5. Wise, A., Sexton, D. and Richards, D. (1971) Effects Due to Groups of Buildings [and Discussion]. *Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences (1934-1990)* **269** (1199):469-485.

6. Lawson, T.V. (1971) Landscape Effects with Particular Reference to Urban Situations. *Philosophical Transactions of the Royal Society of London. Series A, Mathematical and Physical Sciences (1934-1990).* **269** (1199):493-501.

7. Chang, C.-H. and Meroney, R.N. (2003) The effect of surroundings with different separation distances on surface pressures on low-rise buildings. *Journal of Wind Engineering and Industrial Aerodynamics* **91** (8):1039-1050.

8. Endo, M., Bienkiewicz, B. and Ham, H.J. (2006) Wind-tunnel investigation of point pressure on TTU test building. *Journal of Wind Engineering and Industrial Aerodynamics* **94** (7):553-578.

9. Sharma, R.N. and Richards, P.J. (2005) Net pressures on the roof of a low-rise building with wall openings. *Journal of Wind Engineering and Industrial Aerodynamics* **93** (4):267-291.

10. Ho, T.C.E., Surry, D., Morrish, D. and Kopp, G.A. (2005) The UWO contribution to the NIST aerodynamic database for wind loads on low buildings: Part 1. Archiving format and basic aerodynamic data. *Journal of Wind Engineering and Industrial Aerodynamics* **93** (1):1-30.

11. Pierre, L.M.St., Kopp, G.A., Surry, D. and Ho, T.C.E. (2005) The UWO contribution to the NIST aerodynamic database for wind loads on low buildings: Part 2. Comparison of data with wind load provisions. *Journal of Wind Engineering and Industrial Aerodynamics* **93** (1):31-59.

12. Lin, N., Letchford, C., Tamura, Y., Liang, B. and Nakamura, O. (2005) Characteristics of wind forces acting on tall buildings. *Journal of Wind Engineering and Industrial Aerodynamics* **93** (3):217-242.

13. Voisin, D., Grillaud, G., Solliec, C., Beley-Sayettat, A., Berlaud, J.L. and Miton, A. (2004) Wind tunnel test method to study out-of-service tower crane behaviour in storm winds: A selection of papers derived from those presented at the 11th International Conference on Wind Engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **92** (7-8):687-697.

14. To, A.P. and Lam, K.M. (1995) Evaluation of pedestrian-level wind environment around a row of tall buildings using a quartile-level wind speed descripter. *Journal of Wind Engineering and Industrial Aerodynamics* **54-55** 527-541.

15. Ahmad, K., Khare, M. and Chaudhry, K.K. (2005) Wind tunnel simulation studies on dispersion at urban street canyons and intersections--a review. *Journal of Wind Engineering and Industrial Aerodynamics* **93** (9):697-717.

16. Chang, C.-H. and Meroney, R.N. (2003) Concentration and flow distributions in urban street canyons: wind tunnel and computational data. *Journal of Wind Engineering and Industrial Aerodynamics* **91** (9):1141-1154.

17. Johnson, G.T. and Hunter, L.J. (1998) Urban wind flows: wind tunnel and numerical simulations--a preliminary comparison. *Environmental Modelling and Software* **13** (3-4):279-286.

18. Neff, D.E. and Meroney, R.N. (1998) Wind-tunnel modeling of hill and vegetation influence on wind power availability. *Journal of Wind Engineering and Industrial Aerodynamics* **74-76** 335-343.

19. Green, N.E., Etheridge, D.W. and Riffat, S.B. (2001) Location of air intakes to avoid contamination of indoor air: a wind tunnel investigation. *Building and Environment* **36** (1):1-14.

20. Yang, T., Wright, N.G. and Etheridge, D.W. (2006) A comparison of CFD and full-scale measurements for analysis of natural ventilation. *International Journal of Ventilation* **4**, 337-348.

21. van Moeseke, G., Gratia, E., Reiter, S. and De Herde, A. (2005) Wind pressure distribution influence on natural ventilation for different incidences and environment densities. *Energy and Buildings* **37** (8):878-889.

22. Aynsley, R.M., Melbourne, W.H. and Vickery, B.J. (1977) *Architectural aerodynamics*, London : Applied Science Publishers.

23. Lawson, T.V. (1980) *Wind effects on buildings*, London : Applied Science Publishers.

24. Anonymous (1999) Isyumov, N., (Ed.) ASCE manuals and reports on engineering practice no. 67; American Society of Civil Engineers. 214 p.

25. Etheridge, D.W. (2002) Nondimensional methods for natural ventilation design. *Building and Environment* **37** (11):1057-1072.

26. Mertens, S. (2006) *Wind energy in the built environment*. Multi-Science. Essex, UK. 169 p.

27. Carey, P. (2005) Direct wind tunnel modelling of natural ventilation for design purposes. *PhD. Dissertation. University of Nottingham, UK.* 221pp

28. Chiu, Y.H. and Etheridge, D.W. (2004) Experimental technique to determine unsteady flow in natural ventilation stacks at model scale. *Journal of Wind Engineering and Industrial Aerodynamics* **92** (3-4):291-313.

29. Dantec Dynamics (2003) Probes for Hot-wire Anemometry. Publication no. 196-105-01. 25 pp. Available on line at:<www.dantecdynamics.com> Accessed: 04/02/06.

30. JØrgensen F. E. (2002) How to measure turbulence with hot-wire anemometers. A practical guide. *Dantec Dynamics.* Publication no. 9040U6151. 73 pp. Available on line at: <www.dantecdynamics.com> Accessed: 03/02/07

31. Nakamura, Y. and Oke, T.R. (1988) Wind, temperature and stability conditions in an east-west oriented urban canyon. *Atmospheric Environment (1967)* **22** (12):2691-2700.

32. Grosso, M. (1992) Wind pressure distribution around buildings: a parametrical model. *Energy and Buildings* **18** (2):101-131.

33. Cermak, J.E., Isyumov, N., American Society of Civil Engineers and Task Committee on Wind Tunnel Testing of Buildings and Structures (1999) *Wind tunnel studies of buildings and structures*, Reston, VA : American Society of Civil Engineers.

34. Davenport, A.G., Isyumov, N., King, J.P.C. and International Association for Wind Engineering (1988) *International Conference on Wind Engineering: Progress in wind engineering : Proceedings of the 8th International Conference on Wind Engineering, held under the auspices of the International Association for Wind Engineering, London, Ont., Canada, July 8-12, 1991*, Amsterdam, Netherlands : Elsevier.

35. Farell, C. and Iyengar, A.K.S. (1999) Experiments on the wind tunnel simulation of atmospheric boundary layers. *Journal of Wind Engineering and Industrial Aerodynamics* **79** (1-2):11-35.

36. Tieleman, H.W. (2003) Wind tunnel simulation of wind loading on low-rise structures: a review: ENGINEERING SYMPOSIUM To Honour ALAN G. DAVENPORT for his 40 Years of Contributions. *Journal of Wind Engineering and Industrial Aerodynamics* **91** (12-15):1627-1649.

5 CFD modelling

5.1 Introduction

In this chapter the Computational Fluid Dynamics (CFD) simulations conducted for nine different building configurations to investigate the influence of building orientation and passage width on wind velocity at different heights using commercial code FLUENT 6.1.18 are reported.

5.1.1 Chapter overview

This chapter is divided into four main sections (Box 5.1). The first section describes how the computational fluid dynamics simulations are commonly used. It also provides a general overview of the relevant theories and methodologies related to the use of the commercial CFD code FLUENT 6.1.18 to simulate airflow around buildings.

BOX 5.1 chapter five overview

5.1 The background	A description of how the CFD simulations are commonly used along with an overview of the theory and requirements for a sucessful CFD simulation using commecial code FLUENT 6.1.18.
5.2 The CFD simulations	An explanation of how the domain, geometry and mesh were created and how the boundary conditions and turbulence model were specified for the CFD simulations in this study.
5.3 The results	The simulation results of the effects of building orientation, passage width and height on wind velocity are presented and possible locations for small wind turbines are suggested.
5.4 The key points	Summarised in Box 5.5

The second section (5.2) is the description of the procedure used in this study to carry out the CFD simulations. A detailed explanation on how the domain, geometry, grid and boundary conditions were specified in GAMBIT and in FLUENT for the simulations is provided.

The results of the CFD simulations are presented in section 5.3. These results will be discussed in chapter six against those found with the wind tunnel experiments described in chapter four. Additionally, some conclusions and key points are presented in section 5.4.

5.1.2 Overview of Computational Fluid Dynamics (CFD)

Computer simulations are credited with speeding up the design process, increasing its efficiency, and enabling the comparison of a broader range of design variants. Moreover, simulation tools can provide with a better understanding of the consequences of design decisions, which increases the effectiveness of the design process as a whole[1].

In recent years, advances in computer-aided design and simulation tools have been accompanied by a proliferation of applications in the built environment, especially in architectural design, structural analysis and –to some extent– operation of buildings. Since the mid-1970's a number of sophisticated dynamic thermal energy software have been developed, which are now being taken up by the building industry[2]. Those for CFD, however, have progressed relatively more slowly. The reason for this is that the mathematical representation of the physics of fluid flows and their numerical solution procedures are very complex.

Nevertheless, as explained in chapter three, the application of CFD simulations in the built environment is an alternative method to field measurements and wind tunnel simulations, and it is rapidly gaining acceptance.

CFD simulations for studying wind flows around buildings have many advantages; some of the major errors in the measurements of the flow properties calculated on scaled versions of the full-scale building were introduced in chapter four, it was concluded that measured flow properties at the scaled model should represent the flow properties at full-scale. In wind tunnel experiments, this requires that some similarity rules between scaled model and full-scale building be fulfilled. CFD calculations do not have this scaling problem because the flow can be simulated at full-scale. They present the alternative in the case of inevitable scaling problems in measurements with wind tunnels.

Moreover, CFD has been profusely used in recent years for different types of studies regarding flows in and around buildings:

- To study natural ventilation effects[3],[4],[5],[6],[7],[8],[9]
- To investigate airflow around buildings[10],[11],[12],[13],[14] including pedestrian comfort[15]
- To analyse the thermal performance of glazed double facades[16]
- To study mean pressure coefficients[17]
- To describe the three-dimensional characteristics of flow regimes within urban canyons [18],[19]
- To model urban air quality[20],[21]
- To study wind effects on tall buildings[22]

These and other numerous similar studies have led to an apparent general acceptance that, provided that the domain, grid, boundary conditions and turbulence model are properly set up, it is possible to obtain realistic predictions with CFD simulations and can be utilized with confidence to replicate full scale flow conditions in and around buildings, such as air movement (natural ventilation), temperature and pressure distributions in and around buildings and contaminant concentration.

5.1.3 CFD modelling requirements

Because the environmental and building designers' interests are focused only on the results of CFD simulations, rather than the processes involved in obtaining these results, only a brief explanation of the numerical methods behind the CFD simulations for studying building aerodynamics is presented in this section, along with some requirements that have to be taken into consideration to minimise the possibility of error in the results.

CFD modelling is not a straightforward process, and issues of quality control in the CFD simulations have been raised by many authors[3],[23],[24],[25],[26],[27] in fact, there is evidence that even with a specific training in fluid dynamics, obtaining accurate results with CFD simulations is not a simple task.

In a very interesting study, Chen and Zhai[3] asked a group of graduate students majored in mechanical engineering that possessed sufficient knowledge of fluid dynamics, heat transfer and numerical methods, to run a simulation of a given problem with a commercial CFD code. They reported that none of the students could obtain correct results in their first attempt to solve the problem[I]. The errors during the simulation set up of the given problem included:

- Difficulty in selecting a suitable turbulence model
- Incorrect setting of boundary conditions
- Inappropriate selection of grid resolution
- Failure to estimate correctly convective portion of the heat from the heat sources
- Improper use of numeric techniques, such as relaxation factors and internal iteration numbers

Since the above mentioned are detected as the most common causes of error in CFD simulations, the next sections will provide a brief explanation of those topics that are relevant for this study (i.e. turbulence model, boundary conditions, grid resolution, and the use of numeric techniques)[II].

The atmospheric surface layer flow varies considerably depending on the thermal stability (i.e. stable, unstable or neutral) and the nature of the terrain and since turbulence normally dominates; an unsteady treatment is required in principle. Nevertheless, a time average leading to a statistically steady description of the turbulent flow is commonly used within urban environments and it is in general modelled with CFD by solving the fundamental governing equations of fluid

[I] Chen and Zhai[3] had experimental data for the given problem that allowed them to verify the results of the CFD simulations carried out by the students.
[II] In this study no heat sources are used, therefore no explanations of the heat transfer processes are provided.

dynamics that describe the exact behaviour of a Newtonian fluid, including the effects of turbulence. These five equations are[III]:

- The equation of motion in each of the three components of direction (the Navier-Stokes equations)
- The continuity equation and
- The energy equation.

These governing equations are complex mathematical procedures, but they become more amenable when they are averaged in time[28]. This means that the governing equations are rewritten using a decomposition into an averaged component and a fluctuating component[IV]. When this decomposition procedure is used for the velocity, pressure and temperature variables, it is called the Reynolds decomposition.

The decomposed flow equations are time-averaged or ensemble averaged[V], and when they are used for the Navier Stokes equations, they receive the name of Reynolds Averaged Navier Stokes equations (RANS)[29]. Most of the turbulence terms disappear in the RANS equations, but new terms; the Reynolds stresses, appear. Modern numerical techniques, (with appropriate turbulence models), allow solutions to be computed when assumptions are made about these Reynolds stresses in order to close the RANS equations[VI].

[III] A comprehensive explanation on the governing equations can be found in Anderson[30]

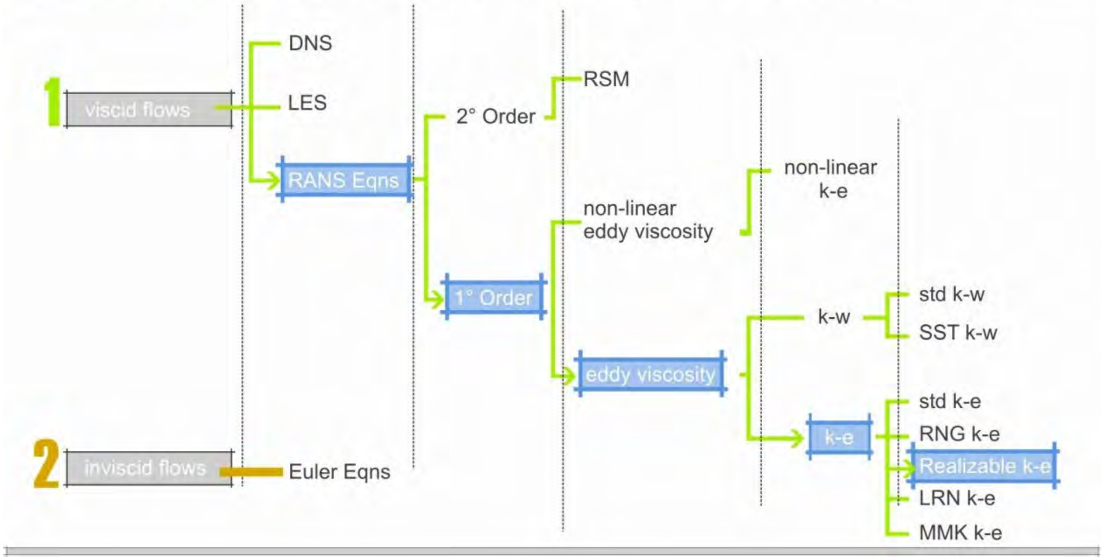
[IV] For instance, the instantaneous velocity component u_i is decomposed by: $u_i = \overline{u}_i + u'_i$, being $\overline{u}_i$ the averaged component.

[V] The most mathematically general average is the ensemble average, in which a given experiment is repeated a large number of times and average the quantity of interest (i.e. velocity) at the same position and time in each experiment[31].

[VI] It is possible to directly solve (without the use of turbulence models) the RANS equations for laminar and turbulent flows when all of the relevant length scales can be contained on the grid; this is called a

Different approximations of the flow equations or turbulence models can increase or decrease the computational effort to solve the equations. The constraints lie in the choice of the turbulence model and the grid configuration; nevertheless, it is important to realize that even if the governing equations of the given problem are solved accurately, the result may be wrong if the governing equations – and boundary conditions – do not model the intended physics, like using the wrong turbulence model, neglecting buoyancy effects, or over simplifying the studied geometry[26].

BOX 5.2 Turbulence models commonly used in CFD codes



Different turbulence models are able to cope with different flow elements. It is important then, to briefly discuss some of these turbulence models in order to show their advantages and constraints[VII]. Box 5.2 summarises the most commonly

---

direct numerical simulation (DNS). In general, however, the range of length scales appropriate to the problem is larger than even today's massively parallel computers can model, limiting DNS to very simple geometries.

[VII] See Bhaskaran and Collins[31] for a comprehensive explanation on the use of turbulence models in commercial CFD codes.

turbulence models used in commercial CFD codes, highlighting the choice of turbulence model used for this study.

Turbulence models

The standard k- ε model

The k-**ε** turbulence model is the most used and most popular of all the RANS models. It needs a small computational effort, but the model has limited physical background. It is based on a two-equation model; one transport equation for turbulent kinetic energy (k) and one transport equation for the dissipation of turbulent kinetic energy (**ε**).

The limitations of this model to simulate airflows around buildings are summarised by Mertens[29] as follows:

- It has a limited applicability restricted to fully turbulent wall bounded and free shear flows.
- It is not sensitive to free stream turbulence
- It cannot be trusted in flows that involve strong streamline curvature, and
- It produces a spuriously large generation of k and a resulting over prediction of the eddy viscosity (μ_t) around a stagnation point.

Although standard k-**ε** turbulence model has been proved to give accurate results on roof-top mean pressure coefficients simulations[17] and some researchers are also in favour of k-**ε** turbulence model to simulate wind flow conditions around buildings[32], there is evidence that it has limitations when representing the atmospheric surface layer[29].

The realizable k- ε model

In order to reduce some shortcomings of the standard k-**ε** model, the CFD package FLUENT 6.1.18, which was used for the calculations in this research, provides more sophisticated models[VIII], as well as more elaborated options to the k-**ε** model: the realizable k-**ε** model and the RNG k-**ε** model. The last one is not discussed here, because initial studies[29] showed that the realizable model gives the best performance of all k-**ε** models, for simulating flows around buildings.

The realizable k-**ε** model has additional terms and functions in the transport equation for k and **ε**, which make the model applicable to a wider range of flows. Differently than the standard k-**ε** model, where the eddy viscosity (μ_t) is calculated with:

(**Eqn. 5.1**)

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon}$$

In this equation, Cμ is assumed with constant value (0.09). The improvement of the realizable k-**ε** model is that Cμ is no longer constant but sensitised to gradients in the main flow. Moreover, this model reduces the large generation of k around the stagnation point and it gives physical or realizable results at large strain rate. The

---

VIII Models that use more than two transport equations such as the seven-equation Reynolds Stress Model (RSM) and time-dependent Large Eddy Simulation (LES), are not presented here because their large need of computational resources has been documented[33],[29],[26], and thus exceed the limitations of this research.

additional information in the realizable k-**ε** model results in some important advantages compared to the standard k-**ε** model.

According to Mertens[34] the realizable k-**ε** model should give better results for:

- The spreading rate of planar and round jets
- Boundary layers under strong pressure gradients
- Separation
- Recirculation

These conditions provide with a rationale for utilizing the realizable k-**ε** model for the CFD calculations carried out in this study.

5.1.4 FLUENT CFD code

For this study, the commercial CFD code FLUENT 6.1.18 was used to simulate full-scale airflow around buildings. It has been explained that CFD codes work by solving the governing equations with the use of a turbulence model. Different CFD commercial codes have different discretisation methods to solve those governing equations[IX].

FLUENT code uses a finite-volume discretisation method[X] to solve the governing equations; that means that the region of interest (the domain) is divided into a finite number of cells or control volumes (the mesh or grid). In the simulation, the

[IX] Some discretisation methods used by other commercial codes include: finite element, finite difference and boundary element method.
[X] The finite-volume method is regarded to be better suited for modelling flow past complex geometries[33].

variables are solved at the centre of the cell. The values at other locations are determined by interpolating those values. In other words, the accuracy of the numerical solution will usually improve with an increased number of grid points, especially if the increase is made in spatial regions with complex geometries[26]. For this reason the creation of the mesh (or grid) is one of the most important issues to consider for a successful CFD simulation.

FLUENT software includes three main components[33]:

- A pre-processor: where the domain, geometry and mesh are created; and fluid flow properties and the type of boundaries are specified.
- A solver: where boundary conditions created – or modified – the turbulence model is selected and converge criteria is applied.
- A post-processor: where the modelled results are analysed both numerically and graphically.

The computational domain

For 3D problems or 2D complex geometries, it is common that the domain, the grid and the geometry of interest (buildings) are created in FLUENT bundled pre-processor software, GAMBIT. GAMBIT uses a user friendly graphic interface that allows easy access to the commands via either keyboard or icons.

As reported by Hu and Wang[11], there are no explicit rules dictating the size of the domain; nevertheless, some authors[10] suggest that the size of domain can be a multiple of the characteristic height of the building[XI] and many others test the sensitivity of the flow field to the domain size and thus determine their domain size by trial-and-error[32], [11]. Hall[35] suggests that the distance between any edge of the domain and the buildings must be at least five times the characteristic height of the building.

The extent of the built area (e.g. surrounding buildings) that is represented in the computational domain depends on the influence of those features on the region of interest. An experience from wind tunnel simulations is that a building with height H_x may have a minimal influence if its distance from the region of interest is greater than 6-10 H_x. Thus, as a minimum requirement, a building of height H_x should be represented if its distance from the studied building complex is less than $6H_x$. In case of uncertainty regarding the influence of distant buildings on the flow in the area of interest, COST[23] best practice guidelines recommend to perform simulations with and without the surrounding buildings.

Specifying the right size for the domain and the extent of the area to be modelled in the CFD model, has similar implications in the simulation results as those explained in section 4.1.3 regarding scaling ratios and blockage corrections for the wind tunnel simulations. Ultimately, the size of the entire computational domain in the vertical, lateral and flow directions depends on the area that is to be represented and on the boundary conditions that will be used.

[XI] It was only with the recent publication of the COST[23] best practice guidelines for the CFD simulation of flows in urban environments that more educated recommendations of domain size can be made.

The computational grid

Once the domain and the geometry of interest are generated in the pre-processor, the next step to be followed is the division of the domain, known as space discretisation; which divides the continuous space into a finite number of grid points: a computational grid. FLUENT offers two different types of grids to be used: structured grids and unstructured grids.

In the structured grids, the number of cells that share a common vertex is uniform within the domain; this means that the geometric domain is decomposed into sub-domain blocks, within which a structured grid is generated[XII]. The generation of the domain decomposition into blocks requires much time and effort as it has to be defined by the user.

Unstructured grids have triangular (tri) or quadrilateral (quad) mesh elements for 2D problems, and tetrahedral, pyramidal, hexahedral or wedge computational cells for 3D problems[XIII]. Unstructured grids are more flexible and easier to generate than structured grids and better for refinement near the zone of interest. The disadvantage of an unstructured grid is the irregularity of the data structure, which means that the development of accurate discretisation and efficient solution methods are more difficult to achieve than with a structured grid.

The standard procedure to create the grid is to first mesh the edges of the geometry, then the faces and lastly, the volumes.

[XII] For 2D and 3D problems in GAMBIT the commands Map and Submap are used to create a regular, structured grid of mesh elements[36].

[XIII] GAMBIT[36] uses the commands Pave, Tri Primitive and Wedge Primitive to create an unstructured grid of mesh elements for 2D problems and Tet Primitive, Cooper, TGrid, Stairstep and Hex Core for 3D problems.

As discussed before, the accuracy of the results is directly related to the number of grid points used; nevertheless, if the mesh continues to be refined, it will reach a point where the solution will not vary much. This is known as grid convergence or grid-independency. This can be achieved by comparing two converged solutions for two different grids: a fine and a course grid.

Zone type specifications

In GAMBIT, zone-type specifications define the physical and operational characteristics of the model at its boundaries and within specific regions of its domain. There are two classes of zone-type specifications: boundary types and continuum types.

Boundary type specifications define the physical and operational characteristics of the model at those topological entities that represent model boundaries. Typically, they consist of the selection of a surface (in 3D problems) specifying it as inlet, outlet, wall and/or symmetry boundary conditions.

Continuum type specifications define the characteristics of the model within specified regions of its domain. These specifications include the definition of a volume entity as fluid or solid[XIV].

The definition of boundary conditions greatly affects the results of the simulations. Different authors have different approaches to specify the boundary conditions in the

[XIV] Note that because each computational solver is associated with a unique set of available boundary types, in order to define them in GAMBIT a solver has to be selected beforehand, preferably before the creation of the domain and geometry. For this study the solver used is FLUENT 5/6.

computational domain. Table 5.1 summarises the specified boundary conditions for relevant studies where wind flows were simulated around buildings[XV].

Table 5.1 Boundary conditions used for studies of wind flows around buildings.

AUTHORS	Inlet	Outlet	Top	Bottom	Sides	CFD Code
1 Blocken et al[39].	Incident power law velocity (Eqn. **4.1**), turbulence and ε profiles from experimental data	Zero static pressure	Non-slip wall	Wall with standard wall functions	Non-slip walls	Fluent 6.1.22
2 Hargreaves and Wright[37]	Velocity, k and ε profiles specified from experimental data (Eqn **5.2**, **5.3**, **5.4** and **5.5**)	Pressure outlet	Constant sheer stress	Symmetry	Symmetry	Fluent 6.1 and CFX
3 Hu and Wang[11]	Incident power law velocity (Eqn. **4.1**), turbulence and ε profiles from experimental data	Gauge pressure=0	Non-slip wall	Wall with standard wall functions	Symmetry	PHOENICS
4 Mertens[29]	Velocity, k and ε profiles specified with Eqn **5.2**, **5.3**, **5.4** and **5.5**	Velocity, k and ε profiles specified with Eqn **5.2**, **5.3**, **5.4** and **5.5**	Velocity, k and ε profiles specified with Eqn **5.2**, **5.3**, **5.4** and **5.5**	Wall with standard wall functions	2D	Fluent 6.1.18
5 Meroney et al[17].	Incident power law velocity (Eqn. **4.1**), turbulence and e profiles from experimental data	Outflow	Symmetry	Wall	Symmetry	Fluent 4.4.8
6 Richards and Hoxey[38]	Velocity, k and ε profiles specified from experimental data (Eqn **5.2**, **5.3**, **5.4** and **5.5**)	Not specified	Constant sheer stress	Retarding sheer stress, locally calculated with Eqn. **5.6** and **5.7**	2D	PHOENICS
7 Xiaomin et al[19].	Inlet conditions	Free boundary conditions V=0	Free boundary conditions	Wall	Symmetry	PHOENICS

From Table 5.1 it can be appreciated that some authors specify the inlet velocity with the logarithmic profile equation[37],[29],[38]:

[XV] Note that method to specify velocity, turbulence and dissipation profiles varies according to the CFD code being used. In FLUENT, a User Defined Function is needed, as will be explained in the following sections.

(**Eqn. 5.2**) $$U = \frac{u_*}{K} \ln\left(\frac{z + z_0}{z_0}\right)$$

Where K is the von Karman constant equal to 0.4, z_0 is the aerodynamic roughness length and the friction velocity (u^*) is usually calculated from a specified velocity U_h at a reference height h, with equation **5.3**:

(**Eqn. 5.3**) $$u^* = \frac{KU_h}{\ln\left(\frac{h + z_0}{z_0}\right)}$$

While some authors specify the inlet velocity profile with the power law equation[39],[11],[17] (**Eqn. 4.1**), and other authors specify a uniform velocity at the inlet[19].

The turbulent kinetic energy and turbulence dissipation profiles are calculated with equations **5.4** and **5.5** respectively.

(**Eqn. 5.4**) $$k = \frac{u_*^2}{\sqrt{C\mu}}$$

(**Eqn. 5.5**) $$\varepsilon = \frac{u_*^3}{K(z + z_0)}$$

The shear stress at a height g the bottom of the boundary[XVI] is calculated locally with equation 5.6 and the friction velocity at a height g is calculated with equation 5.7.

(Eqn. 5.6)
$$\tau_g = \rho u_{*g}$$

(Eqn. 5.7)
$$u_{*g} = \frac{KU_g}{\ln\left(\frac{z_g + z_0}{z_0}\right)}$$

Once the domain, geometry, mesh and zone-types are created in GAMBIT, a mesh file (*.msh) can be created and exported to FLUENT to initiate calculations and solve the given problem.

Solver

Once the mesh file is read in FLUENT as a case, the first step is to check the grid for errors and verify that the boundary types created in GAMBIT are correct. It is possible to modify an existing boundary-type specification created in GAMBIT. The boundary conditions represent the influence of the surroundings that have been cut off by the computational domain and they specify the flow and thermal variables on the boundaries of the physical model. They determine to a large extent the solution inside the computational domain and are; therefore, a critical component of CFD simulations and it is important that they are specified appropriately.

FLUENT available boundary types are classified as follows[33]:

[XVI] Only Richards and Hoxey[38] use this approach for specifying the bottom boundary.

- Flow inlet and exit boundaries: pressure inlet, velocity inlet, mass flow inlet, intake fan, inlet vent, pressure outlet, pressure far-field, outflow, outlet vent and exhaust fan.
- Wall, repeating, and pole boundaries: wall, symmetry, periodic and axis.
- Internal cell zones: fluid and solid.
- Internal face boundaries: fan, radiator, porous jump, wall and interior.

Once the boundary conditions are revised and specified, the next step is to select the turbulence model to be used to run the simulations. As explained before, the selection of the turbulence model depends on the nature of the flow to be modelled.

Because the governing equations are non-linear, it is necessary to iterate through successive approximations until a converged solution is obtained. This means that the residuals[XVII] fall below a determined chosen value which is referred to as the convergence criterion. The scaled residuals option available is a more appropriate indicator of convergence for most problems[33]. Although there are no universally accepted criteria for judging the final convergence of a simulation[27], the default convergence criterion in FLUENT requires that the scaled residuals decrease to 10^{-3} for all the equations. Nevertheless, the convergence criterion chosen for each conservation equation is problem-dependent.

At this stage, the user has defined the convergence criterion and the number of iterations to run, hence, the solution will stop automatically when each variable

[XVII] Residuals are 3D fields associated with a conservation law, such as conservation of mass or momentum. They indicate how far the present approximate solution is away from perfect conservation (balance of fluxes).

meets its specified convergence criterion. In the post-processing stage, the results are analysed, either graphically (i.e. using contour or vector plots) or numerically (i.e. exporting data files to further analyse with the use of a spreadsheet).

5.2 CFD tests configuration

For this study several CFD tests using commercial code FLUENT 6.1.18 were carried out before determining an optimal set up configuration that would best model real conditions. Those tests include changing the domain size, modelling the studied building complex with and without surroundings, working with different grid types and sizes, change of turbulence models, change of boundary conditions (especially the velocity inlet), and changes in the convergence criterion.

In the following sections only the set up for the CFD simulations that were used to compare with the wind tunnel results are described.

The nine cases were modelled using real scale magnitudes to allow direct interpretation of the results. Box 5.3 summarises the standard procedure[XVIII] used to carry out the CFD simulations in this study.

5.2.1 Pre-processor

The physical domain and the three-dimensional geometry of the buildings for the nine cases in this study were created and meshed using FLUENT pre-processor

[XVIII] Adapted from [40] and [41]. Further details can be consulted in [33].

GAMBIT 2.2. Boundary types and fluid flow properties were also defined in GAMBIT, as explained in the following sections.

BOX 5.3 CFD simulations procedure

1 The domain and the geometry	The domain of dimensions 1150 X 704 X 400 m and the tower buildings and context geometries were created in GAMBIT.
2 The grid	The geometry was meshed in GAMBIT using an unstructured tetrahedral grid.
3 The boundary types	Boundary and continuum type specifications were made in GAMBIT.
4 Set up problem	The selection of the turbulence model and the specification of the convergence criterion were made in FLUENT.
5 Solve the problem	Using FLUENT segregated solver, the solution was initialised and the model was left to iterate.
6 Analise results	Exporting numerical data and using vector and contour plots.

The creation of the domain and the geometry

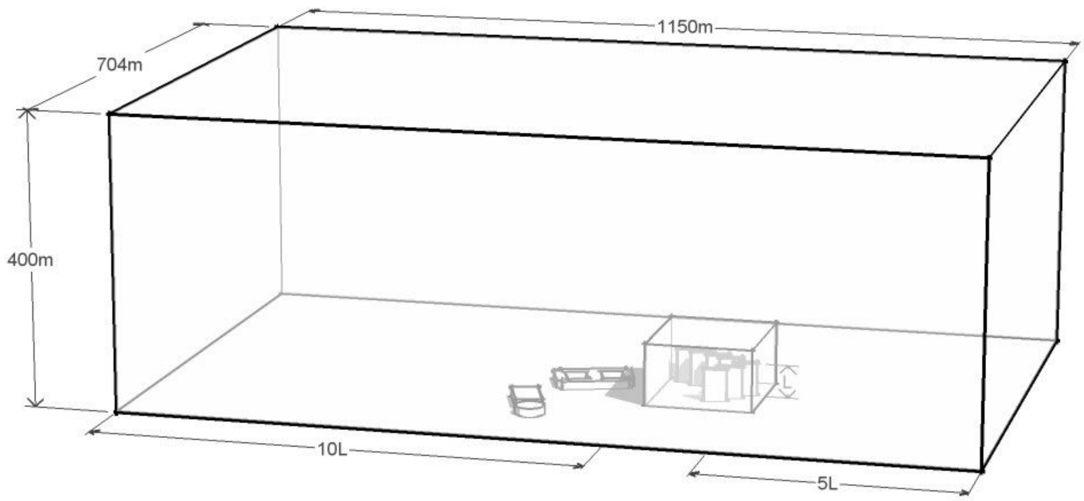


Fig. 5.1 Computational domain and geometry.

The domain size was 1150 m X 704 m X 400 m in all simulations. Figure 5.1 illustrates the studied building complex and surrounding buildings within the computational domain. The domain was sub-divided into two zones; a smaller sub-domain was created around the tower buildings to allow for generating a finer mesh next to the buildings and a coarser mesh at the domain boundaries, thus reducing the computational resources needed to solve the simulation. The windward distance was 10 times the characteristic height of the tower buildings (L) and the leeward distance was 5L.

Setting the meshing scheme

An unstructured tetrahedral grid was used to mesh all the faces and volumes in the domain. The Tri-Pave meshing scheme was applied to all faces. This creates a face mesh consisting of irregular triangular mesh elements. This scheme was used because there are no restrictions on the edge mesh intervals for the Tri-Pave meshing scheme, allowing smaller cells near the interest zones, between the tower buildings.

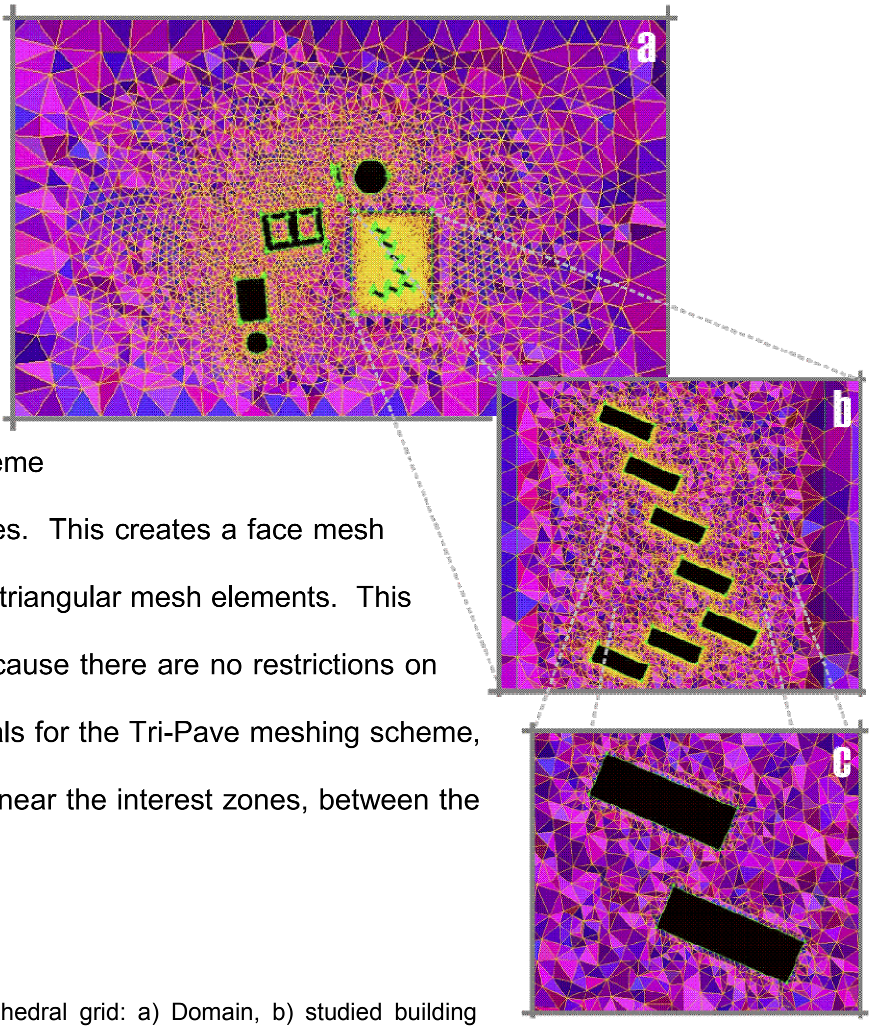


Fig. 5.2 Unstructured tetrahedral grid: a) Domain, b) studied building complex and c) between towers.

Volumes were meshed using the TGrid scheme, which creates a mesh that consists primarily of tetrahedral elements, but which may also contain hexahedral, pyramidal, and wedge mesh elements. Nevertheless, because of the fact that for this study all faces were previously meshed with Tri-Pave scheme, only tetrahedral elements were created. Figure 5.2 (a-c) shows the domain, the geometry and the mesh as used in case one.

The TGrid meshing scheme was used to mesh all volumes in the nine cases as it imposes no restrictions on the types of edge or face meshes that can be previously applied to the volume and it allows easier control of the refinement for the tetrahedral mesh by means of the GAMBIT program defaults[36].

Defining the boundary types

The boundary conditions for this study were defined to simulate a neutral atmospheric boundary layer. The effect of changes in wind direction with height was to be included in the model by properly selecting the incoming flow profile. The velocity profile at the domain inlet was specified with a User Defined Function (UDF). The UDF is a function written in the C programming language using any text editor that can be dynamically loaded with the FLUENT solver using predefined macros and functions supplied by Fluent Inc[42].

The boundaries types of the domain for this study are illustrated in Figure 5.3, and were specified as follows:

- Bottom plane: wall with standard wall functions.
- Top and sides planes: Symmetry.

- Outlet: Outflow.
- Internal sub-domain: Interior.
- Inlet: velocity profile specified using an UDF.

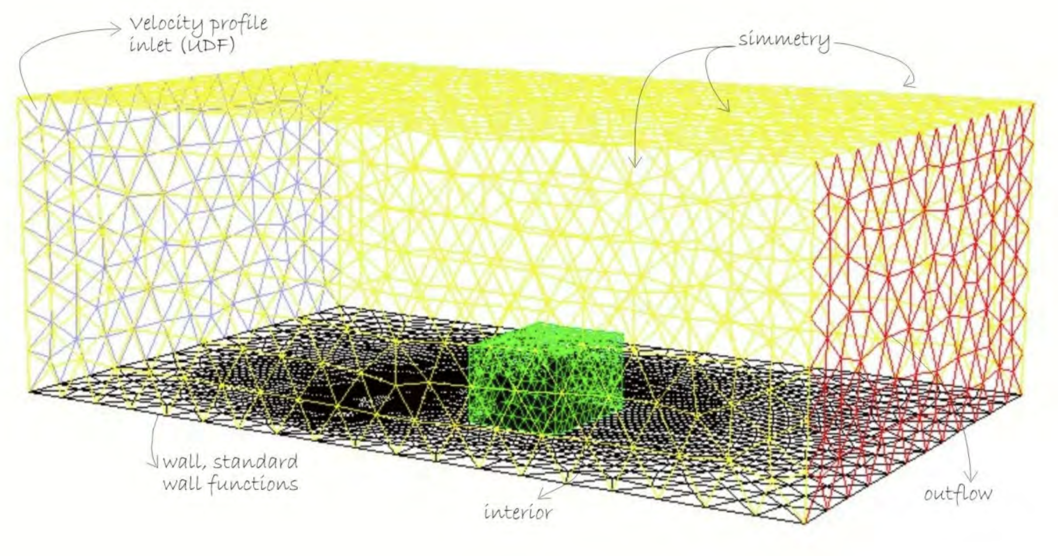


Fig. 5.3 Domain boundary types.

Bottom plane

Wall boundary conditions are used to bound fluid and solid regions. The standard wall functions provided in FLUENT were used at the bottom boundary because they work reasonably well for a broad range of wall-bounded flows and the wall boundary conditions for the solution variables, including mean velocity, temperature, species concentration, k, and **ε**, are all calculated with the standard wall functions.

Top and sides planes

FLUENT assumes a zero flux of all quantities across a symmetry boundary. Symmetry boundary conditions were chosen at the top and side faces of the domain because they enforce a parallel flow by requiring a vanishing normal velocity component at the boundary, having mentioned that it is recommended that the boundary face be positioned far enough away from the built area of interest in order not to lead to an artificial acceleration of the flow in the region of interest[33].

Outlet

Outflow boundary conditions in FLUENT are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. Outflow condition was selected for the outlet plane because outflow conditions are extrapolated from within the domain and have no impact on the upstream flow. The extrapolation procedure used by FLUENT updates the outflow velocity and pressure in a manner that is consistent with a fully-developed flow assumption[XIX]. Only the diffusion fluxes in the direction normal to the exit plane are assumed to be zero.

Internal sub-domain

In order to reduce the computational resources needed, an internal sub-domain was created enclosing the studied building complex. This allowed creating a finer mesh around the tower buildings, where more grid points were necessary. The boundary conditions of this sub-domain were regarded as interior. In FLUENT this creates an internal 'wall' that does not obstruct the wind flow and, hence, does not have any effect on wind velocity.

[XIX] Fully-developed flows are flows in which the flow velocity profile (and/or profiles of other properties such as temperature) is unchanging in the flow direction.

Inlet

To create the UDF C source code file, an equation that defined the user-defined function has to be specified. For the simulations described in this study this equation was the power law equation using the coefficients found in the experimental work carried out with the wind tunnel as described in chapter four (Eqn 4.1). For UDF writing purposes, equation 4.1 has to be re-arranged as shown in equation 5.8, being the complete UDF source code shown in Appendix 1:

(Eqn. 5.8) F_PROFILE(f, thread, index) = 5.4775 * pow ((z/50),.2275);

Source code files containing UDFs can be either interpreted or compiled in FLUENT. The major difference between interpreted and compiled UDFs is that interpreted UDFs cannot access FLUENT solver data using direct structure references; they can only indirectly access data through the use of Fluent-supplied macros[42], making the compiled UDFs more versatile. Interpreted UDFs are interpreted from the source code using a graphical user interface in a single-step process. The process, which occurs in this study, when defining the boundary conditions before running the simulation, involves a visit to the Interpreted UDFs panel to interpret the function in a source file.

Although interpreted UDFs have some advantages as not requiring a C compiler and the fact that they are portable to other platforms; their main disadvantages are that they are slower than compiled UDFs, they are restricted in the use of the C programming language and they can access data stored in a FLUENT structure only by using a predefined macro. For these reasons the use of interpreted UDFs is restricted for small, straightforward functions, which is the case for defining the inlet velocity profile boundary for the simulations carried out in this study.

5.2.2 Problem set up in FLUENT

This study represents the simplest case in which the density can be treated as a constant as is normally done in the context of flows in urban areas. Box 5.4 summarises how the CFD simulations carried out in this study were set up, including 3D space domain extents and grid size. Standard wall functions were used for all simulations. The problem was regarded as steady and the solver was used in segregated mode. First order upwind discretisation schemes were used for the momentum, turbulent kinetic energy and turbulence dissipation rate equations.

BOX 5.4 CFD problem set up

A Domain extents (m)		min	max
	x-coordinate	0.00E+00	1.15E+03
	y-coordinate	0.00E+00	7.04E+02
	z-coordinate	0.00E+00	4.00E+02
B Grid size	Tetrahedral cells:	379,517	
	Nodes:	80,764	
C Models	Model	Settings	
	Space	3D	
	Time	Steady	
	Solver	Segregated	
	Viscous	Realizable k-epsilon	
	Wall treatment	Standard wall functions	
D Discretization scheme	Variable	Scheme	
	Pressure	Standard	
	Pressure-Vel coupling	SIMPLE	
	Momentum	First order upwind	
	T. kinetic energy	First order upwind	
	T. dissipation rate	First order upwind	

Turbulence model

The choice of turbulence model used for the simulations in this study was based on the flow's physics to be modelled, the best practice guidelines, the available computational resources, and the amount of time available for each simulation. Air turbulence was represented by the Realizable k–**ε** turbulence model. The air flowing between the buildings was regarded as an incompressible flow.

Tests Convergence and solution accuracy

As stated before, there are no universal metrics for judging convergence. The chosen convergence criterion for all the simulations in this study was configured so that the scaled residuals decrease to 10^{-4} for all the equations. The calculations will stop when the number of iterations are finished or when the solution of the problem has converged.

5.3 CFD tests results

5.3.1 Velocity profile

Before simulating the flow over obstacles, it is good practice to carry out an analysis to ascertain whether the chosen grid and boundary conditions are consistent and there is no substantial change in the specified inflow boundary profiles[23].

Therefore, in order to verify that the simulations settings were able to satisfactorily reproduce the atmospheric surface layer, a test was done with the empty domain to assess the velocity profile at different points along the domain. Figure 5.4 presents a plot of the cell-centred values of the streamwise wind speed, *u*, profiles at the inlet, outlet and at the middle of the domain. The profiles are to a height of $50Z_{ref}$ (300m) and at distances of 10, 600 and 1000m along the domain. It can be appreciated that the velocity profile of the *u*-component of velocity was maintained throughout the entire length of the domain.

Similar plots of the velocity profiles were made for every simulated case to a height of $50Z_{ref}$ and at different distances along the fetch to analyse any variation from a developed surface layer. The plots were made at distances of X=10m for the inlet, X=1000m for the outlet, X=350m (upstream surrounding buildings) and X=600m at the studied building complex.

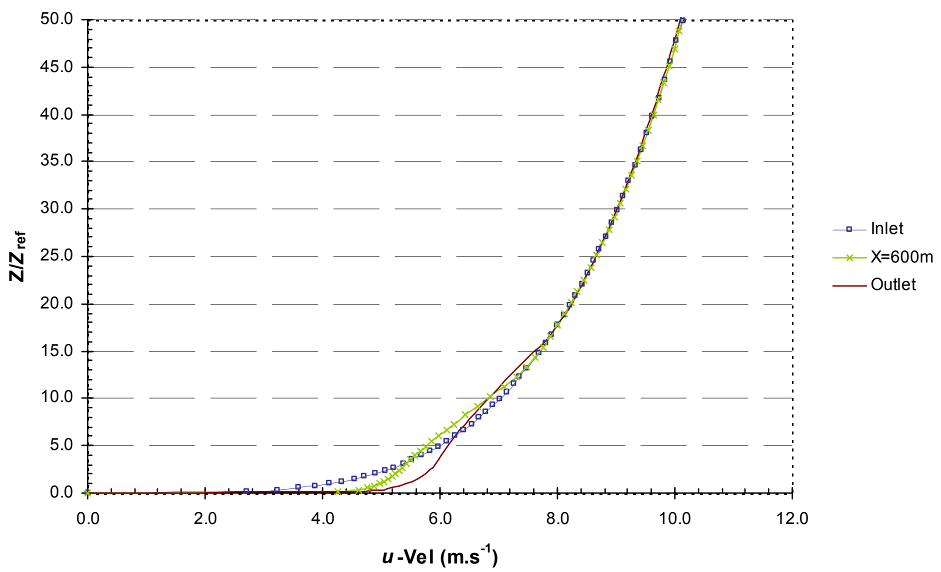


Fig. 5.4 Velocity profiles at different points along the empty domain.

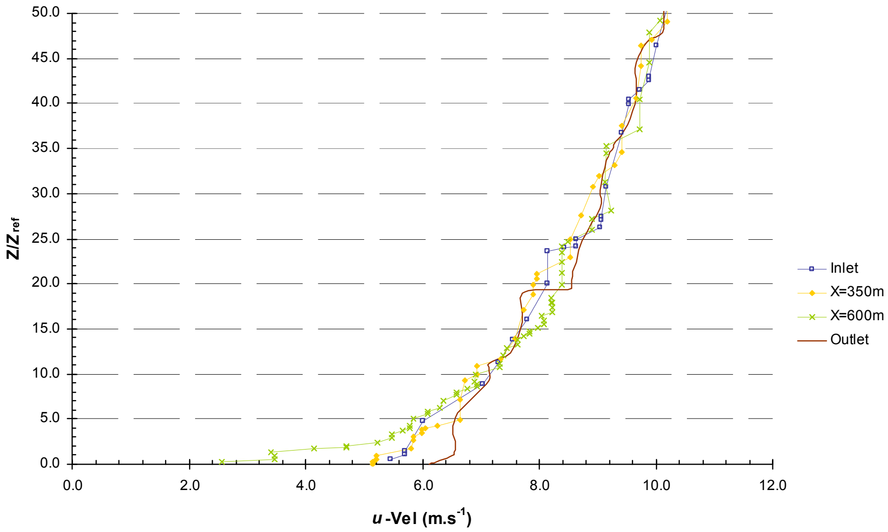


Fig. 5.5 Velocity profiles. Case one (θ= 260˚, B= 9 m).

In general terms, in the case of a homogeneous equilibrium boundary layer flow, the velocity profiles should not change until the built area is reached[23]. Figure 5.5 illustrates the plots at cell-centred values for case one (θ= 260˚, B= 9 m). It appears that the velocity profile of the *u*-component of velocity was maintained reasonably well along the domain, with an observable change when reaching the studied building complex at X=600 m. It should be noted that the velocity profile at the outlet was not completely smooth for this case and for case four (Appendix 2), both cases with B=9 m. It remains an open question whether this could be improved with a modification on the CFD domain, by increasing the leeward distance from the building complex to the outlet boundary, therefore allowing the flow to develop even further before reaching the outlet.

Nevertheless, as the plots of the velocity profiles in Appendix 2 show, the rest of the cases present a reasonably well maintained velocity profile throughout the domain including those at the outlet. Small differences in the *u*-component of velocity were detected in case three, especially at ground level. Possible reasons for those differences will be discussed in next chapter.

5.3.2 Velocity magnitude ratios

The ratio u/U_{ref} ranges from 0.1 to 1.1 in all cases, at the five different heights. The lower computed ratios were located at h_1 (z/H = 0.2) ranging from 0.1 to 0.8 and the higher ratios were found at h_5, varying from 0.1 to 1.1.

The cases that presented better possibilities for small wind turbines integration - i.e. the cases with higher computed velocity ratios - were those with orientation θ= 260˚ that is, cases one, two and three.

Overall, the higher ratios were located at the five different reference points between buildings D-E for orientation θ= 260˚ and B= 15 m (case three), at all heights, as follows: h_1=0.87, h_2=0.91, h_3=0.95, h_4=0.90 and h_5=1.01.

Table 5.2 summarises the points where the computed u/U_{ref} ratios are most favourable for small vertical wind turbine integration, indicated with a dot between buildings on the schematic plan.

The highest velocity ratio computed was found between buildings D-E for orientation θ= 260˚ and B= 15 m (case three) at h_5 (z/H = 1). Other locations of high velocity ratios were recorded between buildings E-F for the same case at h_5. At the same height (z/H = 1) high ratios were found between buildings E-F for orientation θ= 248˚

and B= 15 m (case 9) and between buildings F-G and for θ= 260˚ and B= 12 m (case 2).

Various points at h_4 (z/H = 0.8) also presented high computed u/U_{ref} ratios; between buildings D-E and E-F for orientation θ= 248˚ at B= 15 m and between buildings E-F and F-G for orientation θ= 260˚ at B=9 and between buildings F-G for the same orientation and B= 12 m.

At h_3 (z/H = 0.8) the reference point that registered the higher computed ratio was between buildings D-E for orientation θ= 260˚ at B=15 m (case three).

Table 5.2 Computed velocity ratios suitable for wind turbine integration.

Velocity Ratio (u/U_{ref})	Case Number	Orientation (θ)	Gap (B) (m)	Height (z/H)	Buildings	Schematic Plan
0.9 - 1.0	1	260°	9	1.0	E-F	
0.9 - 1.0	1	260°	9	0.8	E-F	
0.9 - 1.0	1	260°	9	0.8	F-G	
0.9 - 1.0	2	260°	12	1.0	E-F	
0.9 - 1.0	2	260°	12	1.0	F-G	
0.9 - 1.0	2	260°	12	0.8	E-F	
0.9 - 1.0	2	260°	12	0.8	F-G	
0.9 - 1.0	2	260°	12	0.6	E-F	
0.9 - 1.0	2	260°	12	0.6	F-G	
0.9 - 1.0	3	260°	15	1.0	C-D	
1.0 - 1.1	3	260°	15	1.0	D-E	
0.9 - 1.0	3	260°	15	1.0	E-F	
0.9 - 1.0	3	260°	15	0.8	D-E	
0.9 - 1.0	3	260°	15	0.8	E-F	
0.9 - 1.0	3	260°	15	0.6	D-E	
0.9 - 1.0	3	260°	15	0.6	E-F	
0.9 - 1.0	3	260°	15	0.4	D-E	
0.9 - 1.0	6	246°	15	1.0	E-F	
0.9 - 1.0	7	248°	9	1.0	F-G	
0.9 - 1.0	8	248°	12	1.0	F-G	
0.9 - 1.0	8	248°	12	0.8	F-G	
0.9 - 1.0	9	248°	15	1.0	B-C	
0.9 - 1.0	9	248°	15	1.0	C-D	
0.9 - 1.0	9	248°	15	1.0	D-E	
0.9 - 1.0	9	248°	15	1.0	E-F	
0.9 - 1.0	9	248°	15	0.8	D-E	
0.9 - 1.0	9	248°	15	0.8	E-F	
0.9 - 1.0	9	248°	15	0.6	D-E	

• possible wind turbine location

5.4 Chapter five – key concepts

BOX 5.5 chapter five key concepts

a5 On the CFD modelling requirements

- It is fundamental to specify appropriately the turbulence model, the domain size and the boundary conditions as they determine to a large extent the accuracy of the solution.

b5 On the configuration of the CFD tests

- The commercial CFD code FLUENT 6.1.18 was used for the simulations carried out in this study. The geometry was meshed using an unstructured tetrahedral grid in a domain of 1150 m X 704 m X 400 m with a velocity profile specified in the inlet boundary, outflow for the outlet, symmetry for the top and side boundaries and wall for the bottom boundary with standard wall functions. The realizable k-epsilon turbulence model was used for computing the solution.

c5 On the results

- The velocity profile of the u-component of velocity was maintained reasonably well along an empty domain and in all the simulations.
- The location that showed less potential for the integration of small wind turbines was between buildings A-B for case one θ=260° and B= 9 m at any height.
- The highest velocity ratio computed was found between buildings D-E for orientation θ=260° and B= 15 m (case three) at h_5 (z/H = 1).

5.5 Chapter five – reference list

1. Augenbroe, G. (2004) Trends in building simulation. In: Malkawi, A.M. and Augenbroe, G., (Eds.) *Advanced Building Simulation*, pp. 4-24. New York, UK: Spon Press]

2. Alamdari, F. (1994) Applications of CFD in the Built Environment. *FLOVENT user meeting proceedings.* BSRIA. 14 pp

3. Chen, Q. and Zhai, Z. (2004) The use of Computational Fluid Dynamics tools for indoor environmental design. In: Malkawi, A.M. and Augenbroe, G., (Eds.) *Advanced Building Simulation*, pp. 119-140. New York, UK: Spon Press]

4. Chen, Q.Y. (2004) Using computational tools to factor wind into architectural environment design. *Energy and Buildings* **36** (12):1197-1209.

5. Gan, G. (2006) Simulation of buoyancy-induced flow in open cavities for natural ventilation. *Energy and Buildings* **38** (5):410-420.

6. Hirano, T., Kato, S., Murakami, S., Ikaga, T. and Shiraishi, Y. (2006) A study on a porous residential building model in hot and humid regions: Part 1--the natural ventilation performance and the cooling load reduction effect of the building model. *Building and Environment* **41** (1):21-32.

7. Tan, G. and Glicksman, L.R. (2005) Application of integrating multi-zone model with CFD simulation to natural ventilation prediction. *Energy and Buildings* **37** (10):1049-1057.

8. Yang, T., Wright, N.G. and Etheridge, D.W. (2006) A comparison of CFD and full-scale measurements for analysis of natural ventilation. *International Journal of Ventilation* **4**, 337-348.

9. Yang, T. (2004) CFD and field testing of a naturally ventilated full-scale building. *PhD. Dissertation. University of Nottingham, UK.* 227 pp.

10. Baskaran, A. and Kashef, A. (1996) Investigation of air flow around buildings using computational fluid dynamics techniques. *Engineering Structures* **18** (11):861-873.

11. Hu, C.-H. and Wang, F. (2005) Using a CFD approach for the study of street-level winds in a built-up area. *Building and Environment* **40** (5):617-631.

12. Lien, F.S. and Yee, E. (2004) Numerical modelling of the turbulent flow developing within and over a 3-D building array, Part I: A High-Resolution Reynolds-Averaged Navier–Stokes Approach. *Boundary-Layer Meteorology* **112** 427-466.

13. Lien, F.S., Yee, E. and Wilson, J.D. (2005) Numerical modelling of the turbulent flow developing within and over a 3-D building array, Part II: A Mathematical Foundation for a Distributed Drag Force Approach. *Boundary-Layer Meteorology* **114** 243–283

14. Lien, F.S. and Yee, E. (2005) Numerical modelling of the turbulent flow developing within and over a 3-D building array, Part III: A Distributed Drag Force Approach, its Implementation and Application. *Boundary-Layer Meteorology* **114** 287–313

15. Stathopoulos, T. and Baskaran, B.A. (1996) Computer simulation of wind environmental conditions around buildings. *Engineering Structures* **18** (11):876-885.

16. Hien, W.N., Liping, W., Chandra, A.N., Pandey, A.R. and Xiaolin, W. (2005) Effects of double glazed facade on energy consumption, thermal comfort and condensation for a typical office building in Singapore. *Energy and Buildings* **37** (6):563-572.

17. N. Meroney, R., Leitl, B.M., Rafailidis, S. and Schatzmann, M. (1999) Wind-tunnel and numerical modeling of flow and dispersion about several building shapes. *Journal of Wind Engineering and Industrial Aerodynamics* **81** (1-3):333-345.

18. Skote, M., Sandberg, M., Westerberg, U., Claesson, L. and Johansson, A.V. (2005) Numerical and experimental studies of wind environment in an urban morphology. *Atmospheric Environment* **39** (33):6147-6158.

19. Xiaomin, X., Zhen, H. and Jiasong, W. (2006) The impact of urban street layout on local atmospheric environment. *Building and Environment* **41** (10):1352-1363.

20. Chang, C.-H. and Meroney, R.N. (2003) Concentration and flow distributions in urban street canyons: wind tunnel and computational data. *Journal of Wind Engineering and Industrial Aerodynamics* **91** (9):1141-1154.

21. Neofytou, P., Venetsanos, A.G., Vlachogiannis, D., Bartzis, J.G. and Scaperdas, A. (2006) CFD simulations of the wind environment around an airport terminal building: Urban Air Quality Modelling. *Environmental Modelling & Software* **21** (4):520-524.

22. Huang, S., Li, Q.S. and Xu, S. (2007) Numerical evaluation of wind effects on a tall steel building by CFD. *Journal of Constructional Steel Research* **63** (5):612-627.

23. COST (2007) Best practice guideline for the CFD simulation of flows in the urban environment. Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B., (Eds.) COST Action 732. Quality assurance and improvement of micro-scale meteorological models. 52 pp. Hamburg, Germany:

24. de Wit, S. (2004) Uncertainty in building simulaton. In: Malkawi, A.M. and Augenbroe, G., (Eds.) *Advanced Building Simulation*, pp. 25-59. New York, UK : Spon Press]

25. Schatzmann, M. and Britter, R. (2005) Quality assurance and improvement of micro-scale meteorological models. *COST 732 report. European Science Foundation.* 6 pp.

26. Sørensen, D.N. and Nielsen, P.V. (2003) Quality control of computational fluid dynamics in indoor environments. *Indoor Air* **13** (1):2-17.

27. WS Atkins Consultants. Best practice guidelines for marine applications of Computational Fluid Dynamics. Marnet-CFD. 84 pp. Available at: <https://pronet.wsatkins.co.uk/marnet/publications/bpg.pdf> Accessed: 17/07/07

28. Cook, N.J. (1985) *The designer's guide to wind loading of building structures*, Garston, Watford : Building Research Establishment, Dept. of the Environment ; London ; Boston : Butterworths.

29. Mertens, S. (2006) *Wind energy in the built environment*. Multi-Science. Essex, UK. 169 p.

30. Anderson, J.D. (1995) *Computational Fluid Dynamics. The basics with applications.* Singapore: McGraw-Hill, Inc.

31. Bhaskaran, R. and Collins, L. (2003) Introduction to CFD Basics . Cornell University. Available on line at:ttp://instruct1.cit.cornell.edu/courses/fluent/cfd/intro.pdf. Accessed 25/07/05.

32. Baskaran, A. and Stathopoulos, T. (1992) Influence of computational parameters on the evaluation of wind effects on the building envelope. *Building and Environment* **27** (1):39-49.

33. FLUENT Inc. (2005) FLUENT 6.2 User's guide. Manual. Available on line at: <https://www.nottingham.ac.uk/is/services/software/local/fluent> Accessed 27-6/06.

34. Mertens, S., van Kuik, G. and van Bussel, G. (2003-) Performance of an H-Darrieus in the Skewed Flow on a Roof. *Journal of Solar Energy Engineering* **125** (4):433-440.

35. Hall, R.C. (1996) *Evaluation of modelling uncertainty. CFD modelling of near-field atmospheric dispersion.*, WS Atkins Consultants Ltd, UK: Project EMU final report.

36. FLUENT Inc. (2004) GAMBIT 2.2 User's guide. Manual. Available on line at: <https://www.nottingham.ac.uk/is/services/software/local/fluent> Accessed 27-6/06.

37. Hargreaves, D.M. and Wright, N.G. (2007) On the use of the k–e model in commercial CFD software to model the neutral atmospheric boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics.* **95** 355–369

38. Richards, P.J. and Hoxey, R.P. (1993) Appropriate boundary conditions for computational wind engineering models using the k-[epsilon] turbulence model. *Journal of Wind Engineering and Industrial Aerodynamics* **46-47** 145-153.

39. Blocken, B., Carmeliet, J. and Stathopoulos, T. (2007) CFD evaluation of wind speed conditions in passages between parallel buildings--effect of wall-function roughness modifications for the atmospheric boundary layer flow: The Fourth European and African Conference on Wind Engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **95** (9-11):941-962.

40. Cornell University (2002) FLUENT tutorials. Available on line at: <http://instruct1.cit.cornell.edu/courses/fluent/index.htm> Accessed 4/6/05.

41. Universidad de Oviedo. (2003) FLUENT. Prácticas de simulación numérica de Mecánica de Fluidos. Available on line at: <http://uniovi.es/Areas/Mecanica.Fluidos/investigacion/s3a4_publicaciones_pub.php> Accessed 28/9/05.

42. FLUENT Inc. (2005) FLUENT 6.2 UDF Manual . Manual. Available on line at: <https://www.nottingham.ac.uk/is/services/software/local/fluent> Accessed 27-6/06.

6 comparison and discussion of **results**

6.1 Introduction

In this chapter the experimental results obtained with the wind tunnel modelling are presented and discussed against those obtained with the computational fluid dynamics (CFD) simulations. The influence that building orientation (θ) and building passage width (B) have on wind velocity at different heights (h_1-h_5) is examined. Conclusions and recommendations on the optimal requirements for simulating wind flow over urban environments around buildings are provided.

6.1.1 Chapter overview

This chapter is divided in four main sections (Box 6.1). The first section presents a summary of the required procedures to correctly simulate wind flows around buildings in urban environments using boundary layer wind tunnels and computational fluid dynamics.

BOX 6.1 chapter six overview

6.1 The summary of findings	A summary of findings of chapter 4 and 5 is presented in this section. The locations of higher velocity ratios are presented.
6.2 The analysis of results	The compared results of the variations in height, in passage width (B) and building orientation on wind velocity are discussed and analysed.
6.3 Other uses	The visualisation capabilities for modelling air flows around buildings with CFD packages are briefly presented.
6.4 The key points	In this section some conclusions and recommendations relevant to architectural building design are presented. The chapter's key points are summarised in box 6.2.

The second section discusses the general similarities and differences between the experimental and the computed results using CFD as a visualization tool. Additionally, the influence that building orientation (θ) and building passage width (B) have on wind velocity at different heights (h_1-h_5) are examined. In the third section, the key concepts of the chapter are summed up. Finally, in the last section of this chapter conclusions and recommendations – relevant to architectural building design – on the simulations of wind flows in the atmospheric surface layer for urban environments using wind tunnels and CFD are presented.

6.1.2 Summary of wind tunnel and CFD results

The experiments conducted on the SBE wind tunnel and the CFD simulations made using FLUENT 6.1.18, aimed to investigate possible locations for small wind turbines between a hypothetical building complex integrated by seven tower buildings of 50m height. The objective of both experimental and computational modelling was to simulate the real conditions of the atmospheric surface layer and afterwards to translate those numerical findings into practical architectural design decisions. To illustrate some of those design decisions some case studies will be discussed further in the next chapter.

To ensure the correct simulation of the atmospheric surface layer, the experimental procedure observed, where possible, the geometric and dynamic similarity requirements: the mean velocity profile was reasonably well reproduced and reference was presented that the turbulence intensity profile has been previously properly simulated. Similar cautions were implemented during the CFD simulations. Special attention was made on the specification of boundary conditions; the inlet velocity profile was specified during the simulations to concur with that measured with the wind tunnel and described by the power law (Eqn. 4.1).

The results presented in chapter five showed reasonable agreement with those obtained during the wind tunnel experiments (chapter four). The contour plots (Figures 4.12 and 5.6) of the velocity ratios obtained show a similar trend for all the cases. It was concluded that in both, experimental and computed data, the highest velocity ratio was located between buildings D-E at h_5 in case three (θ= 260˚ and B= 15 m).

In general terms, the case that showed more potential to integrate small wind turbines between the studied building complex – in both, wind tunnel and CFD results – was case three (θ= 260˚ and B= 15 m), specifically between buildings D-E. Other points that were found in both experimental and computed results that showed the potential for integrating wind turbines are presented in Table 6.1.

Table 6.1 Concurring locations of experimental and computed velocity ratios.

CASE	Vel. Ratio	z/H	Buildings
1	0.9 – 1.0	0.8	F - G
2	0.9 – 1.0	1.0	E - F
2	0.9 – 1.0	1.0	F - G
2	0.9 – 1.0	0.8	E - F
2	0.9 – 1.0	0.8	F - G
2	0.9 – 1.0	0.6	E - F
3	1.0 – 1.1	1.0	D - E
3	0.9 – 1.0	1.0	E - F
3	0.9 – 1.0	0.8	D - E
3	0.9 – 1.0	0.6	D - E
3	0.9 – 1.0	0.4	D - E
8	0.9 – 1.0	1.0	F - G
8	0.9 – 1.0	0.8	F - G
9	0.9 – 1.0	1.0	E - F

It can be appreciated from Tables 4.1, 5.1 and 6.1 that higher velocity ratios were registered in cases two and three and between buildings D-E, E-F and F-G; in all cases, these are the buildings located further north. It is for this reason that in the following sections the analysis will focus between those buildings at different reference points (h_1-h_5). Special attention will be placed to case three (θ= 260˚ and B= 15 m) as the highest velocity ratio was registered with both wind tunnel modelling and CFD simulations.

6.2 Comparison of results

The study of local wind flows and the measurement of the wind speed are vital in wind resource assessments for successfully siting wind turbines. Higher wind velocities increase the power output of wind turbines; therefore, in this section, the points where the highest velocity ratios were found will be used to compare the results obtained with wind tunnel measurements and the CFD simulations.

6.2.1 Influence of height, building passage and building orientation on wind velocity

Influence of variations in height (h_1-h_5) on wind velocity

The cases two and three presented more locations with the highest velocity ratios, especially between buildings F-G and D-E. To investigate how the velocity varies at those specific locations at different heights, a comparison of the vertical flow variation between the model prediction (CFD) and the experimental data (WT) for case two (θ= 260˚ and B= 12 m) between buildings F-G, three (θ= 260˚ and B= 15 m) between buildings D-E, case eight (θ= 248˚ and B= 12 m) between buildings F-G and case nine (θ= 248˚ and B= 15 m) between buildings D-E was carried out.

Figure 6.1 shows a comparison of the velocity ratios variation; the plots are made to a Z_{ref} of 50m.

The results are generally satisfactory, especially for those points located at z/Z_{ref} = 0.40, z/Z_{ref} = 0.60 and z/Z_{ref} = 1. The maximum discrepancies between the experimental and numerical data appeared at z/Z_{ref} = 0.20 and z/Z_{ref} = 0.80, in the order of 11% for this particular case. The difference between correlations was not statistically significant showing only a 5% deviation between the measured and the computed results.

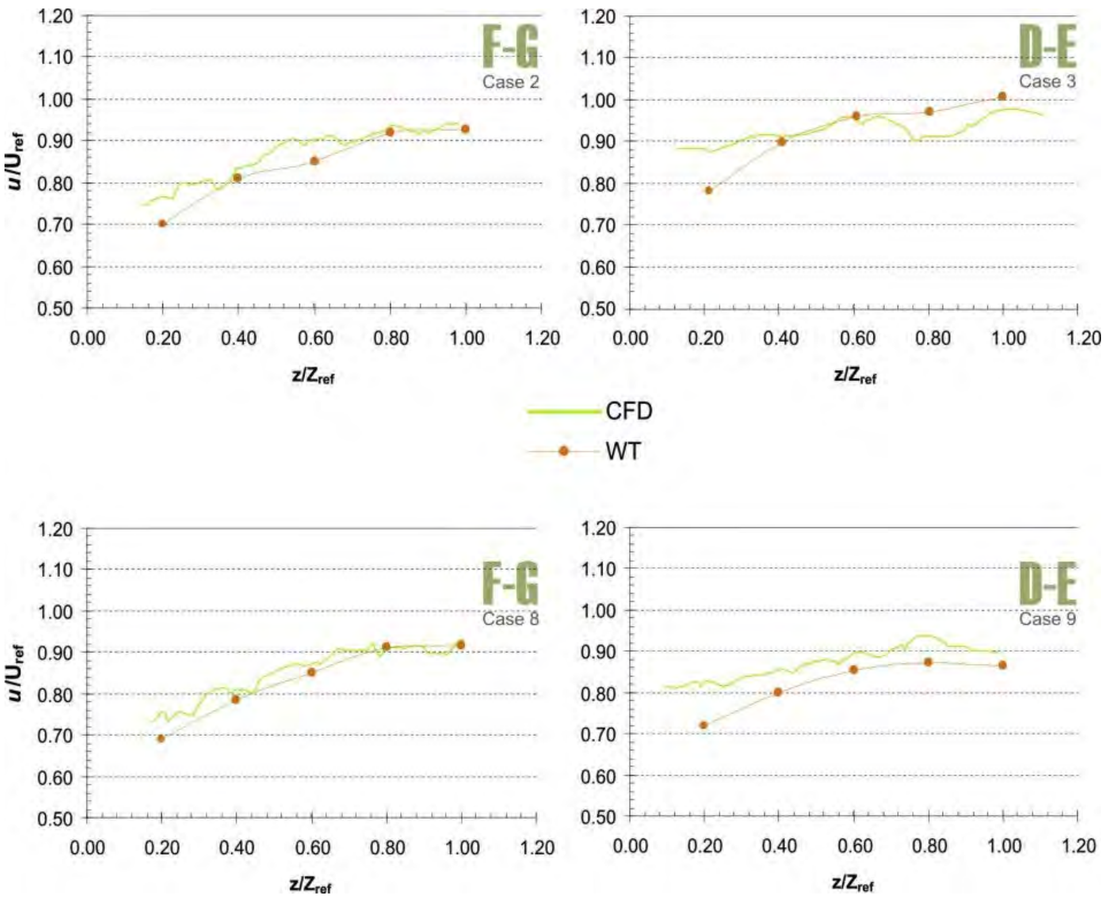


Fig. 6.1 Comparison of predicted vertical flow variation between buildings F-G for θ= 260˚, B= 12 m and θ= 248˚, B= 12 m and between buildings D-E for θ= 260˚, B= 15 m and for θ= 248˚, B= 15 m.

Additionally, it can be observed from Figure 6.1 that the differences between the computed and measured results are more apparent near the ground at h_1 and that they show better agreement as height increases. One possible explanation for this observable fact can be related with the treatment used at the near wall region (roughness and mesh specifications at pedestrian level) for the CFD simulations in this study. The near wall treatment significantly influences the accuracy of numerical solutions, because it is in that region that the solution variables have large gradients, and the momentum and other scalar transports occur most vigorously. Therefore, accurate representation of the flow in the near wall region determines successful predictions of turbulent flows[1].

In FLUENT there are two approaches to modelling the near wall region: the wall function approach and the near wall model approach. According to FLUENT Inc[1]., the wall function approach is commonly used because it is economical, robust, and reasonably accurate. There are two choices of wall function approaches: standard wall functions and non-equilibrium wall functions[I].

In the CFD simulations carried out for this study the standard wall functions provided by the simulation software were used. These wall functions are applied as an alternative approach to compute the wall shear stress, reducing the number of grid points in the wall-normal direction and therefore the computational costs.

By using the wall function approach, the wall shear stress is computed from the assumption of a logarithmic velocity profile between the wall and the first

---

[I] In this approach, the turbulence models are modified to enable the viscosity-affected region to be resolved with a mesh all the way down to the wall, including the viscous sub-layer, this means that more control volumes are required throughout the domain, significantly increasing the computational demands[1].

computational node in the wall-normal direction, which is valid for equilibrium boundary layers and fully developed flows[2]. Given the fact that the wall functions determine the way the governing equations are calculated at pedestrian level, it would be fair to assume that it is well established approach.

Yet, even though it has been established in chapters 4 and 5 that there is a vast amount of published work regarding the study of wind patterns in urban environments using boundary layer wind tunnels and CFD simulations, the published literature regarding the near wall treatment on CFD simulations of the atmospheric surface layer is scarce. Many authors do not elaborate further on the procedure carried out when specifying boundary conditions or wall functions[3],[4],[5]. Moreover, only the most recent sources[6],[2],[3],[7] on the numerical simulation of flows around buildings in urban areas deal with the subject and they show wide discrepancies on the use of nomenclature and on the use of near wall treatment approaches (Table 6.2).

Table 6.2 Reported values of aerodynamic roughness length, physical roughness height and roughness constant used for the calculation of standard wall functions.

AUTHORS	Aerodynamic roughness length		Physical roughness height		Roughness constant		Wall function
	Authors' nomenclature		Authors' nomenclature		Authors' nomenclature		
1 Blocken et al.	y_0	0.03	K_S	1E-04	CK_S	0.5	Launder and Spalding standard wall functions
2 Hargreaves and Wright	z_0	0.01	ε_R	$20z_0$	C_S	0.5	Launder and Spalding standard wall functions
3 Mertens	z_0	0.03	$\overline{H}$	$13z_0$	C_{hb}	0.75	Launder and Spalding standard wall functions

Some authors use a mesh near the bottom boundary that is aligned with the wall surface and is also refined and stretched within the viscous boundary layer grid[8], or that use a geometric progression in the vertical grid direction[3].

In the light of these different approaches, the guidelines were consulted to further investigate the use of wall functions.

Recently published guidelines[2] on the simulation of flows in urban environments indicate that the roughness of the bottom wall have major influence on the inflow boundary profiles and suggest an approach of an explicit modelling of the roughness elements that are used in a corresponding wind tunnel study and then using only smooth wall boundary conditions. This approach was acknowledged and discarded by Mertens[7] for being very time consuming. Nevertheless, modelling the individual roughness blocks, instead of the average roughness that they produce has shown to led to a significant improvement of the computed results compared to the results obtained with an approaching flow over a smooth flat wall[2].

COST guidelines[2] also suggest that for low-Reynolds number approach, a very fine mesh resolution in wall-normal direction is required. This leads to a first computational node that should be positioned at a non-dimensional wall distance of

(**Eqn. 6.1**)
$$y^+ = \frac{yu_\tau}{\mu} \approx 1$$

Where in equation **6.1**, y is the distance normal to the wall and u_τ is the shear velocity, computed with equation **6.2**, from the time averaged wall shear stress τ_w and μ the kinematic viscosity.

(**Eqn. 6.2**)
$$u_\tau = \left(\frac{\tau_w}{\rho}\right)^{\frac{1}{2}}$$

Nevertheless, it should be noted that in FLUENT[1] this non-dimensional distance from the wall to first node is expressed by y^*, and when y^* >11.225, a logarithmic profile is employed in the near wall region for the calculation of mean velocity yields when using the standard wall functions. This value of y^* is not calculated with equation 6.1[II] but instead it is calculated with:

(Eqn. 6.3)
$$y^* \equiv \frac{\rho C_\mu^{\frac{1}{4}} k_p^{\frac{1}{2}} y_p}{\mu}$$

In equation 6.3, ρ is the air density, k_p is the turbulent kinetic energy at the point P and y_p is the distance from the wall to the centre point P and μ is the kinematic viscosity. This value of y^* should be between 30 and 300 for smooth walls[III].

This requirement of a high mesh resolution in wall-normal direction has been further explored by Blocken et al[6]. Additionally, they include in the argument that for wall functions that are expressed by a physical roughness[IV] K_s, three other requirements should be **simultaneously satisfied.** The four requirements are schematically represented in Figure 6.2 and are:

[II] FLUENT Inc[1]. indicates that the quantities of y^+ and y^* are approximately equal in equilibrium turbulent boundary layers; therefore, hereafter the symbol y^+ will be used when referring to the non-dimensional distance from the wall to the first control volume node.
[III] Note that different authors provide different ranges for the non-dimensional value y^+, for example: Blocken et al[6]. indicate a value of y^+ between 30 and 100 and COST[2] guidelines suggest a range of $30<y^+<500$.
[IV] For the built environment the average physical roughness height is equal to the average building height[7].

- A sufficiently high mesh resolution in the vertical direction close to the ground surface (at least two or three control volume layers below pedestrian height 1.75 ~ 2.00 m)[V].
- A horizontal homogeneous approach flow.
- A distance y_P from the centre point *P* of the wall-adjacent cell to the wall (bottom boundary condition) that is larger than the physical roughness height K_s i.e. ($y_P > K_s$).
- A relationship between the physical roughness K_s and the aerodynamic roughness length z_0 denoting that z_0 is only a small fraction of K_s. (See Table 6.2)[VI]

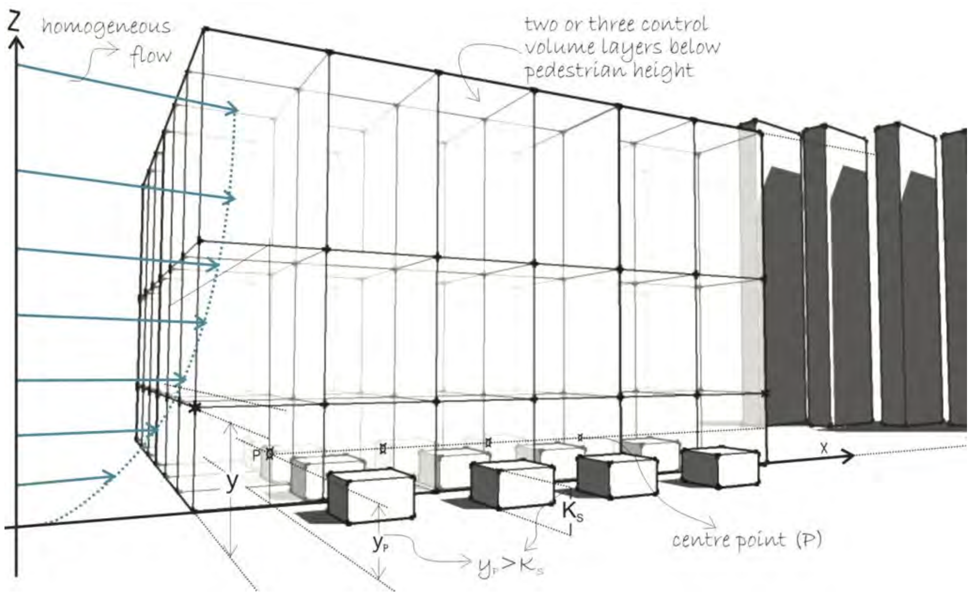


Fig. 6.2 Schematic of the cells next to the bottom boundary wall. (Not to scale).

[V] This recommendation is especially important if the results are to be used for dispersion studies.
[VI] COST[2] and Blocken et al[6] both suggest the relationship $K_s \approx 30z_0$, but it can be appreciated from Table 6.3 that the value used by Blocken et al[6] for the reported simulations does not satisfy this equation.

From the above four requirements it is worth noting two important things: 1) most commercial CFD codes advise to abide by the requirement of $y_P > K_s$, in order to avoid inaccuracies in the solution[6],[1] and that; 2) it is a mistake to set K_s equal to z_0.[3]

This is explained because for a fully rough surface (when $K_s^+ > 90$), the roughness length z_0 and the roughness height K_s are analytically related by:

(**Eqn. 6.4**) $$K_s = z_0 \exp(\kappa B),$$

where κ is the von Karman constant ($\boldsymbol{K} \approx 0.4$) and $B \approx 8.5$ is the constant in the logarithmic velocity profile for rough surfaces[2]. This leads to the following relation:

(**Eqn. 6.5**) $$K_s \approx 30z_0,$$

Equation **6.5** shows that the roughness height is one order larger than the roughness length, but also brings to light the fact that this often generates very large computational cells, clearly meaning a poor mesh resolution that conflicts with the first of the four requirements.

Blocken et al[6] provide a very eloquent example to illustrate this conflict, and so demonstrating why satisfying all four requirements is generally impossible for simulations of the atmospheric surface layer. The authors' explanation utilised the following scenario:

For a grass-covered plain with a typical aerodynamic roughness length z_0 = 0.03 m, the value of K_s can be obtained using equation 6.5 to comply with the fourth requirement, resulting in a K_s =0.9 m. This value obliges y_P to be set at least 0.9 m to observe requirement number three; that $y_P > K_s$. As a result the height of the first cell is y = 1.8 m at full scale values. With this procedure, the addition of heights of the first three control volume layers, need to obey requirement number one, gives a total height of 5.4 m, assuming no geometric progression in the vertical grid direction (Z) is applied as illustrated in Figure 6.2. This would mean that those three cells would not be located within the pedestrian height of 1.75 ~ 2.00 m as mentioned in the first requirement.

The implications of this example for simulating wind flows in an urban area, as with the one described in chapter three of this study, are of great importance. This is because the typical aerodynamic roughness length parameter z_0, which characterises the roughness of the ground surface, changes by more than two orders of magnitude over the range of terrain found in nature. Consequently, in an urban site where one would expect to find buildings that offer the potential for integrating wind turbines, where the average building height is around 25 m, the value of z_0 ranges from 0.3 m to 0.8 m[9], yielding to even bigger control volumes and no accurate solutions for near ground flow can thus be obtained.

However, most commercial CFD codes provide the user with only one option to modify the wall functions and it is based on data for sand grain roughness, i.e. data obtained for roughened pipes and channels, which clearly does not correspond to an adequate simulation of roughness in urban areas.

To overcome this shortcoming, the authors presented in Table 6.2, present different approaches for the near wall treatment, which vary in complexity.

Mertens[7] used the standard wall functions provide in FLUENT, modifying the default values of physical roughness height and roughness constant to $K_s = 13z_0$ and $C_{Ks} = 0.75$. Blocken et al[6] approach also involved modifying K_s and C_{Ks} default values (see Table 6.3). Additionally, to obtain suitable values for y^+ and to account for the fact that the non-dimensional roughness height FLUENT holds for sand-grain roughness, the modelled geometry used was at a reduced scale and a correction in the reference wind velocity was made.

Finally, Hargreaves and Wright[3] approach is more complex. The authors tried to replicate Richards and Hoxey[10] boundary conditions using the values presented in Table 6.2. They found that in their simulation the turbulent kinetic energy profile was not well maintained throughout the domain as in Richards and Hoxey's because of the differences in the treatment of the turbulence kinetic energy and dissipation rate in the cell next to the ground. The proposal was to replace the standard k-**ε** turbulence model provided in FLUENT by another one written by them, using the User-Defined Scalars (UDSs) and then using the laminar solver, modifying the specification of air viscosity and specifying a symmetry boundary condition for the bottom face of the domain. It is clear that this approach requires an ample knowledge of wind engineering and a certain degree of familiarity with computer programming.

Even though the three approaches mentioned above, are good examples of trying to overcome the limitations of the commercial CFD packages, regarding near wall treatment, there are certain aspects worth noting. The value for z_0 used by

Mertens[7] and Blocken et al[6] are those corresponding to typical UK farmlands, nearly flat countryside, fences or low boundary hedges[9] and they are not representative for urban areas. In addition, the scaling approach used by Blocken et al[6] is only valid if the flow features to be modelled around the buildings are Re independent. With those considerations in mind, the Hargreaves and Wright[3] approach appears to have wider applicability and to produce reliable results, nevertheless it is complex and requires a degree of understanding that goes beyond the understanding of the average designer, therefore surpassing the research objective of this work of identifying the minimum standards and requirements that an architect or environmental designer should observe for a reliable simulation of wind flows around buildings in urban environments using CFD.

The approach used for the simulations carried out in this research regarding the near wall treatment, is similar to Mertens', whose simulations of the atmospheric surface layer were intended for evaluating the sizing and siting of wind turbines in urban environments. In addition to the above discussion, wall functions were utilised for the CFD simulations in this study because they provide an economical solution in terms of CPU use and most importantly, the limitations of this approach are more evident at pedestrian level, therefore having low impact on the results that are of interest in this research.

Nevertheless, the definition of y^+ and the specification of the turbulence kinetic energy and dissipation in the cell next to the ground by the use of standard wall functions when representing the atmospheric surface layer is a fact often ignored by computational wind engineers as noted by Blocken et al.[6] and Hargreaves and Wright[3], and should not be neglected, especially for studies involving pedestrian comfort and dispersion studies. This is also of high importance for studies that intend to evaluate wind conditions in urban environments around low rise edifications like small structures, detached or terrace houses.

Influence of variations in passage width (B) on wind velocity

To investigate the influence of the variations in the passage width between buildings on wind velocity, a detailed analysis of the results presented as velocity ratios for both CFD and wind tunnel simulations was carried out between building D-E, for cases one, two and three, where building orientation remains constant, with an incidence angle of θ= 260˚, and for cases seven, eight and nine, where building orientation remains constant at θ= 248˚.

The results of this analysis are presented in a plot of the velocity ratios (*y*-axis) as function of the length ratio B/H (*x*-axis) at the five measured and computed heights (h_1-h_5), illustrated in Figure 6.3. Where B is the gap between buildings and H is the building height (Fig. 6.3).

From Figure 6.3 it can be appreciated that the computed and measured results show a fairly good agreement. For cases one, two and three, the influence of variations in distance between buildings D-E in the compared results, show that the discrepancies increase as the ratio B/H decreases; in other words, the closer the buildings, the higher the difference between computed and measured velocity ratios. This is difference in results is more apparent in Figure 6.3 at h_4 and h_5. It has to be noted that this is not the case at h_1; this could be due the near wall region treatment as explained in the previous section.

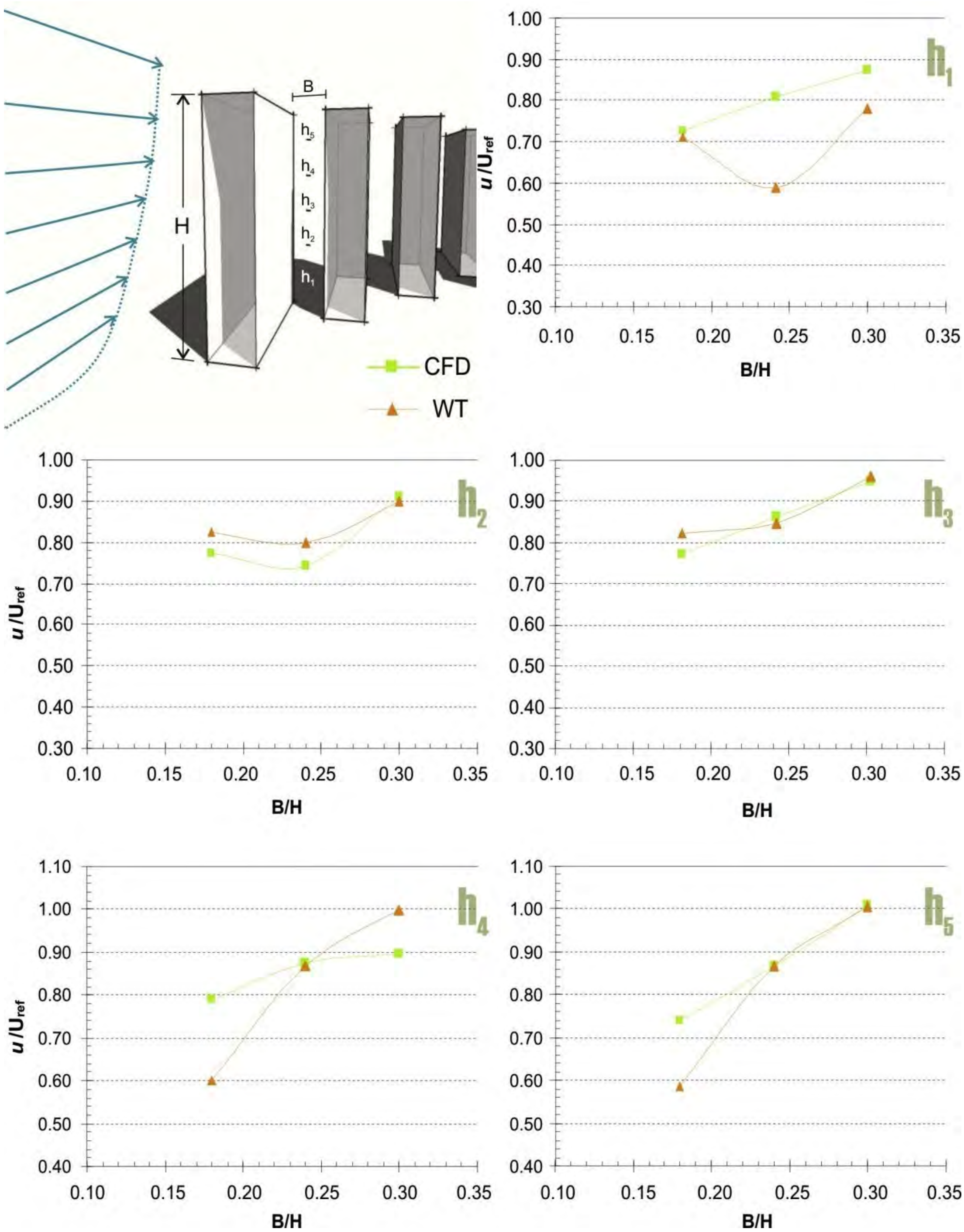


Fig. 6.3 Comparison of predicted vertical flow variation between buildings D-E for θ= 260˚ (cases one, two and three) at h_1 (z/H = 0.2), h_2 (z/H = 0.4), h_3 (z/H = 0.6), h_4 (z/H = 0.8) and h_5 (z/H = 1).

Comparable conclusions can be drawn from Figure 6.4, (cases seven, eight and nine) where it can be noticed that the computed and measured results show a fairly good agreement, with less discrepancy than the previous cases.

Likewise, for cases seven, eight and nine, the discrepancies between computed and measured velocity ratios increase as the ratio B/H between buildings D-E decreases. Further CFD studies could help to confirm these results and to further clarify the influence of B/H ratio on wind velocity at different heights.

Nevertheless, similar findings in the discrepancies between wind tunnel experiments and CFD simulations were reported by Stathopoulos[11] on the work of Bottema. Stathopoulos acknowledged that the maximum discrepancies between the experimental and numerical data were found in highly complex re-circulating flow regions. Such regions occur in the present study. This can be appreciated in Figure 6.3 at h_5 for B/H= 0.18 and further visualized in Figure 6.5.

Those zones of highly complex re-circulating flow can be visualized in Figure 6.5 a, where the streamlines coloured by velocity magnitude are shown for case one (θ= 260° and B/H= 0.18) in a horizontal plane at h_5. A zone of flow separation in the northern face of building A that extends to almost 1/3 of the building passage can be observed in Figure 6.5 b. It is in that passage that the lowest velocity ratios were found. In contrast, there is no flow separation at the southern face of building E is observed (Figure 6.5 c), and the flow separation at the northern face of building C appears closer to the wake zone. The flow in the passage between buildings C-E interacts, creating higher velocity ratios.

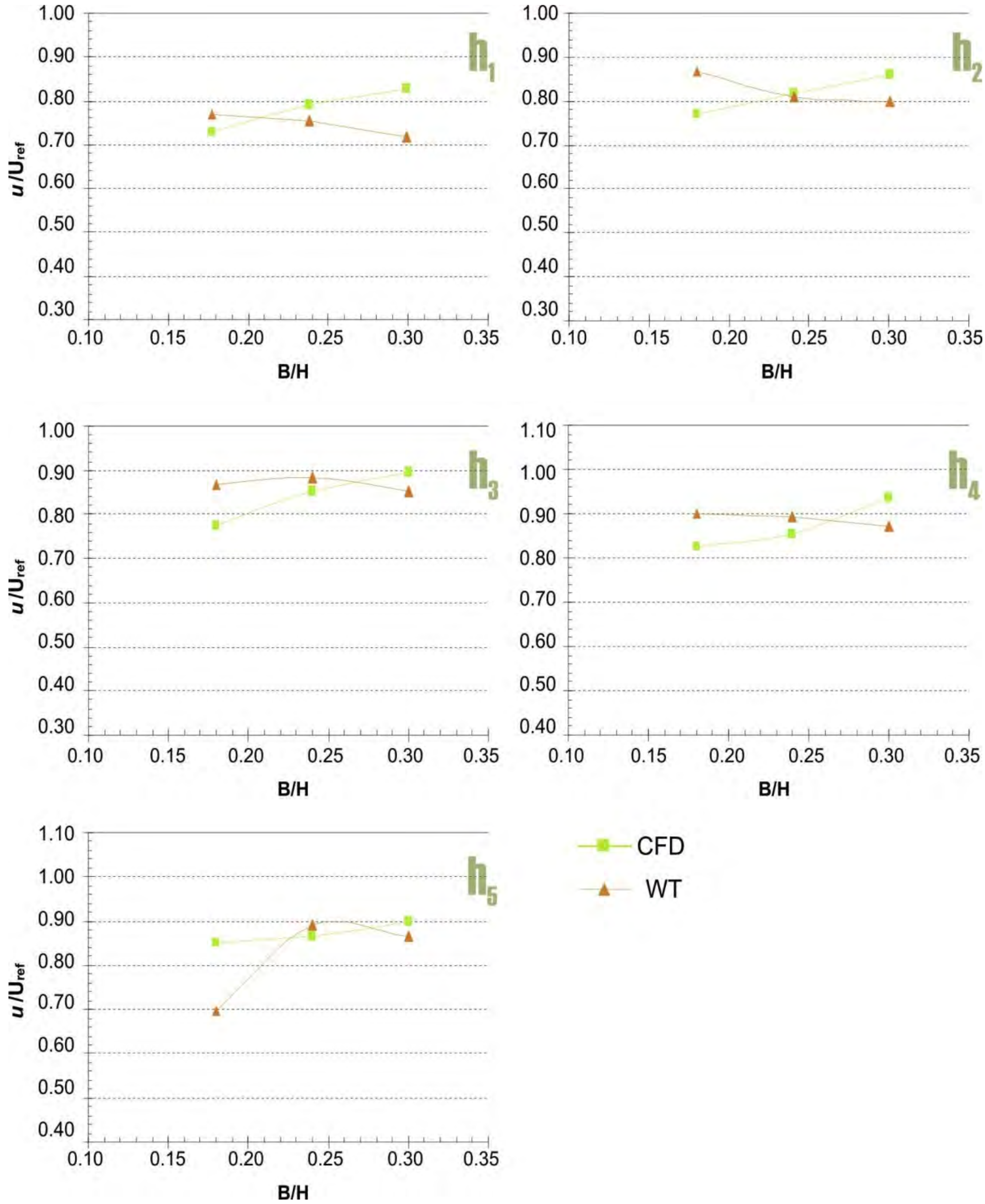


Fig. 6.4 Comparison of predicted vertical flow variation between buildings D-E for θ= 248˚ (cases seven, eight and nine) at h_1 (z/H = 0.2), h_2 (z/H = 0.4), h_3 (z/H = 0.6), h_4 (z/H = 0.8) and h_5 (z/H = 1).

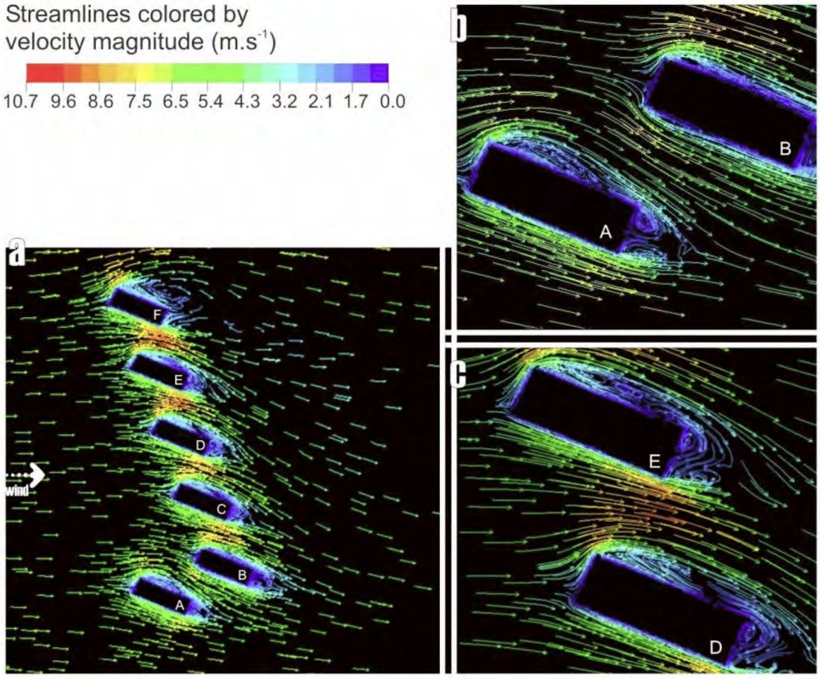


Fig. 6.5 Streamlines case one (θ= 260˚ B=9) at h_5 (z/H = 1).

On several occasions, the increased wind speed values in building passages have been attributed to a decrease of the flow section, i.e. between buildings that are closer together, a rather common assumption supported mainly by experience and often associated with the so called Venturi effect. The results presented in Figures 6.3 and 6.4 and Table 6.3 show that this is not entirely accurate. Furthermore, as the ratio B/H increases, the velocity ratios increase as well, for cases one, two and three consistently, in both wind tunnel measurements and CFD simulations.

Table 6.3 Computed and measured velocity ratios increasing as ratio B/H increases. Cases one, two and three.

	CFD			WT		
z/H	B/H= 0.18	B/H= 0.24	B/H= 0.30	B/H= 0.18	B/H= 0.24	B/H= 0.30
0.20	0.73	0.81	0.87	0.71	0.59	0.78
0.40	0.77	0.74	0.91	0.83	0.80	0.90
0.60	0.77	0.86	0.95	0.82	0.84	0.96
0.80	0.79	0.87	0.90	0.60	0.87	1.00
1.00	0.74	0.87	1.01	0.59	0.87	1.01
	0.76	0.83	0.93	0.71	0.79	0.93

The same effect was reported by Blocken et al[6] in a similar study where flow between buildings of variable passage distance was studied at pedestrian level. They classified the flow type between buildings according to the variation in the distance between them. For distances between buildings in the order of 2 m, Blocken et al[6] regarded the passage flow as resistance flow, for distances between buildings greater than 30 m; the flow was regarded as isolated flow and for passages in the order of 10 m the flow was regarded as interaction flow, where the higher amplification factors were found.

This amplification factor (K) is defined as the ratio of the mean wind speed at certain location to the mean wind speed at the same location in the absence of buildings (free stream velocity). Since the velocities are taken at the same height for a specific location, this ratio directly indicates the influence of buildings on the local wind velocity, i.e. values greater than the unity indicate an augmentation in the velocity.

It is possible to produce contour plots for the amplification factor K in FLUENT by creating a custom field function for a reference height. Figure 6.6 illustrates the contours of amplification factor K in a horizontal plane at h_5 for case one (θ= 260˚ and B/H= 0.18).

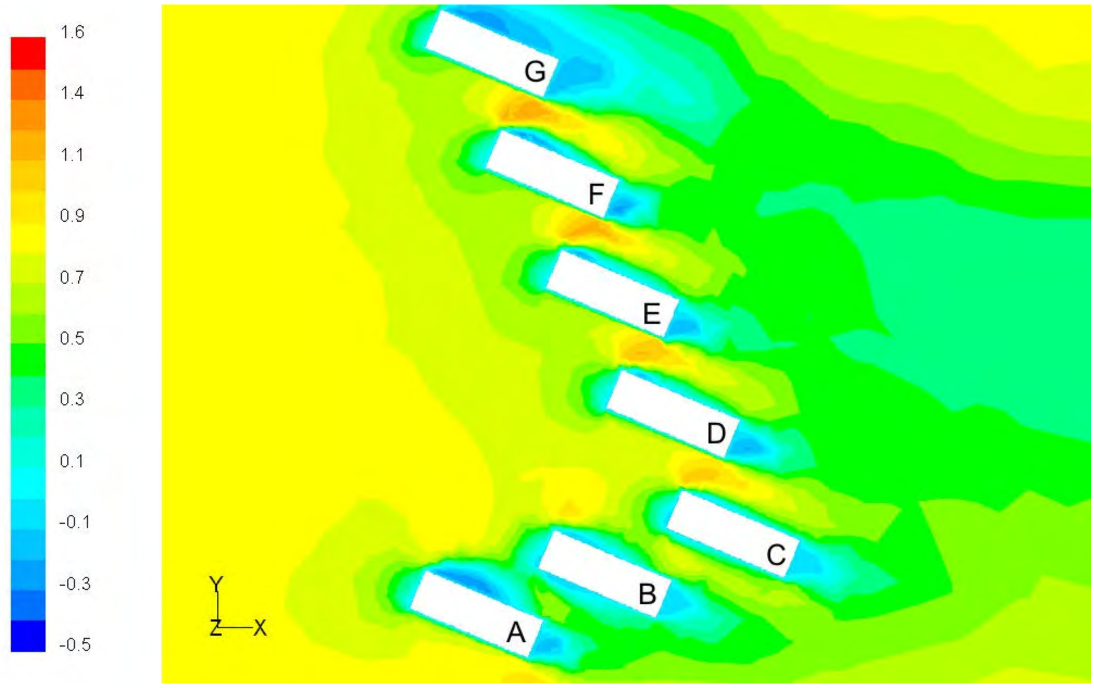


Fig. 6.6 Contours of amplification factor K at h_5 ($z/H = 1$) in case one (θ= 260˚ B=9).

Augmentation in the velocity can be found between buildings where the amplification factors (K) are greater than the unity. For case one at h_5 this occurs between buildings F-G with K= 1.3, between buildings E-F with K=1.2 and between buildings D-E with K=1.1. These results suggest a rather weak augmentation in velocity due to the Venturi effect. One possible explanation is the buildings are creating a wind blocking effect and, as discussed above, the flow resistance found in the narrower passage widths. According to Blocken et al[6]., a better term to describe the flow conditions in passages between buildings, similar to the ones in this study, might be 'channelling effect' which refers to the changes in wind direction due to the effect of the buildings rather than changes in wind speed.

Moreover, the results presented and discussed in this section show that with an increase of 6 m between the studied buildings an increase of almost 25% in wind velocity can be obtained.

Influence of variations in building orientation on wind velocity

To investigate the influence that the three tested building orientations (θ) had on wind velocity, a series of plots of the measured velocity ratio u/U_{ref} at h_1 (z/H=0.2) and h_5 (z/H=1.0) between the buildings D-E and E-F were created.

From Figure 6.7 it can be noticed that the wind speed ratios between buildings D-E and E-F at h_5 (z/H=1.0) in the case of wind direction θ=260° are considerably lower than those in the case of wind direction θ=236° for the narrowest building passage width (B/H=18). At first instance, this could be attributed to the fact that θ=236° represents the cases where the long side of the buildings are parallel to the prevailing winds. Nevertheless, when passage width between buildings increases, the results showed that the highest velocity ratios were for those cases with orientation θ=260° (case one, two and three) and the lowest velocity ratios for those cases with orientation θ=236° (four, five and six). This suggests that orientation is interacting with passage width to modify the wind conditions between buildings.

Therefore the relationship between building orientation and passage width is not the same between all buildings and at the different heights as can be appreciated in Figure 6.8, where the same plots were created at h_1.

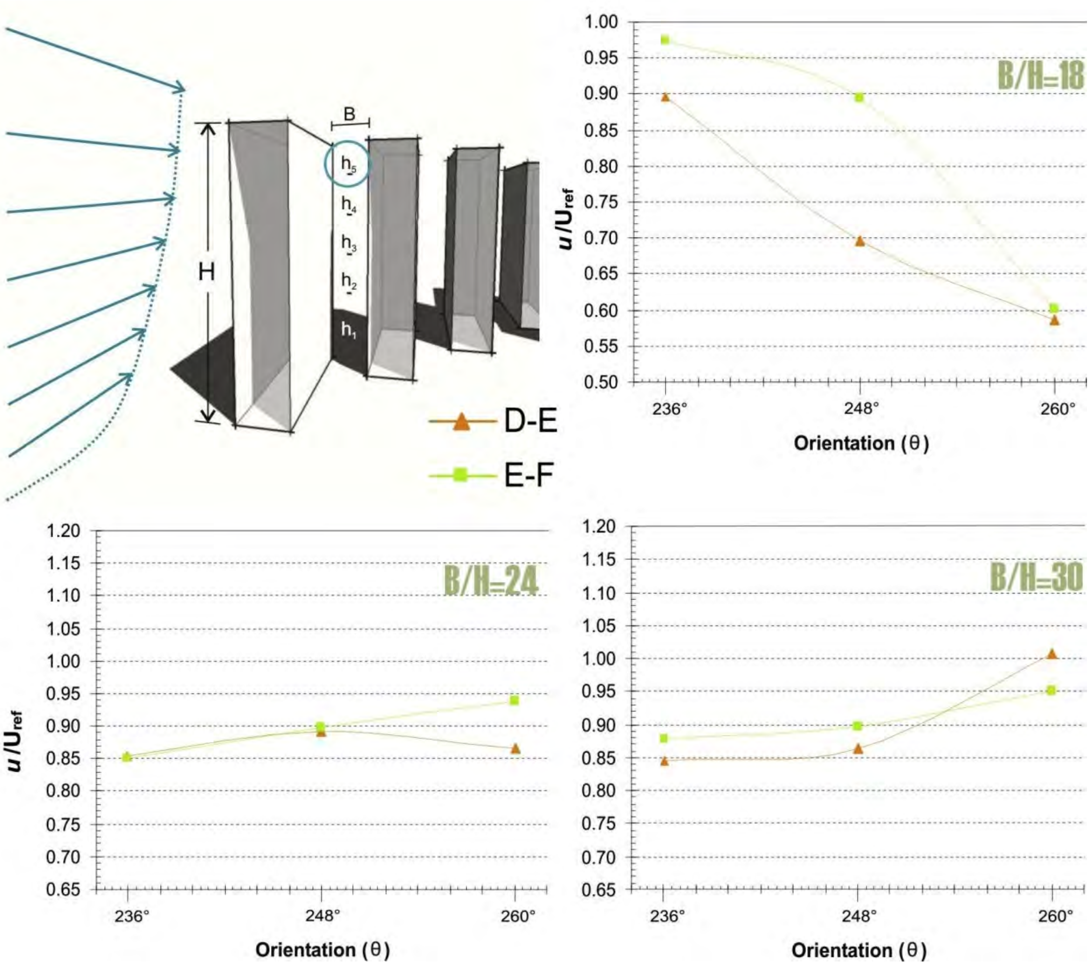


Fig. 6.7 Variation in velocity ratios at h_5 (z/H = 1) between buildings D-E and E-F for orientations (θ= 236°, 248° and 260˚) at B/H=18, B/H=24, and B/H=30.

Comparing Figures 6.7 and 6.8 it can be suggested that for passage width B/H=30, the orientation that produces higher velocity ratios is θ=260°, at both, pedestrian and top levels.

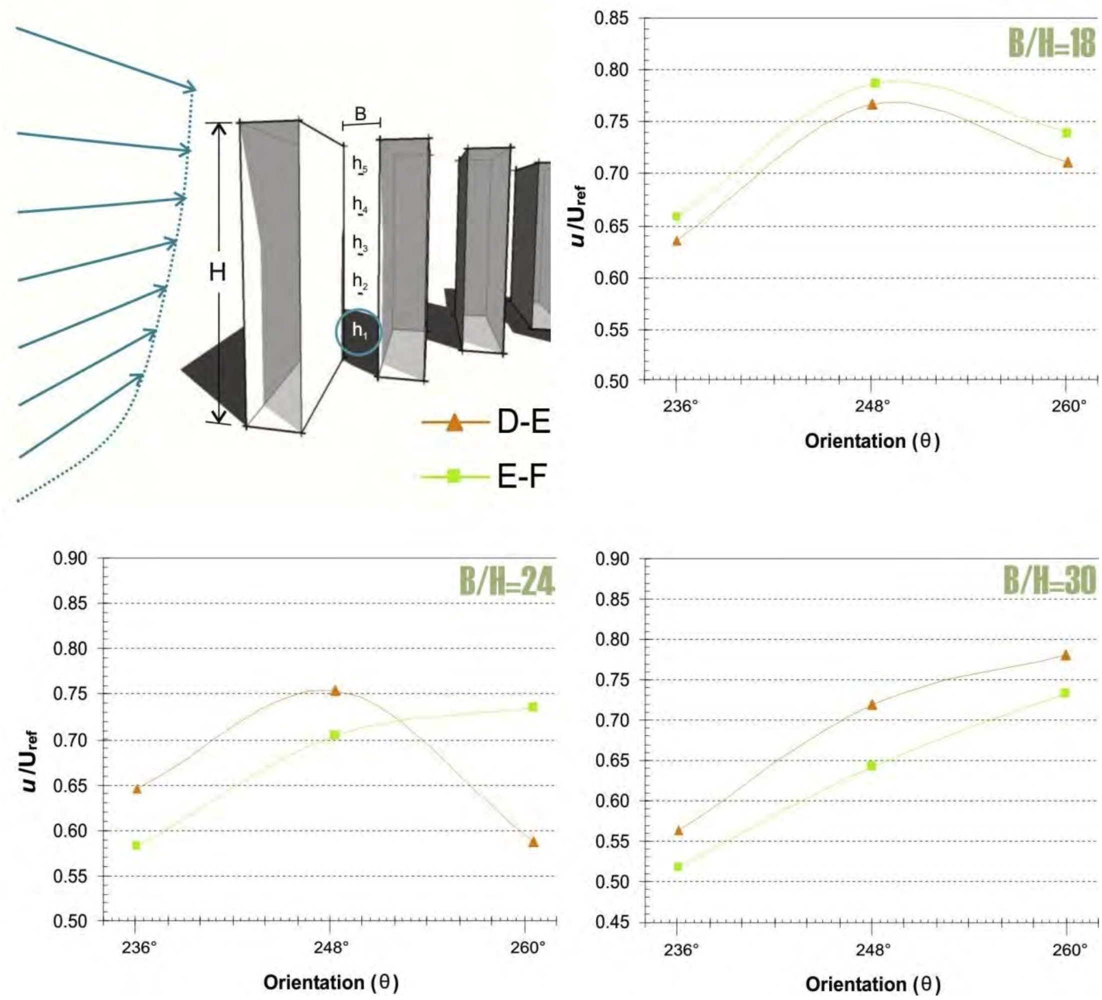


Fig. 6.8 Variation in velocity ratios at h_1 (z/H = 0.2) between buildings D-E and E-F for orientations (θ= 236°, 248° and 260°) at B/H=18, B/H=24, and B/H=30.

Nevertheless, when buildings are closer, i.e. for passage width B/H=18, the effects of orientation on wind velocity are different at pedestrian level (h_1) and at h_5. For B/H=18, orientation θ=248° produces higher wind velocities at pedestrian levels, and at h_5, orientation θ=236° produces higher wind velocities. This might suggest that building design would benefit with a different orientation at pedestrian level to avoid high wind speeds and that the building could be rotated at the top, facing an orientation with high wind speeds for wind turbine integration. However, because this is not conclusive between all buildings, design decisions would significantly improve by taking advantage of the CFD visualization capabilities.

6.3 Visualisation

CFD plots provide data visualisation that allows the graphical representation of various aspects of the studied flow to be easily observed. Some of those aspects are difficult to observe experimentally.

An example of the visualisation capabilities of the CFD commercial packages is the observation of changes in wind direction around buildings. This is of great utility when assessing wind conditions around buildings for the siting of wind turbines.

The flow angle over roofs of sharp edged buildings is of great importance when selecting the type of wind turbine, as can be appreciated from Figure 6.9, the flow direction close to the edges of a sharp edged building is not parallel to the roof and separation occurs. As a result, the wind flow makes an angle with the roof that varies according to the roof inclination. This angle is called skew angle[12]. This phenomenon is of special importance when selecting a building mounted wind turbine. As it can be appreciated in Figure 6.9, flow separation results in a region with low velocities, high turbulence level and recirculation of the flow at the roof and sides of a building. Ideally, the recirculation region should be avoided for siting a wind turbine; this is why it is crucial to know the size of the recirculation region. This is one case where the visualisation capabilities of the CFD simulations are very useful.

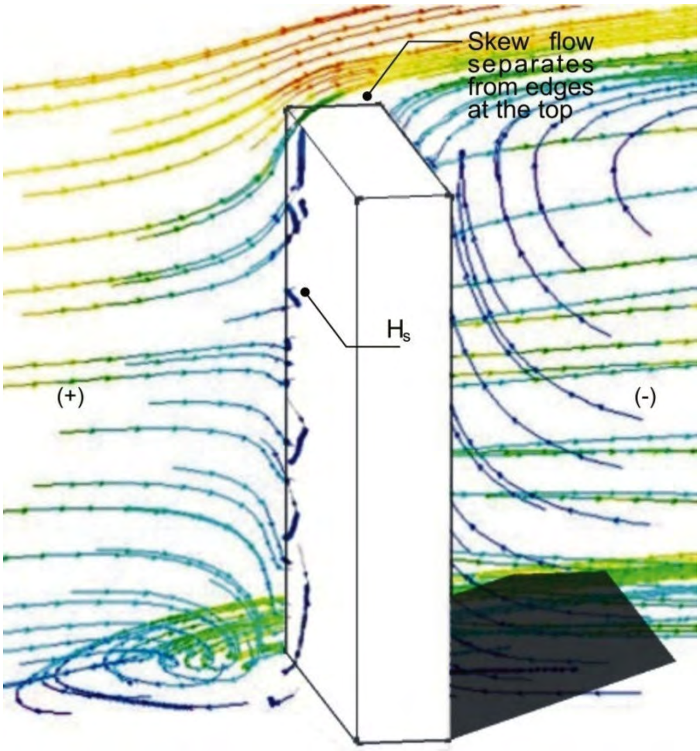


Fig. 6.9 CFD calculation of velocity vectors showing separation at top of the building, stagnation point and the skew flow on the top.

The height at which the stagnation point (H_s) is located is of aerodynamic importance because it gives the highest pressure point on the upwind building façade. It can be appreciated as well in Figure 6.9. Knowing the location of the stagnation point is of special important for natural ventilation studies as it characterises the flow around a building[VII]. Additionally, at the downwind roof edge, the CFD simulation shows a change in orientation of the separation streamlines.

Similarly, skewed flows that are found besides sharp edged buildings can be clearly visualised as illustrated in Figure 6.10.

[VII] It is possible to give an estimation of H_s, as a function of the roughness properties where the building is located. See for example Mertens[7].

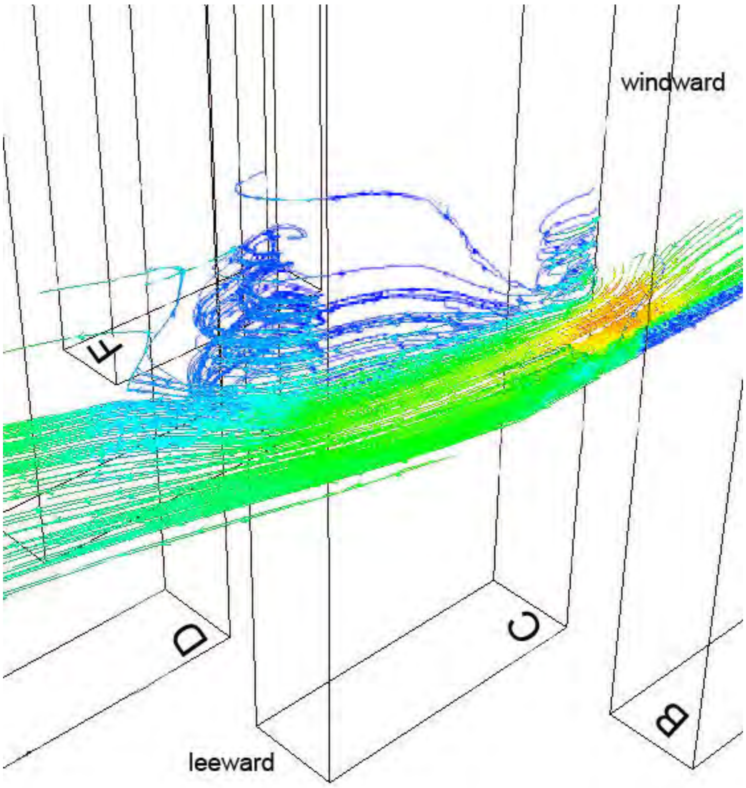


Fig. 6.10 CFD calculation of velocity vectors showing skew flow at the sides of the building (h_3).

Recirculation regions appear in the area around the corner of the north side and east side of building C and the strong wind blows into the space between buildings B and C.

The employment of CFD techniques as a visualization tool when simulating flow conditions in urban environments can be of optimal use for the architect or designer, especially at early stages during the design process, mainly because the designer can have an easy access to the required tools. In other words, CFD simulations using a practical, but reasonably fine, mesh containing approximately 1 million control volumes can be carried out with a realistic desktop computer with 1GByte of RAM and a Pentium 4 processor. It can be expected that the solutions will take around 2 to 10 of hours to converge, depending on the problem definition, representing an achievable computing effort for a practical architectural design applications[VIII].

6.4 Chapter six – key concepts

In spite of the shortfalls mentioned in this chapter for the CFD simulations carried out in this study, and the fact that some authors have already acknowledged that the process of comparison between computational and experimental results appears to be problematic on its own, the comparison made in this study of computed and measured velocity ratios around the cluster of buildings of interest A, B, C, D, E, F and G, at the five measured heights (h_1-h_5), is generally satisfactory. The maximum discrepancies between the experimental and numerical data were found at ground level. It can be concluded, that for wind flow in the atmospheric surface layer, the results of CFD simulations are highly dependent on the type of roughness modification applied to the wall functions[6]. The key findings of this chapter are summarised in Box 6.2.

[VIII] See for example Stathopoulos[11].

BOX 6.2 chapter **six key concepts**

On the agreement of results

- Results of wind tunnel and CFD simulations showed reasonable agreement.
- Case three (θ= 260° and B= 15 m) presented the greatest potential to integrate small wind turbines with both methods, especially between buildings D-E.

On the influence of variations in height on wind velocity

- The differences between measured and computed results were more apparent at ground level (h_1).
- The near wall region treatment on CFD simulations significantly influences the accuracy of numerical solutions.
- The procedure to correctly represent the urban flows in the near wall region is not well established for commercial CFD packages.
- The definition of y^+ and the specification of K and ε in the cell next to the ground by the use of standard wall functions is often ignored in CFD simulation of urban flows.

On the influence of variations in passage width on wind velocity

- Computed and measured results presented a fairly good agreement.
- The discrepancies between computed and measured results increase as the ratio B/H decreases.
- Amplification factor (K) were in the range of 1.2 suggesting a rather weak augmentation in velocity due to the Venturi effect.

On the influence of variations in building orientation on wind velocity

- Wind speed ratios vary in every case suggesting that orientation is interacting with passage width to modify the wind conditions between buildings.
- Although results are not conclusive, it could be suggested that building design would benefit with a different orientation at pedestrian level to avoid high wind speeds and that the building can be rotated at the top to harness high wind speeds for wind turbine integration.

6.5 Chapter six – reference list

1. FLUENT Inc. (2005) FLUENT 6.2 User's guide. Manual. Available on line at: <https://www.nottingham.ac.uk/is/services/software/local/fluent> Accessed 27-6/06.

2. COST (2007) Best practice guideline for the CFD simulation of flows in the urban environment. Franke, J., Hellsten, A., Schlünzen, H. and Carissimo, B., (Eds.) COST Action 732. Quality assurance and improvement of micro-scale meteorological models. 52 pp. Hamburg, Germany.

3. Hargreaves, D.M. and Wright, N.G. (2007) On the use of the k–e model in commercial CFD software to model the neutral atmospheric boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics.* **95** 355–369

4. N. Meroney, R., Leitl, B.M., Rafailidis, S. and Schatzmann, M. (1999) Wind-tunnel and numerical modeling of flow and dispersion about several building shapes. *Journal of Wind Engineering and Industrial Aerodynamics* **81** (1-3):333-345.

5. Xiaomin, X., Zhen, H. and Jiasong, W. (2006) The impact of urban street layout on local atmospheric environment. *Building and Environment* **41** (10):1352-1363.

6. Blocken, B., Carmeliet, J. and Stathopoulos, T. (2007) CFD evaluation of wind speed conditions in passages between parallel buildings--effect of wall-function roughness modifications for the atmospheric boundary layer flow: The Fourth European and African Conference on Wind Engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **95** (9-11):941-962.

7. Mertens, S. (2006) *Wind energy in the built environment*. Multi-Science. Essex, UK. 169 p.

8. Huang, S., Li, Q.S. and Xu, S. (2007) Numerical evaluation of wind effects on a tall steel building by CFD. *Journal of Constructional Steel Research* **63** (5):612-627.

9. Cook, N.J. (1985) *The designer's guide to wind loading of building structures*, Garston, Watford : Building Research Establishment, Dept. of the Environment ; London ; Boston : Butterworths.

10. Richards, P.J. and Hoxey, R.P. (1993) Appropriate boundary conditions for computational wind engineering models using the k-[epsilon] turbulence model. *Journal of Wind Engineering and Industrial Aerodynamics* **46-47** 145-153.

11. Stathopoulos, T. (2006) Pedestrian level winds and outdoor human comfort: Urban Civil Engineering (UCE), Impact of wind and storms on city life and built environment., Impact of wind and storms on city life and built environment. *Journal of Wind Engineering and Industrial Aerodynamics* **94** (11):769-780.

12. Mertens, S., van Kuik, G. and van Bussel, G. (2003-) Performance of an H-Darrieus in the Skewed Flow on a Roof. *Journal of Solar Energy Engineering* **125** (4):433-440.

7 architectural integration

7.1 Introduction

In this chapter the architectural integration of six vertical wind turbines will be presented. The annual electricity output, the payback period and the savings in CO_2 emissions will also be presented for the case that presented higher velocity ratios (θ= 260˚, B= 15).

7.1.1 Chapter overview

This chapter is divided in two main sections (Box 7.1). The first section presents a summary of recommendations to correctly simulate wind flows around buildings in urban environments using boundary layer wind tunnels and computational fluid dynamics. The second section illustrates the architectural integration of the small vertical wind turbines in case three (θ= 260˚, B= 15), along with the calculations for the predicted annual electricity generation, the CO_2 savings and the payback period of the system.

BOX 7.1 chapter seven overview

7.2 The recommendations	Recommendations on wind tunnel and CFD simulations in urban environments.
7.3 The predicted annual electricity generation	Calculations for the annual electricity output, CO_2 savings and payback period. Using case three and six Turby wind turbines at selected locations.

7.2 The recommendations

It has become apparent in the last few years that much of the early data gathered from CFD studies will need to be updated with information that takes into account a range of velocity profiles and turbulence characteristics, especially in the near wall region. Some wind effects can now be satisfactorily estimated using existing theory of fluid mechanics, wind tunnel modelling or empirical relationships. The complex and variable characteristics of atmospheric conditions and the almost infinite variety in the geometry of built forms, produce an endless variety of wind effects, increasing the necessity of individually simulate each architectural project, especially when undertaking wind energy projects. Nevertheless, as a result of this study, some recommendations can be made when planning a similar project involving the representation of an atmospheric surface layer in urban environments using wind tunnel measurements and CFD calculations.

However, it is expected that it will probably take many more years of carefully conducted wind tunnel studies and CFD simulations, together with full scale correlation, to produce sufficient information to allow a significant increase in theoretical means of estimating wind effects.

7.2.1 Recommendations on wind tunnel experiments

During the wind tunnel experiment:

- Fulfil geometric and dynamic similarity requirements (section 4.1.3).
- Do not exceed a blockage of 5% of surface area (section 4.1.3).

- Measure incident and approach velocity profiles, turbulent kinetic energy and Reynolds stresses, in addition to the measurements of interest for the planned study.

7.2.2 Recommendations on CFD simulations

The flows around buildings even in simple surrounding environments, let alone in complex urban settings are still extremely difficult to predict by computational methods[1]. However, there is increasing evidence[2],[3],[4] that for such problems CFD techniques may provide adequate responses for cases related primarily with environmental issues.

Additionally, from the results presented in chapter six, it is clear that for CFD simulations of atmospheric surface layer in urban environments: (1) the choice of the turbulence model and wall functions, (2) the modelling of the approach-flow wind speed profile and (3) the construction of the computational grid have a major impact on the computed results[5].

Reasonably good results can be expected for CFD simulations of turbulent flows in the atmospheric surface layer in urban environments if the following recommendations are observed:

- Abide by the size domain recommendation in section 5.1.4.
- Use the calculated value of turbulent dissipation rate and the measurements of incident velocity profile, turbulent kinetic energy and Reynolds stresses

obtained with the wind tunnel experiment to specify inflow boundary conditions. (If using FLUENT CFD code this will be done using an UDF[i]).

- Top and side boundaries as in Hargreaves and Wright (Table 5.1).
- Bottom boundary condition is to include an adequate representation of aerodynamic roughness and wall functions as in Richards and Hoxey (Table 5.1).

7.3 Energy generation and saved CO_2 emissions

The buildings utilized in this study present a simple geometry with flat roof and sharp edges; an augmentation in wind velocity was reported in some of the cases by changing building orientation (θ) and building passage width (B). Cases with B/H = 0.18 presented a channelling effect that refers to the changes in wind direction due to the effect of the buildings rather than changes in wind speed.

From the results discussed in chapter six, and in agreement with Blocken et al[2] findings, it can be suggested that the optimum distances between buildings to actually observe an increase in velocity ratios are between 10 and 30 m. The maximum augmentation factor (K) found in this study is of 1.5 for case three (θ= 260° and B/H= 0.30) between buildings D-E for h_5.

---

[i] Only if no experimental data of this type is available, those profiles can be obtained from the assumption of an equilibrium boundary layer, see Richards and Hoxey[6].

If a small vertical wind turbine was to be located at that point, like the Turby[ii] - a revolutionary vertical axis developed by Mertens[7], especially designed for use in an urban or built-up environment with a swept area of 5.3 m^2 -, according to the wind frequency distribution presented in chapter two, it would generate 3610 kWh during a normal year (Table 7.1and Figure 7.1).

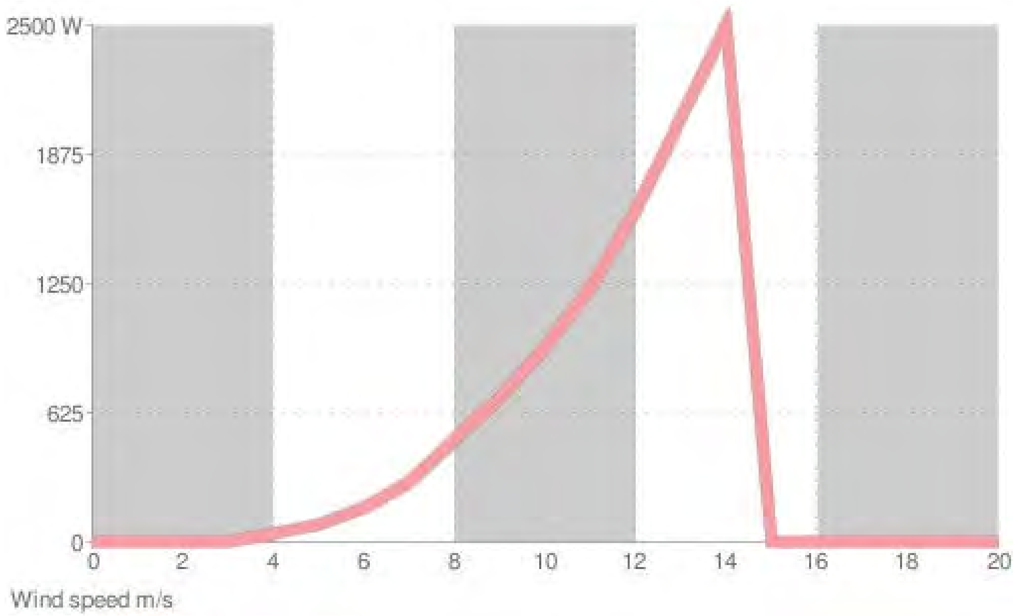


Fig. 7.1 Turby's power curve. From manufacturer: http://www.turby.nl/

[ii] Turby's power curve compared to other small wind turbines, including the quiet revolution qr5, is available at: http://www.bettergeneration.com/wind-turbines/comparing-vawts.html

Table 7.1 Energy calculation for six VAWT at selected locations.

ENERGY CALCULATION FOR TURBY

wind speed (m/s)	power (W)	frequency (%)	frequency (hrs/year)	energy (Wh)
0.0	0.0	0.0	0.0	0.0
2.7	100.0	41.5	3635.4	363540.0
5.5	312.0	38.3	3355.1	1046785.0
8.3	625.0	15.4	1349.0	843150.0
11.1	312.5	3.4	297.8	93075.0
13.9	2300.0	0.3	26.3	60444.0
			8663.64	2406994.0
				2407 kWh/year

location	augmentation factor (K)	energy (kWh/year)
DE h5	1.5	3610.5
DE h4	1.4	3369.8
DE h3	1.1	2647.7
EF h5	1.2	2888.4
EF h4	1.1	2647.7
CD h5	1.1	2647.7
		17811.8

Furthermore, according to the results here presented and using case three (θ= 260˚, B= 15) as the optimum example; six Turby wind turbines could be located between buildings D-E at h_3, h_4 and h_5, between buildings E-F at h_4 and h_5 and between buildings C-D at h_5 as illustrated in Figure 7.2.

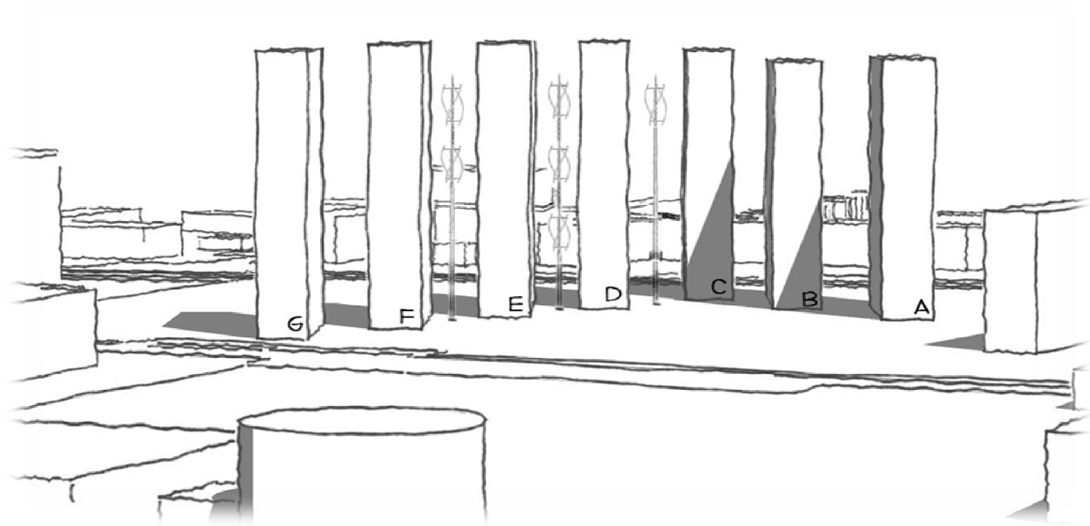


Fig. 7.2 Architectural wind turbine integration in case three (θ= 260˚, B= 15).

The electrical outputs shown in Table 7.1 are calculated in kWh per year. The British Research Establishment (BRE) holds data for the production of electricity within the UK. This includes all the impacts associated with the extraction and transport of fuels, production of electricity and distribution to the consumer. This gives an impact of 0.632 kg CO_2 equivalent per kWh of electricity. However, some studies do recommend the use of marginal electricity models based on the replacement of coal-fired generation with CO_2 impact figures of about 0.800 kg/kWh. The impact of electricity used within the study follows the BRE recommendation[8] of 0.632 kg CO_2 equivalent.

For assessing the CO_2 savings, the output from the micro-wind turbines is assumed to replace electricity sourced by the consumer from the low voltage grid (220–240 V) and the impact of this has been used to model the benefit of micro-wind generation. In other words, the 17, 812 kWh/year generated by the six wind turbines would be preventing 11.3 Tons of CO_2 to be released into the atmosphere.

In order to calculate the financial payback period, the cost of the complete system should be considered. The integrated cost of the turbine[III] with convertor, mast, foundations, accessories and installation is £12,195.00; making a total of £73,170.00 for the six wind turbines. In UK, an average of costs per unit charged by electricity suppliers[IV] is 26.776p/kWh and this is the rate applied to the turbine generation.

For assessing the financial payback period, it was considered that the cost of producing the 17, 812 kWh/year by the wind turbines, was the same to the price of that generated by the electricity supplier. That cost would be then £4,769 per year, and therefore, it would take 15.3 years to recover the initial cost of the complete system.

Taking that into account, the costs comparison between the six wind turbines' kWhs and traditionally generated electricity, it can be concluded that renewable energy is not yet competitive with traditional electricity generated from large power plants. Nevertheless, the fuel for turbines (wind) is free; there is neither maintenance required nor personnel to run it. Its operational costs are nil. Another advantage is that the electricity generated with the wind turbines do not need to be delivered to the customer – as the turbines are already there between the buildings. No costs for the use of electricity networks and no grid losses in those networks. (10 % of traditionally generated electricity is lost during transport and distribution). Additionally, it is expected that, when built in series, the price of the wind turbines like the Turby will reduce with about 20 – 40 %, making shorter their financial

[III] Prices of Turby wind turbine are available at http://www.turby.nl/

[IV] Electricity costs can be obtained as a reference from http://www.britishgas.co.uk/products-and-services/energy/electricity/fixed-price-2012.html.

payback period. Last but not least, the environmental benefits of wind energy generation must be also taken into the economy equation.

Box 7.2 summarises the annual energy production of the six wind turbines, their cost, financial payback period and the annual saved emissions in tonnes of CO_2. These figures will vary for different reasons:

- If the location of the wind turbine changes, the power output will vary as the augmentation factor (K) will also vary. Although wind velocity on the top of the buildings was not measured in this study, the computed results (as visualisations of wind flows illustrate) showed an increase on wind velocity at those points, making an interesting proposition for retrofitting.

- If a different model of wind turbine is selected. For example, a bigger vertical wind turbine like the 5kW qr5, designed by Quiet Revolution[V], it can generate up to 10, 000kWh/year at 5.8 $m.s^{-1}$ according to the manufacturer.

[V] Information available at http://www.quietrevolution.co.uk/

BOX 7.2 energy generated and saved emissions for case three

A The cost of the turbines	£73,170 for six vertical wind turbines. Turby 2.5kW @ $14 m.s^{-1}$
B The predicted electricity generation	The six turbines sited at the selected locations, will produce an estimated 17, 812 kWh in a typical year.
C The payback period	The financial payback period of the project is 15.3 years.
D The saved CO_2 emissions	Installing the chosen model of wind turbine at the selected locations will save 11.3 Tons of CO_2 emissions.

It should be noted that these results are considered to be optimistic as the calculations do not take account of turbulence effects. Also, although the general effect of the town terrain and surrounding buildings is accounted for, it is assumed that the buildings between on which the turbine is sited is not overshadowed by larger buildings, trees or other obstructions.

Finally, although variations in building shape were not investigated during wind tunnel experiments or CFD simulations in this study, aerodynamic building shapes might be used to further enhance wind speed. However, it is worth mentioning that in complex geometries, the procedure to generate the geometry for CFD simulations explained in section 5.1.4 will not be appropriate. Complex geometries need of a different approach, involving the use of an alternative software to generate the geometry and later importing it into Gambit as thoroughly explained by Adkins[9].

7.4 Chapter seven – reference list

1. Baskaran, A. and Stathopoulos, T. (1992) Influence of computational parameters on the evaluation of wind effects on the building envelope. *Building and Environment* **27** (1):39-49.

2. Blocken, B., Carmeliet, J. and Stathopoulos, T. (2007) CFD evaluation of wind speed conditions in passages between parallel buildings--effect of wall-function roughness modifications for the atmospheric boundary layer flow: The Fourth European and African Conference on Wind Engineering. *Journal of Wind Engineering and Industrial Aerodynamics* **95** (9-11):941-962.

3. Hargreaves, D.M. and Wright, N.G. (2007) On the use of the k–e model in commercial CFD software to model the neutral atmospheric boundary layer. *Journal of Wind Engineering and Industrial Aerodynamics.* **95** 355–369

4. Mertens, S. (2006) *Wind energy in the built environment*. Multi-Science. Essex, UK. 169 p.

5. Blocken, B. and Carmeliet, J. (2007) Validation of CFD simulations of wind-driven rain on a low-rise building facade. *Building and Environment* **42** (7):2530-2548.

6. Richards, P.J. and Hoxey, R.P. (1993) Appropriate boundary conditions for computational wind engineering models using the k-[epsilon] turbulence model. *Journal of Wind Engineering and Industrial Aerodynamics* **46-47** 145-153.

7. Mertens, S., van Kuik, G. and van Bussel, G. (2003-) Performance of an H-Darrieus in the Skewed Flow on a Roof. *Journal of Solar Energy Engineering* **125** (4):433-440.

8. Phillips, R., Blackmore, P., Blackmore, J., Clift, M., Aguiló-Rullįn, A. and Pester, S. (2007) Micro-wind Turbines in Urban Environments: An assessment. *Building Research Establishment* **BRE FB17**, 47 pp

9. Adkins, D. (2005) Computational Fluid dynamic study of fish-like propulsion. University of Leicester. MSc. Advanced Mechanical Engineering Final Report. 91 pp.

8 conclusions and further work

8.1 Introduction

In this chapter, final thoughts regarding the role that architects and wind energy systems play on achieving sustainable development are explored; key points for further research are outlined; and the thesis' conclusions are presented.

8.1.1 Chapter overview

This chapter is divided in three main sections as illustrated in Box 8.1. Section 8.2 explores some final thoughts on the role of the designer to achieve sustainable development. In section 8.3 four points for further work are outlined and finally in section 8.4 the conclusions of this thesis are presented.

BOX 8.1 chapter eight overview

The final thoughts	In this section the importance of the designer to actively engage with the environmental design principles and building physics concepts is explored.
The further work	Four lines for further enquiry are presented: modelling and simulation, building design, post-occupancy and energy storage and finally, education.
The conclusions	Conclusions on the integration of small wind turbines into the built environment and the simulation results are presented.

8.2 Final thoughts

"There is no reason why environmental design's science-based enquiry and architecture's traditional concern with form should not co-exist; indeed, why architectural form should not be enriched by an environmental agenda, as long as that agenda is not prescriptive"

S. Haggan, 2001[i]

The UK is beginning to develop the knowledge, technology and a policy framework to create sustainable places.

Some actions to raise public awareness and support for renewable energy through the installation of household and community scale schemes have been developed. This is a valuable way of demonstrating the contribution that renewable energy can make to the building's energy demands and to increase the degree of community involvement with renewable energy systems, especially small scale wind turbines.

However, it is important to be realistic about the capabilities of wind energy systems in urban environments. It is evident that wind turbines in urban surroundings receive lower wind speeds and have lower capacity factors. The reliability of the systems will improve as the technologies mature, the available quantity of historical wind data grows and designers gain experience with the modelling of wind flows in urban environments for siting wind turbines.

---

[i] Hagan, S. (2001) *Taking shape : a new contract between architecture and nature*, Oxford ;, Boston : Architectural Press.

The incentives and educational awareness programs carried out by the government and other organisations are exactly the reason why the architectural practice must respond professionally by delivering new buildings that successfully integrate these renewable energy technologies and by being able to tackle the existing building stock to retrofit these systems. This can only be achieved if the designer actively engages with the environmental design principles and improves his understanding of building physics.

In the case of wind resource assessment, the tools to simulate wind flows around buildings exist and in spite of their limitations, when properly used, they can provide with vast amounts of information that can lead to better design decisions and an enhanced performance of building integrated wind turbines.

8.3 Conclusions

Renewable energies are a critical element for achieving sustainable development. However, to successfully implement wind energy into the built environment three major issues have to be addressed:

- Wind resource assessment and wind characterisation around buildings
- Structural integration of wind turbines with buildings
- Install a wind turbine that has been especially design to work in urban environments

In this thesis, wind velocities were measured around 7 fifteen-storey towers at a reduced scale. The measurements were carried out, for nine different configurations, using a boundary layer wind tunnel and CFD simulations. The following conclusions were obtained:

- Computed and measured results showed reasonable agreement.
- Case three ($\theta = 260^{\circ}$ and B/H= 0.30) presented the greatest potential to integrate small wind turbines with both methods, especially between buildings D-E.
- The differences between measured and computed results were more apparent at ground level.
- The optimum distances between buildings to actually observe an increase in velocity ratios are between 10 and 30 metres.
- Six vertical wind turbines, of 2.5kW each, located at the selected augmentation points in case three can generate 17, 812kWh in a typical year, saving 11.3 Tons of CO_2 emissions.

Reasonably good results can be expected for CFD simulations of turbulent flows in the atmospheric surface layer in urban environments. The use of CFD as a visualisation tool is extremely useful at design stages in projects involving the integration of wind turbines. Nevertheless, the designer must be aware that the results of CFD simulations are highly dependent on different conditions, like the choice of the turbulence model, the modelling of the inlet wind velocity profile (using FLUENT code this will be done using an UDF), the construction of the computational grid and finally, the type of roughness modification applied to the wall functions.

8.4 Further work

The present work is not definitive. It should be seen as a step towards clarifying the relationship between architectural building design and the use of simulation tools for assessing the wind regimes in urban environments for wind turbine integration.

This research contributes towards future research on the understanding of wind regime in built up areas and the local flow effects around buildings and the role of the architect/designer to provide with informed design solutions for the integration of wind turbines in buildings.

The study was limited in that the quantitative analysis looked at only simple building forms for the integration of wind turbines. Nonetheless, the present research confirms many of the theoretical findings and benefits of BUWTs identified from the literature and research review presented in chapters 2, 4, 5 and 6. The conclusions presented on those chapters are consistent with the findings of related UK and international research in this area and it is expected that they would be substantiated by further work.

There are four particular lines of enquiry where further research would be beneficial:

- Modelling and simulation: There are still a lot of uncertainties relating to atmospheric surface layer flows in urban environments and to its design implications for BUWTs. In this matter, research is needed in the modelling of the implications that **high turbulence levels** in regions of high building density have on BUWTs' performance, fatigue loads and noise generation.

- Building design: The design of **aerodynamically shaped buildings** is not new; nevertheless, building design that considers aerodynamically ducted forms that are capable of accelerating wind speed and successfully integrates wind turbines is needed. This is especially true for new designs of high-rise buildings. Some additional research into the structural design of aerodynamically optimised buildings is also necessary.

- Post-occupancy and energy storage: The energy storage and the post-occupancy evaluation of the systems are closely related with energy generation. The production of hydrogen from wind energy offers the potential to store the energy and to create an almost zero-emissions energy system, using fuel cells to power buildings. Additionally, buildings already incorporating renewable or energy efficient systems could make a far greater contribution to sustainable development if their performance was measured and understood.

- Education: To truly achieve sustainability, a holistic design approach is required which goes beyond measurement and calculations to consider the quality of places. This obviously requires influencing built environment education and training curricula to improve environmental management and **sustainable design literacy** among both young people and professionals and in this way, ensure integrated approaches to the creation and refurbishment of sustainable buildings, spaces and places.

A

appendices

A.1 Appendix one

A.1.1 UDF code – inlet conditions

```
/***********************************************************************
  realvelprofile.c
  UDF for specifying a steady-state velocity profile boundary condition
 ***********************************************************************/

#include "udf.h"

DEFINE_PROFILE(inlet_velocity, thread, index)
{
  real x[ND_ND];
  real z;
  face_t f;

  begin_f_loop(f, thread)
    {
      F_CENTROID(x,f,thread);
      z = x[2];
      F_PROFILE(f, thread, index) = 5.4775 * pow ((z/50),.2275);
    }
  end_f_loop(f, thread)
}
```

A.2 Appendix two

A.2.1 Velocity profiles. Cases two to nine

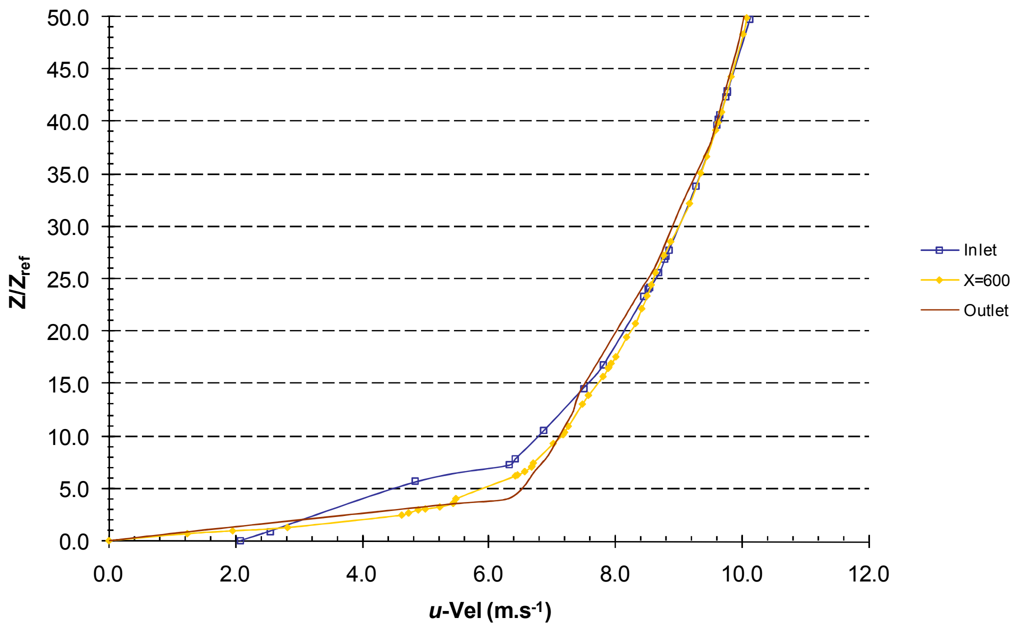


Figure A.2.1 Velocity profiles case two (θ= 260˚, B=12 m).

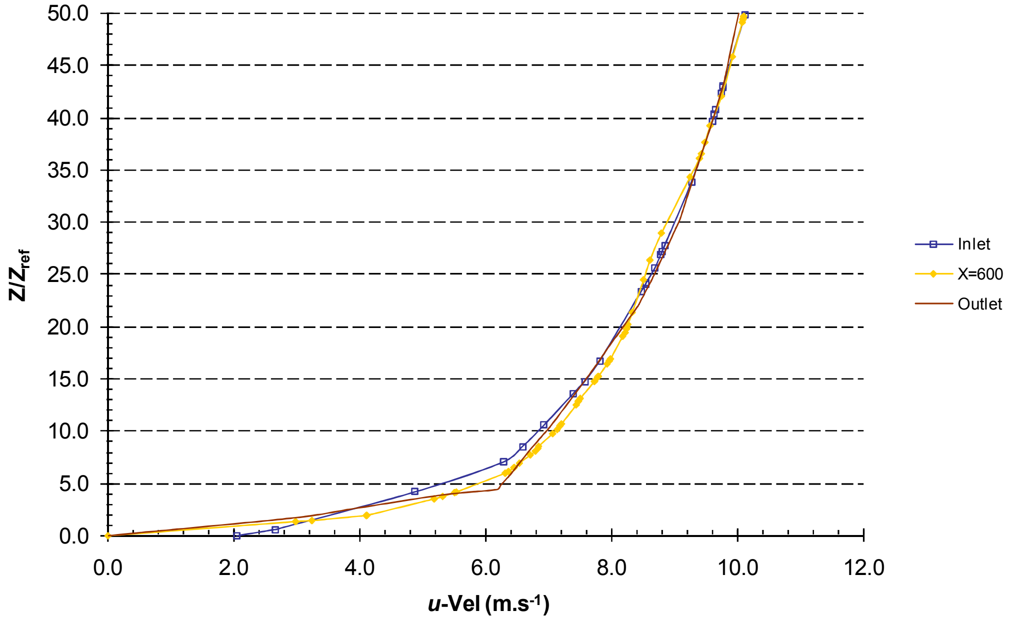


Figure A.2.2 Velocity profiles case three (θ= 260˚, B=15 m).

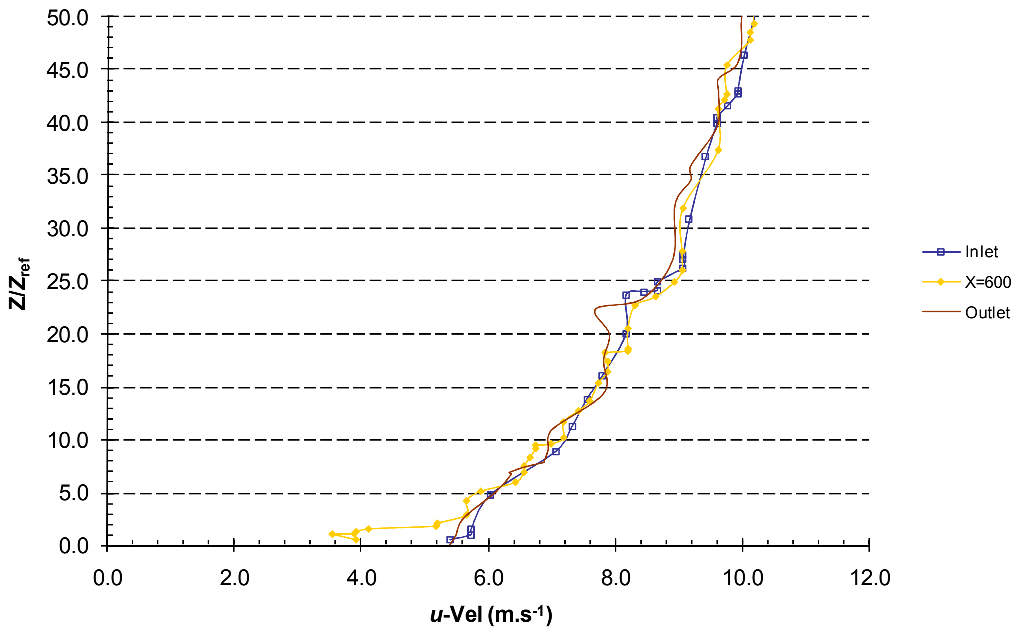


Figure A.2.3 Velocity profiles case four (θ= 236˚, B=9 m).

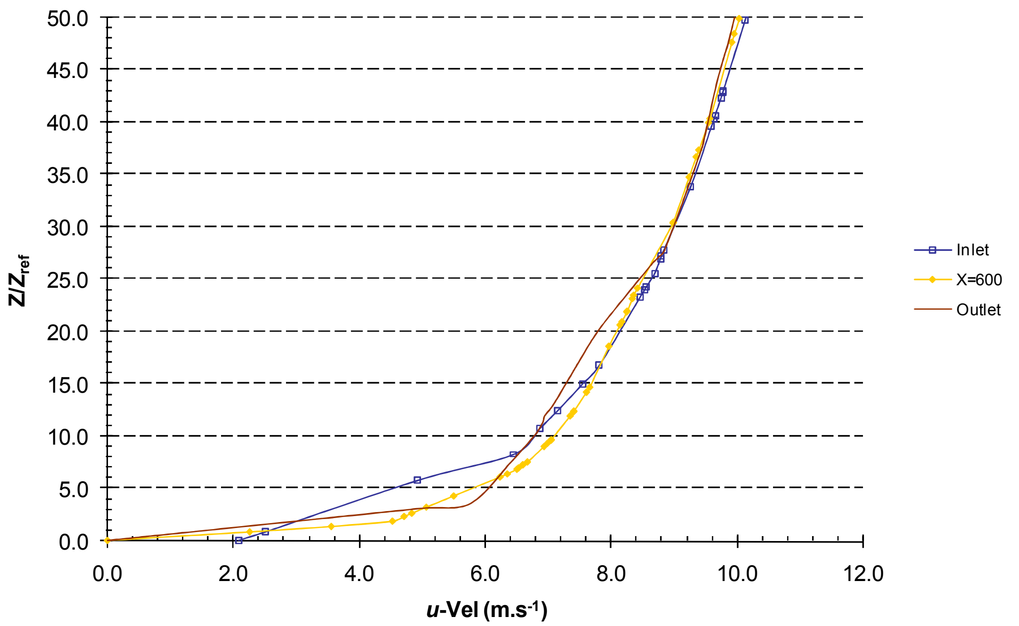


Figure A.2.4 Velocity profiles case five (θ= 236˚, B=12 m).

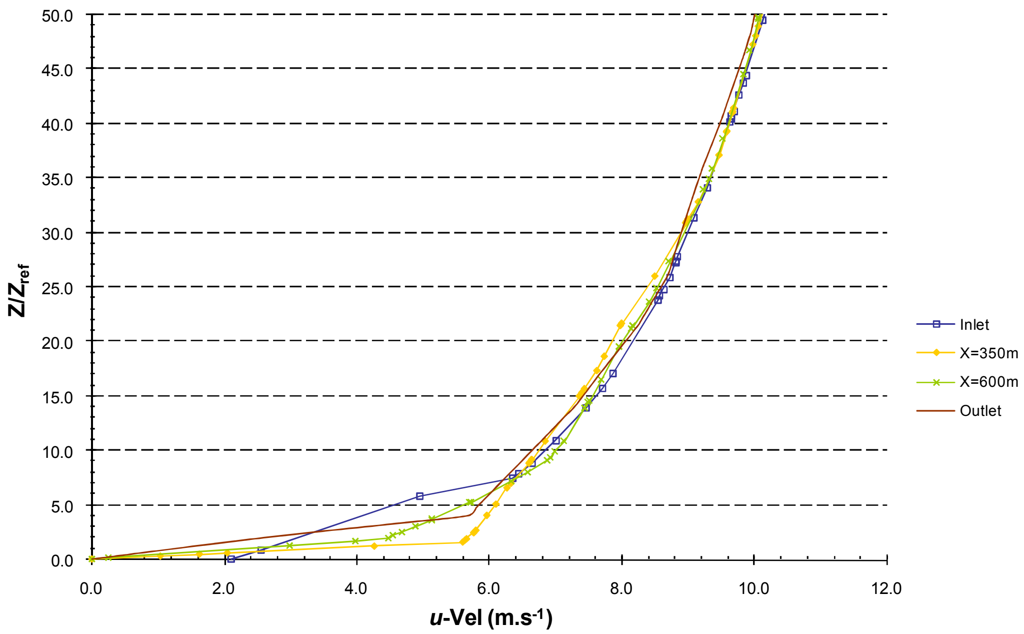


Figure A.2.5 Velocity profiles case six (θ= 236˚, B=15 m).

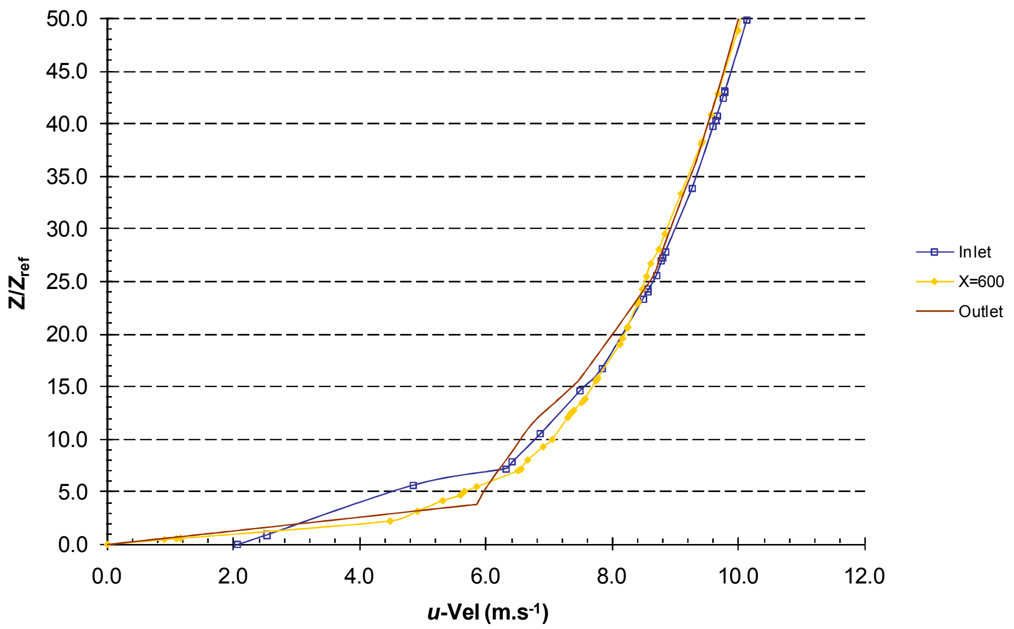


Figure A.2.6 Velocity profiles case seven (θ= 248˚, B=9 m).

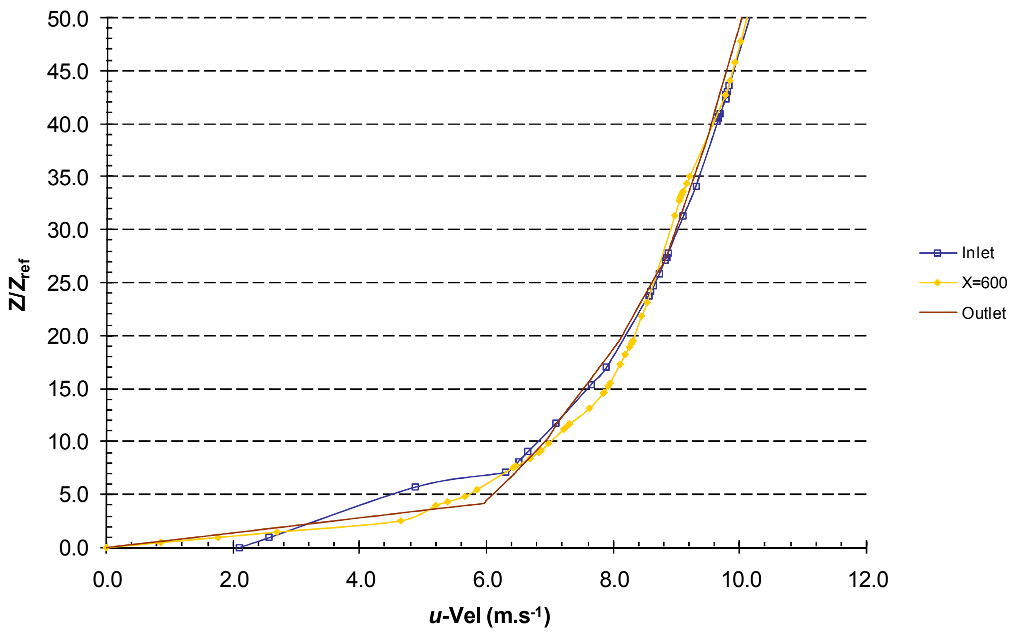


Figure A.2.7 Velocity profiles case eight (θ= 248˚, B=12 m).

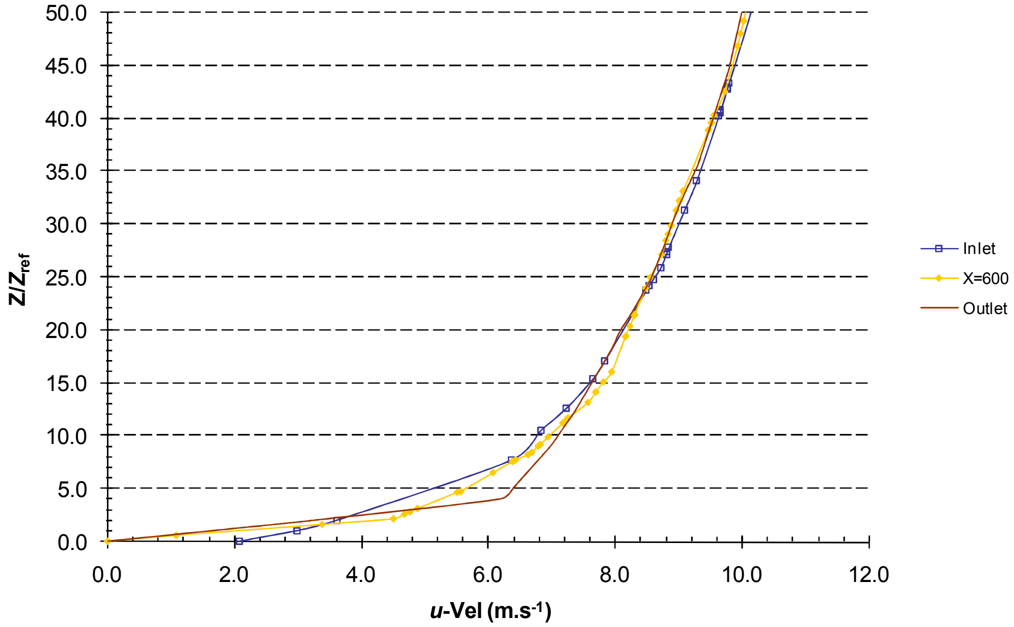


Figure A.2.8 Velocity profiles case nine (θ= 248˚, B=15 m).