UNIVERSITAT POLITÈCNICA DE VALÈNCIA

DEPARTMENT OF HYDRAULIC
ENGINEERING AND ENVIRONMENT

Ph.D Thesis

Computational Fluid Dynamics (CFD) Applied to Buildings Sustainable Design: Natural Ventilation Case Study

AUTHOR
MIGUEL MORA PÉREZ

DIRECTOR
Dr. PETRA AMPARO LÓPEZ JIMÉNEZ

Valencia, July 2017

AGRAÏMENTS

Aquesta tesis ha sigut possible gràcies a la confiança i treball que Amparo, la directora, ha tingut en mi des de pràcticament quan ens vam conèixer. Moltes gràcies Amparo per tot l'esforç, temps i paciència durant tots aquests anys. Aquesta tesis no haguera sigut possible sense la teua col·laboració incondicional per traure endavant els articles. Gràcies també a Gonzalo López per ajudar a millorar la qualitat del treball realitzat i a Nacho Guillén per la seua col·laboració en disposar dels mitjans materials necessaris en la investigació.

Moltes gràcies al meua meu germà Javier, qui m'ha ajudat a millorar la qualitat dels articles dedicant tot el temps que ha sigut necessari, sent sense cap dubte el referent a seguir.

Finalment, voldria dedicar aquest treball a la meua família, especialment a Paula i als meus pares, Mª Luisa i Miguel. Moltes gràcies de tot cor pel recolzament incondicional durant tots aquests anys.

ABSTRACT

Through the last decades, building designers should deal with reliable design strategies to take advantage of natural resources in order to increase energy efficiency in buildings, as well as to promote sustainable development and add value to the society.

This thesis proposes a reliable building design strategy to improve buildings energy efficiency by means of natural ventilation (NV) use. The strategy consists in evaluating the most suitable architectural solution in a particular case study taking into account environmental conditions and building surroundings in order to maximize NV use since the early building design stage. Computational fluid dynamics (CFD) techniques are used to conduct the research. This is a powerful design tool that permits buildings NV behaviour simulation prior to building construction.

Therefore, the aim of the thesis is to provide a real case study building in which the NV design strategy is applied to show a reliable example and support building design decisions since the design stage. This general objective could be subdivided into some more specific objectives, which are detailed as follow:

- Show CFD techniques potential to improve energy efficiency in buildings.
- Create and validate a computational fluid dynamics (CFD) model to simulate and evaluate buildings NV behaviour.
- Evaluate different façade opening configurations to improve NV effect in buildings.
- Analyse and quantify the energy saving potential of naturally ventilated façades.
- Evaluate different building sites in order to select the location that could maximize NV use in buildings.
- Analyse NV effect on indoor comfort conditions during the warm season.

- Validate the design strategy by conducting it through a real building design and construction (case study).

From the proposed objectives, a building design strategy is designed and conducted through a case study from the initial design stage until the final building construction and operation.

The design strategy is based on the use of a commercial numerical code that solves the fluid mechanic equations. The CFD software simulates the features that influence NV and predicts its behaviour in the different building configurations prior to building construction. This numerical technique allows, on the one hand, the visualization of air flow paths in buildings. On the other hand, many quantifiable parameters are calculated by the software. Through the analysis and comparison of those parameters, the best architectural solutions are chosen.

With regards to all possible architectural decisions, the research is focused on the façade configuration selection and the building location. First of all, the NV design strategy feasibility is analysed in a particular region: the Mediterranean Valencian Coastal area (Spain). The region is characterized by the uniform conditions of the prevailing wind during the warm season. Then, a validated CFD simulation is used to analyse qualitatively and quantitatively the building surrounding influence on wind paths through and around buildings. The objective is to compare different façade opening positions and select the alternative that takes more profit of the NV resources available. Additionally, a general quantification of the ventilated façade contribution to buildings energy efficiency is presented under the frame of the façade configuration selection. Secondly, two simulations are conducted to analyse two different building locations. The assessment of surrounding buildings influence on building NV behaviour is done through validated CFD models. Some parameters and visualizations are proposed to be used in the quantitative and qualitative assessment of each solution respectively. Then, the best location alternative with regards to NV performance is selected.

Finally, the research is concluded with the case study building full-scale construction. The indoor CFD simulation used from the beginning is then successfully validated. The NV building behaviour is also successfully verified. Additionally, contrasted performance indexes are used to evaluate indoor comfort conditions: draught risk (DR), predicted mean vote (PMV) and predicted percentage of dissatisfied people (PPD). The results show that comfort conditions can be reached more energy efficiently by means of NV use.

Afterwards, it is verified how the comfortable indoor environment conditions are ensured and optimized by the NV use. Although the design strategy is applied to a particular building design, the design strategy potential is that it could be applied to all buildings. Consequently, major potential energy savings could be achieved.

The present thesis includes six publications, four of those six are published in peer reviewed journals indexed in the "Journal Citation Reports" as detailed below:

- López-Jiménez, P.A., Mora-Pérez, M., La Ferla, G., Roset-Calzada, J. Increasing the value of buildings through environmental design. *COST Action TU1104 – Smart Energy Regions – Cost and Value. ISBN: 978-1-899895-22-9.* February 2016.
  International institutional publisher.
- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Patiño, G.; López-Jiménez, P.A. Natural Ventilation Building Design Approach in Mediterranean Regions—A Case Study at the Valencian Coastal Regional Scale (Spain). *Sustainability* 8, 855, 2016.
  JCR impact factor 1,789 (Q2).
- Mora-Pérez, M.; López-Patiño, G.; Bengochea-Escribano, M.A.; López-Jiménez, P.A. Cuantificación de la eficiencia de la fachada cerámica ventilada. *Boletín de la Sociedad Española de Cerámica y Vidrio*. Vol.50 nº2. 2011.
  JCR impact factor 0,432 (Q4).

- Mora-Pérez, M.; López-Patiño, G.; López-Jiménez, P.A. Quantification of Ventilated Façade Effect Due to Convection in Buildings. Buoyancy and Wind Driven Effect. *Researches and Applications in Mechanical Engineering*, Vol. 3 Iss. 1, March 2014.

  International publisher peer-reviewed.

- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation. *Advances in Engineering Software* 88, 73-82, 2015.

  JCR impact factor 1,673 (Q2).

- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. A CFD study for evaluating the effects of natural ventilation on indoor comfort conditions. *AIMS Environmental Science*, 4(2): 289-309, March 2017.

  Emerging Sources Citation Index (ESCI - Web of Science), extended JCR 2016.

RESUM

Durant les últimes dècades els agents involucrats en el disseny d'edificis utilitzen estratègies fiables de disseny que els permeten aprofitar els recursos naturals de l'entorn amb l'objectiu d'augmentar l'eficiència energètica dels edificis així com promoure el desenvolupament sostenible i generar valor afegit per la societat.

Aquesta tesi proposa una estratègia fiable de disseny d'edificis per a millorar la seva eficiència energètica mitjançant l'ús de la ventilació natural (NV per les sigles en anglès "*natural ventilation*"). L'estratègia consisteix a avaluar la solució arquitectònica més adequada tenint en compte les condicions ambientals i l'entorn dels edificis amb l'objectiu de maximitzar l'ús de la ventilació natural des de la fase inicial del seu disseny. En aquesta tesi s'aplica l'estratègia de disseny a un cas d'estudi real i particular. L'estratègia de disseny està basada en la utilització de tècniques de dinàmica de fluids computacionals (CFD per les sigles en anglès "*computational fluid dynamics*"). Les tècniques CFD són una potent eina de disseny que permet la simulació del comportament de la ventilació natural en els edificis abans de la seua construcció.

D'aquesta manera, l'objectiu de la tesi és proporcionar un cas d'estudi real en el qual l'estratègia de disseny de ventilació natural s'aplica per a proporcionar un exemple fiable i ajudar en la presa de decisions des de l'etapa inicial de disseny. Aquest objectiu general es subdivideix en una sèrie d'objectius específics que es detallen a continuació:

- Mostrar el potencial de les tècniques CFD per a millorar l'eficiència energètica en els edificis.
- Crear i validar un model de dinàmica de fluids computacional (CFD) per a simular i avaluar el comportament de la ventilació natural dels edificis.
- Avaluar el comportament de diferents configuracions d’obertures (finestres) de les façanes per a millorar l'efecte de la ventilació natural dels edificis.

- Analitzar i quantificar el potencial d’estalvi energètic de les façanes ventilades degut a la ventilació natural.
- Avaluar diferents ubicacions per seleccionar aquella en què es pugui maximitzar l'ús de la ventilació natural en els edificis.
- Analitzar l'efecte de la ventilació natural sobre les condicions interiors de confort durant la temporada càlida.
- Validar l'estratègia de disseny duent-la a terme a través d'un cas d'estudi real de disseny i construcció d'un edifici.

A partir dels objectius anteriors, es dissenya un edifici seguint l'estratègia de disseny d'edificis proposta a través d'un cas d'estudi que comprèn des de la fase inicial de disseny fins a la construcció i operació de l'edifici.

L'estratègia de disseny es basa en l'ús d'un codi numèric comercial que resol les equacions de la mecànica de fluids. El programari CFD simula les característiques que influeixen en la ventilació natural i prediu el seu comportament en els edificis abans de la seva construcció. Aquesta tècnica numèrica permet la visualització del flux d'aire en els edificis. A més, el programari permet calcular paràmetres que són analitzats i comparats posteriorment per triar la solució arquitectònica que supose un millor comportament de la ventilació natural.

Pel que fa a totes les decisions arquitectòniques possibles, la investigació es centra en la selecció de la ubicació de l'edifici i de la configuració de les obertures de la façana. En primer lloc, s'analitza la viabilitat de l'estratègia de disseny en una regió determinada: la zona costanera Mediterrània de la Comunitat Valenciana. La regió es caracteritza per les condicions uniformes del vent predominant durant l'estació càlida. A continuació, s'utilitza una simulació de CFD validada per analitzar qualitativament i quantitativament la influència dels edificis circumdants en els fluxos del vent a través i al voltant dels edificis circumdants. L'objectiu és comparar diferents posicions dels buits de la façana per seleccionar l'alternativa que millor aprofite els recursos de ventilació natural disponibles. A més, en el marc de la selecció de la configuració de la façana es presenta una quantificació general de la contribució de la façana ventilada a l'eficiència energètica dels edificis. En segon lloc, es realitzen dues simulacions per analitzar dues ubicacions diferents de l'edifici cas d'estudi. L'avaluació de la influència dels edificis circumdants en el comportament de la ventilació natural de l'edifici cas d'estudi es realitza mitjançant la utilització de models CFD validats. Es proposen diferents paràmetres i visualitzacions per a l'avaluació quantitativa i qualitativa de cada solució. A continuació es selecciona la millor ubicació pel que fa al comportament de la ventilació natural a l'edifici cas d'estudi.

Finalment, la investigació conclou amb la construcció a escala real de l'edifici cas d'estudi. Es valida amb èxit la simulació CFD de l'interior de l'edifici utilitzada des de l'etapa de disseny. També es verifica amb èxit el comportament de la ventilació natural de l'edifici. A més, s'analitzen les condicions de confort interiors mitjançant l'avaluació dels següents índexs: risc de corrents d'aire (DR per les sigles en anglès "*draught risk*"), mitjana de vots previstos (PMV per les sigles en anglès "*predicted mean vote*") i el percentatge previst de persones insatisfetes (PPD per les sigles en anglès "*predicted percentage of dissatisfied people*"). Els resultats mostren que l'ús de la ventilació natural permet assolir, de manera més energèticament eficient, les condicions de confort.

A continuació es verifica com es garanteixen i optimitzen les condicions de confort interiors mitjançant l'ús de la ventilació natural. Encara que l'estratègia de disseny s'aplica al disseny d'un edifici en particular, el potencial de l'estratègia de disseny és molt més gran ja que aquest podria aplicar-se a tots els edificis. En conseqüència els estalvis potencials d'energia són més grans.

La present tesi inclou sis publicacions, on quatre d'aquestes es publiquen en revistes revisades per parells i indexades en el "Journal Citation Reports" com es presenta a continuació:

- López-Jiménez, P.A., Mora-Pérez, M., La Ferla, G., Roset-Calzada, J. Increasing the value of buildings through environmental design. *COST Action TU1104 – Smart Energy Regions – Cost and Value. ISBN: 978-1-899895-22-9.* February 2016.

  International institutional publisher.
- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Patiño, G.; López-Jiménez, P.A. Natural Ventilation Building Design Approach in Mediterranean Regions—A Case Study at the Valencian Coastal Regional Scale (Spain). *Sustainability* 8, 855, 2016.

  JCR impact factor 1,789 (Q2).
- Mora-Pérez, M.; López-Patiño, G.; Bengochea-Escribano, M.A.; López-Jiménez, P.A. Cuantificación de la eficiencia de la fachada cerámica ventilada. *Boletín de la Sociedad Española de Cerámica y Vidrio*. Vol.50 nº2. 2011.

  JCR impact factor 0,432 (Q4).
- Mora-Pérez, M.; López-Patiño, G.; López-Jiménez, P.A. Quantification of Ventilated Façade Effect Due to Convection in Buildings. Buoyancy and Wind Driven Effect. *Researches and Applications in Mechanical Engineering*, Vol. 3 Iss. 1, March 2014.

  International publisher peer-reviewed.
- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation. *Advances in Engineering Software* 88, 73-82, 2015.

  JCR impact factor 1,673 (Q2).
- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. A CFD study for evaluating the effects of natural ventilation on indoor comfort conditions. *AIMS Environmental Science*, 4(2): 289-309, March 2017.

  Emerging Sources Citation Index (ESCI - Web of Science), extended JCR 2016.

RESUMEN

Durante las últimas décadas los agentes involucrados en el diseño de edificios deben de utilizar estrategias fiables de diseño que les permitan aprovechar los recursos naturales del entorno con el objetivo de aumentar la eficiencia energética de los edificios así como promover el desarrollo sostenible y generar valor añadido para la sociedad.

Esta tesis propone una estrategia de diseño fiable de edificios para mejorar su eficiencia energética mediante el uso de la ventilación natural (NV por sus siglas en inglés "natural ventilation"). La estrategia consiste en evaluar la solución arquitectónica más adecuada teniendo en cuenta las condiciones ambientales y el entorno de los edificios con el objetivo de maximizar el uso de la ventilación natural desde la fase inicial de su diseño. En esta tesis se aplica la estrategia de diseño a un caso de estudio real y particular. La estrategia de diseño está basada en la utilización de técnicas de dinámica de fluidos computacionales (CFD por sus siglas en inglés "computational fluid dynamics"). Las técnicas CFD son una potente herramienta de diseño que permite la simulación del comportamiento de la ventilación natural en los edificios antes de su construcción.

De este modo, el objetivo de la tesis es proporcionar un caso de estudio real en el que la estrategia de diseño de ventilación natural se aplica para proporcionar un ejemplo fiable y ayudar en la toma de decisiones desde la etapa inicial de diseño. Este objetivo general se subdivide en una serie de objetivos específicos que se detallan a continuación:

- Mostrar el potencial de las técnicas CFD para mejorar la eficiencia energética en los edificios.
- Crear y validar un modelo de dinámica de fluidos computacional (CFD) para simular y evaluar el comportamiento de la ventilación natural de los edificios.

- Evaluar el comportamiento de diferentes configuraciones de huecos (ventanas) de las fachadas para mejorar el efecto de la ventilación natural de los edificios.
- Analizar y cuantificar el potencial ahorro energético de las fachadas ventiladas de manera natural.
- Evaluar diferentes ubicaciones para seleccionar aquella en la que se pueda maximizar el uso de la ventilación natural en los edificios.
- Analizar el efecto de la ventilación natural sobre las condiciones interiores de confort durante la temporada cálida.
- Validar la estrategia de diseño llevándola a cabo a través de un caso de estudio real de diseño y construcción de un edificio.

A partir de estos objetivos, se diseña un edificio siguiendo la estrategia de diseño propuesta a través de un caso de estudio que comprende desde la fase inicial de diseño hasta la construcción y operación del edificio.

La estrategia de diseño se basa en el uso de un código numérico comercial que resuelve las ecuaciones de la mecánica de fluidos. El software CFD simula las características que influyen en la ventilación natural y predice su comportamiento en los edificios antes de su construcción. Esta técnica numérica permite la visualización del flujo de aire en los edificios. Además, el software permite calcular parámetros que son analizados y comparados posteriormente para elegir la solución arquitectónica que suponga un mejor comportamiento de la ventilación natural.

Con respecto a todas las decisiones arquitectónicas posibles, la investigación se centra en la selección de la ubicación del edificio y de la configuración de los huecos de su fachada. En primer lugar, se analiza la viabilidad de la estrategia de diseño en una región determinada: la zona costera Mediterránea de la Comunidad Valenciana. La región se caracteriza por las condiciones uniformes del viento predominante durante la estación cálida. A continuación, se utiliza una simulación de CFD validada para analizar cualitativamente y cuantitativamente la influencia de los edificios circundantes en los flujos del viento a través y alrededor de los edificios circundantes. El objetivo es comparar distintas posiciones de los huecos de la fachada para seleccionar la alternativa que mejor aproveche los recursos de ventilación natural disponibles. Además, se presenta en el marco de la selección de la configuración de la fachada una cuantificación general de la contribución de la fachada ventilada a la eficiencia energética de los edificios. En segundo lugar, se realizan dos simulaciones para analizar dos ubicaciones diferentes del edificio caso de estudio. La evaluación de la influencia de los edificios circundantes en el comportamiento de la ventilación natural del edificio caso de estudio se realiza mediante la utilización de modelos CFD validados. Se proponen distintos parámetros y visualizaciones para la evaluación cuantitativa y cualitativa de cada solución. A continuación se selecciona la mejor ubicación con respecto al comportamiento de la ventilación natural en el edificio caso de estudio.

Finalmente, la investigación concluye con la construcción a escala real del edificio caso de estudio. Se valida con éxito la simulación CFD del interior del edificio utilizada desde la etapa de diseño. También se verifica con éxito el comportamiento de la ventilación natural del edificio. Además, se analizan las condiciones de confort interiores mediante la evaluación de los siguientes índices: riesgo de corrientes de aire (DR por sus siglas en inglés “*draught risk*”), voto promedio previsto (PMV por sus siglas en inglés “*predicted mean vote*”) y el porcentaje previsto de personas insatisfechas (PPD por sus siglas en inglés “*predicted percentage of dissatisfied people*”). Los resultados muestran que el uso de la ventilación natural permite alcanzar, de manera más energéticamente eficiente, las condiciones de confort.

A continuación se verifica cómo se garantizan y optimizan las condiciones de confort interiores mediante el uso de la ventilación natural. Aunque la estrategia de diseño se aplica a un diseño de un edificio en particular, el potencial de la estrategia de diseño es mucho mayor ya que ésta podría aplicarse a todos los edificios. En consecuencia los ahorros potenciales de energía son mayores.

La presente tesis incluye seis publicaciones de entre las cuales cuatro de ellas se publican en revistas revisadas por pares e indexadas en el "Journal Citation Reports" como se presenta a continuación:

- López-Jiménez, P.A., Mora-Pérez, M., La Ferla, G., Roset-Calzada, J. Increasing the value of buildings through environmental design. *COST Action TU1104 – Smart Energy Regions – Cost and Value. ISBN: 978-1-899895-22-9.* February 2016.
  International institutional publisher.
- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Patiño, G.; López-Jiménez, P.A. Natural Ventilation Building Design Approach in Mediterranean Regions—A Case Study at the Valencian Coastal Regional Scale (Spain). *Sustainability* 8, 855, 2016.
  JCR impact factor 1,789 (Q2).
- Mora-Pérez, M.; López-Patiño, G.; Bengochea-Escribano, M.A.; López-Jiménez, P.A. Cuantificación de la eficiencia de la fachada cerámica ventilada. *Boletín de la Sociedad Española de Cerámica y Vidrio*. Vol.50 nº2. 2011.
  JCR impact factor 0,432 (Q4).
- Mora-Pérez, M.; López-Patiño, G.; López-Jiménez, P.A. Quantification of Ventilated Façade Effect Due to Convection in Buildings. Buoyancy and Wind Driven Effect. *Researches and Applications in Mechanical Engineering*, Vol. 3 Iss. 1, March 2014.
  International publisher peer-reviewed.
- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation. *Advances in Engineering Software* 88, 73-82, 2015.
  JCR impact factor 1,673 (Q2).

- Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. A CFD study for evaluating the effects of natural ventilation on indoor comfort conditions. *AIMS Environmental Science*, 4(2): 289-309, March 2017.

  Emerging Sources Citation Index (ESCI - Web of Science), extended JCR 2016.

TABLE OF CONTENTS

AGRAÏMENTS III

ABSTRACT V

RESUM IX

RESUMEN XIII

TABLE OF CONTENTS 1

CHAPTER 1: INTRODUCTION 3

  1.1. Case study building 5

CHAPTER 2: OBJECTIVES 7

CHAPTER 3: RESULTS AND DISCUSSION 9

  3.1. Introduction to sustainable design 10

  3.2. Introduction to the Valencian Community region 12

  3.3. The CFD model 14

  3.4. Façade opening position analysis by means of CFD 16

  3.5. Ventilated façade performance assessment by means of CFD 19

  3.6. Building location analysis by means of CFD 24

  3.7. Indoor comfort conditions analysis by means of CFD 26

CHAPTER 4: CONCLUSIONS AND CONTRIBUTIONS 29

CHAPTER 5: REFERENCES 35

ANNEXES 37

CHAPTER 1: INTRODUCTION

In the recent years our society is getting concerned with the greenhouse effect problem and the importance of fulfilling the present needs without compromising future generations to meet their own needs [1]. Sustainable development has been accepted and used to show implicitly the non-sustainable development of the current society, especially in the energy field. In this sense, the need for sustainability is clear as soon as there is a risk in the energy shortage. Consequently, low carbon technologies are being developed for reducing the greenhouse effect and saving energy and natural resources with the ultimate aim to allow future generations to fulfil their necessities.

There are many fields from which the increase of greenhouse effect could be slowed down. The EU is working to reduce greenhouse gases emissions by 20% (from 1990 levels) by 2020. One of the pillars leading the European Union to reduce the greenhouse effect is sustainability in the building environment, which means near 35% of the total European CO_2 emissions [2]. Therefore, the increase in energy efficiency in the building sector could represent an important percentage to reduce CO_2 emissions.

Building energy efficiency is becoming more popular because of the increasingly higher prices of the energy regardless of their origin and the increased awareness of the environmental impact of energy use. Consequently, inexhaustible energy sources and passive methods are starting to play a significant role for sustainable building development. In this respect, natural ventilation (NV) use is presented as one of the best passive mechanisms to improve buildings energy efficiency as well as to provide acceptable indoor environmental quality and acceptable comfort conditions. A strong focus on NV behaviour since the design stage could save many energy resources in the buildings operation with almost negligible initial cost increase. This feature combined with the complete building agent chain awareness, i.e. designers, owners, final users, etc. will lead into quick recovery of the possible initial overinvestment and will generate earnings besides many other social benefits.

Natural ventilation had been used for centuries until the air conditioning became more widespread in the last decades. The technique was generally abandoned due to the difficulties to manage its behaviour in a controlled manner. However, the energy efficiency benefits, the new computational fluid dynamic techniques and the integrated building management systems have streamlined the NV use again.

Natural ventilation relies on buoyancy and/or wind pressure-driven forces that move air through buildings [3]. Thus, the fluid mechanics plays an important role in the study of air movement energetic implications. NV behaviour in buildings depends critically on the building location and the façade configuration. Firstly, the analysis of the prevailing wind flow of the region as well as surrounding environment influence on local wind direction changes are particularly important. Moreover, wind could be modelled by CFD techniques in order to predict its behaviour. Secondly, façade configuration will be responsible for driving the wind into the building. Consequently, the most suitable architectural solution should be selected depending on how it works according to the influence of the environmental surrounding conditions.

There are many techniques for predicting NV behaviour of buildings [4-5]. CFD techniques have been widely used during the lasts years [6-8], although they have not been almost handled during the initial design phase. The present research uses a commercial numerical software, STAR-CCM+, that solves the fluid mechanic equations in a three dimensional domain. The software allows wind and air flow visualization and the calculation of many parameters (i.e. velocity, pressure, temperature, numerical comfort indexes, etc.) that are analysed in order to compare the different alternatives. Then, CFD techniques enable analysing all design architectural solutions by simulation techniques and not by real trial-and-error methods that require higher economic resources and time. Thereby, the initial cost increase is almost negligible in comparison with the future potential energy savings as well as the benefits provided to society.

The present design approach is conducted through a case study building. With the exception of the research conducted to assess generally the particular ventilated façade architectural solution, the case study building NV behaviour is simulated by means of a CFD model. The details of the case study building are presented below.

1.1. Case study building

The design strategy is conducted through a case study building design and construction within the frame "*E3 echo efficient building design*" research project at "Universitat Politècnica de València" in Valencia (Spain). The project consists of designing and constructing an energy-efficient building with the aim to encourage designers and society in general towards sustainable development. Thereby, every architectural component has been designed to accomplish a bioclimatic requirement.

The present thesis is focused on the optimization of the building NV performance attending to the location and the façade configuration, taking into account the local environmental conditions. The analysis done comprises different stages from the early beginning design stage until the building construction and operation. Figure 1 shows an outdoor front view of the case study building. Additional outdoor and indoor views are included in the corresponding annexes.

Figure 1. Case study constructed building

The initial building design is based on a typical 70 m^2 modular regional familiar dwelling. The building is 4 m height and 5 m width. The indoor distribution is made with two rooms connected by a short passageway where the bathroom is located. One of the room has a door to the outside (2.5 m x 3.5 m) and the other has a window (1.5 m x 3.7 m), both in the main façade. Additionally, both rooms have two narrow windows (0.5 m x 2 m and 0.5 m x 1 m) located in the upper part of the back building face. Figure 2 shows a plant view drawing of the case study building simulated and figure 3 the 3D view of the case study building. Moreover, some passive elements have been installed in order to avoid solar gains through the main building openings.

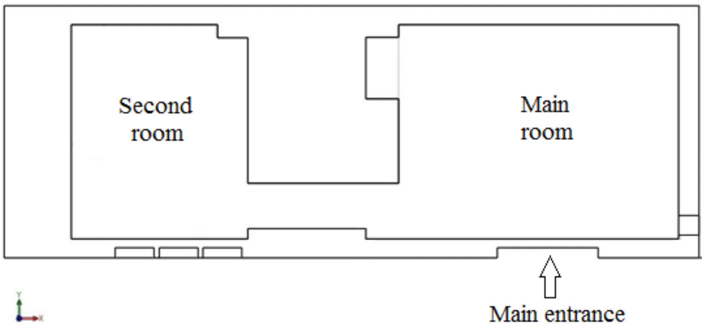


Figure 2. Plant view of the simulated case study building

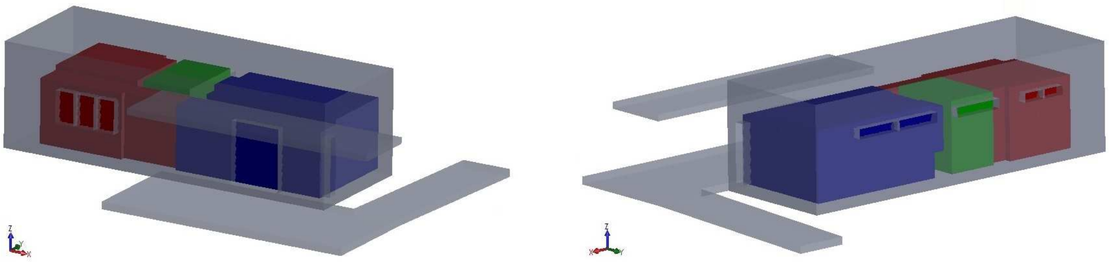


Figure 3. Case study building 3D view (front (left) and back (right) view)

Finally, the building is located in an urban plain terrain 1 km from the Mediterranean Sea coast (39°28'41.8"N 0°20'10.8"W). The environmental conditions of this location are presented in Chapter 3. Besides, the Chapter 3 presents the thesis discussion and results. Chapter 2 presents the thesis objectives and the conclusions are depicted in Chapter 4. Chapter 5 describes the thesis references and finally all manuscripts published within the thesis scope are included in the six annexes.

CHAPTER 2: OBJECTIVES

This thesis proposes a building design strategy to improve buildings energy efficiency by means of NV use. Furthermore, the strategy is conducted in a real case building. The strategy consists in evaluating the most suitable architectural solution taking into account environmental conditions and building surroundings in order to maximize NV use since the early building design stage.

The different architectural alternatives evaluation is done through the analysis of computational fluid dynamics (CFD) simulations. The fluid mechanics analysis to improve kinetic and thermal energy extraction from natural flows is of special interest for the energy efficiency scope nowadays. The present thesis also aims to contribute to the achievement of this general purpose.

The previous general objectives could be subdivided into some more specific objectives, which are detailed as follow:

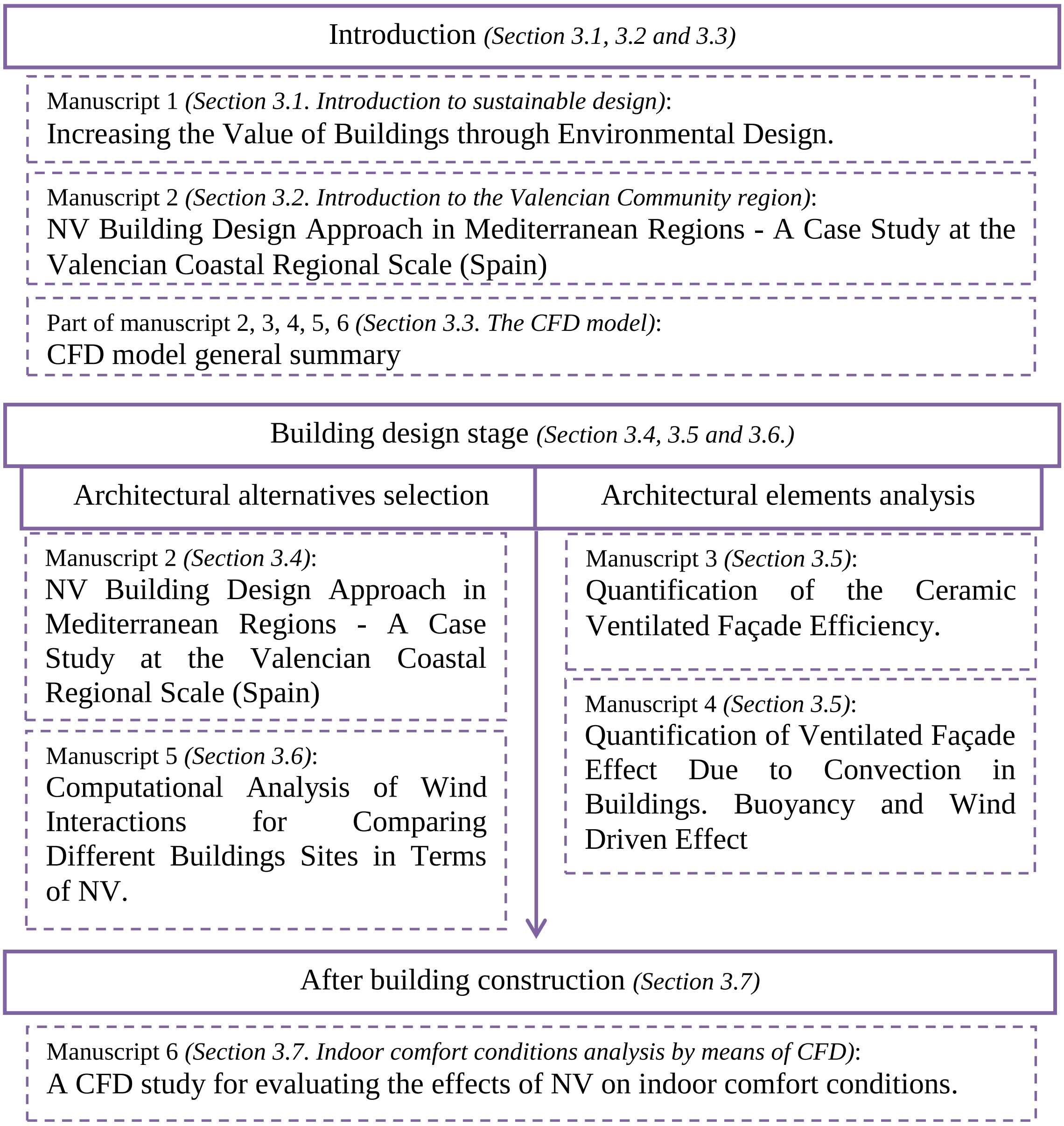
- Show CFD techniques potential to improve energy efficiency in buildings.
- Create and validate CFD models to simulate and analyse buildings NV behaviour.
- Evaluate different façade opening configurations to improve NV effect in buildings.
- Analyse and quantify the energy saving potential of naturally ventilated façades.
- Evaluate different building sites in order to select the location that could maximize NV use in buildings.
- Analyse NV effect on indoor comfort conditions during the warm season.
- Validate the design strategy by conducting it through a real building design and construction (case study).

These specific objectives contribute to the addition to the construction sector of a reliable NV design approach that assists building designers on the sustainable building design since the initial design stage. The novelty of the design approach lies in the fact that NV use is reinforced by the use of CFD techniques since the initial design stage in a real case study building that will be constructed afterwards to validate the design approach and close the design loop. Then, building designers know that the NV measures that had been taken at the initial design stage had the expected NV effect in the building finally. The thesis is an attempt to provide structure to the application of advanced design tools such as CFD to an area (NV) that is traditionally approached with empirical or integral rules [4-7]. The analysis and comparison of this approach together with the process to create and validate the different CFD simulations is summarized in Chapter 3.

Chapter 3 has the references to the annexes in which the particular features of the design approach are depicted in detail. The successful application of the design approach to the case study is the proof that should encourage building designers to use it. Additionally, some features regarding NV performance in ventilated façades are analysed by means of CFD techniques.

CHAPTER 3: RESULTS AND DISCUSSION

The discussion and results of the present research are based on six manuscripts that have been published within the scope of the thesis. Figure 4 summarizes the chapter sections and the manuscripts on which are based on:



Introduction *(Section 3.1, 3.2 and 3.3)*

Manuscript 1 *(Section 3.1. Introduction to sustainable design)*:
Increasing the Value of Buildings through Environmental Design.

Manuscript 2 *(Section 3.2. Introduction to the Valencian Community region)*:
NV Building Design Approach in Mediterranean Regions - A Case Study at the Valencian Coastal Regional Scale (Spain)

Part of manuscript 2, 3, 4, 5, 6 *(Section 3.3. The CFD model)*:
CFD model general summary

Building design stage *(Section 3.4, 3.5 and 3.6.)*

Architectural alternatives selection

Manuscript 2 *(Section 3.4)*:
NV Building Design Approach in Mediterranean Regions - A Case Study at the Valencian Coastal Regional Scale (Spain)

Manuscript 5 *(Section 3.6)*:
Computational Analysis of Wind Interactions for Comparing Different Buildings Sites in Terms of NV.

Architectural elements analysis

Manuscript 3 *(Section 3.5)*:
Quantification of the Ceramic Ventilated Façade Efficiency.

Manuscript 4 *(Section 3.5)*:
Quantification of Ventilated Façade Effect Due to Convection in Buildings. Buoyancy and Wind Driven Effect

After building construction *(Section 3.7)*

Manuscript 6 *(Section 3.7. Indoor comfort conditions analysis by means of CFD)*:
A CFD study for evaluating the effects of NV on indoor comfort conditions.

Figure 4. Research structure

The publications cover all the objectives of the thesis. The particular objectives and results of each paper are summarized in the present chapter. General CFD model features are also outlined in this chapter. The complete contents of each paper are detailed in the annexes.

The here applied strategy aims to evaluate the most suitable architectural solutions taking into account the environmental conditions and the surroundings, in order to maximize NV use since the early building design stage. Therefore, the first stage consists in analysing the environmental conditions of the region in which the case study will be built.

Secondly, some architectural solutions are assessed. On the one hand, a façade opening position is selected considering the most relevant features that influence NV behaviour through buildings. On the other hand, a ventilated façade architectural solution influence on the energy savings in the building is analysed.

Many times the location can be selected by the designers. In these cases, it is important to assess the NV potential of each area. Thereby, next stage faces the selection of the most suitable location in order to improve the building NV behaviour.

Finally, the case study building is constructed and a validation of the model is done in the real building. Besides, the NV effect on indoor comfort conditions is assessed as well as the NV performance to improve the energy behavior of the building.

3.1. Introduction to sustainable design

This section is based on the manuscript:

> López-Jiménez, P.A., Mora-Pérez, M., La Ferla, G., Roset-Calzada, J. Increasing the value of buildings through environmental design. *COST Action TU1104 –*

Smart Energy Regions – Cost and Value. ISBN: 978-1-899895-22-9. February 2016.

The chapter of the book has been published by Cardiff University in the framework of the Cost Action TU1104 Smart Energy Regions project, financed by the U.E.

Paper Summary

The awareness for sustainable development [9] in the built environment has been increased during the last decades. The aim of building designers is to achieve comfort conditions in buildings with the least possible resources taking into account local climatic conditions. These possible resources include since the initial design stage, the construction and the building operation. Through the resources use analysis, the aim is to discuss how the cost and final building value is positively affected by environmental design. Actually, most of the already constructed buildings are made with poor quality materials and components what lead into high operational energy costs. Sustainable design might have higher initial material and design costs although energy use during the building life-time should be lower. Consequently, the payback might be shorter, what means that in the end sustainable design will add value to the complete life-cycle of buildings.

Unfortunately it is still thought that sustainable design requires higher initial costs mainly in the design and construction stage that may increase too much the payback period respect a non-sustainable design. Nevertheless, there are other economic and social benefits that will increase the value of the building such as the increase in comfort that may lead into high workers productivity, longer building lifetime, reduced complaints, social recognition, etc. that will make sustainable design profitable. Moreover, sustainable buildings initial costs should not be greater than traditional ones if the whole building agent chain (i.e. designers, constructors, consultants, owners, users,…) is involved since the beginning. Then, this team should work to do not increase the initial value of buildings as well as to ensure that final building users make profit of initial innovative solutions during the operational lifetime of the building. These solutions entail since initial design cost reduction until midterm energy savings, maintenance cost reduction, water savings and indirect social benefits, among others.

In conclusion, if the complete building agent chain is fully aware of the sustainable building approach, the probable initial overinvestment due to sustainable approach will be profitable because it may generate not only energy savings but also many other social benefits.

Consequently, sustainable development should be promoted, especially in regions with high sustainable resources use potential such as the Valencian Community region in Spain, among others.

3.2. Introduction to the Valencian Community region

This section is based on the manuscript:

> *Mora-Pérez, M.; Guillen-Guillamón, I.; López-Patiño, G.; López-Jiménez, P.A. Natural Ventilation Building Design Approach in Mediterranean Regions—A Case Study at the Valencian Coastal Regional Scale (Spain). Sustainability 8, 855, 2016.*

The journal JCR impact factor is 1,789 in 2016 in the environmental sciences scope. The journal has the position 119 within 229 indexed journals. It belongs to the second quarter (Q2).

Paper Summary

First strategy stage consists in analysing the environmental characteristics of the region in which the case study is located. In this particular case, the Valencian Community Mediterranean coast climate conditions are analysed. The objective is to determine the season in which NV use could be of major interest and the wind conditions that should be analysed first in order to cover the most probable conditions.

The Valencian Community is placed in the east of Spain and it is spread along the Mediterranean Sea coast. The coastal area has a plain contour. Moreover, the Mediterranean Sea effect favors averaged environmental temperature conditions during the whole year. Particularly, temperature conditions are near 20ºC during spring and autumn. Then, temperatures are little bit higher than 25ºC during the warm season and slightly lower than 15ºC during the cold season [10]. These conditions makes the Valencian Community coastal region particularly suitable to apply sustainable building design approaches. More details are provided in annex 2.

The analysis of the sun-heating conditions is also important. In this case, the Valencian Community is an exceptionally sun-heated region (average of 8.3 h/day), especially during the warm season that lasts more than 7 months/year [10]. Consequently, heating sun gains in the building decrease buildings energy efficiency performance during the warm season. Thereby, the research is focus on the warm season.

The wind conditions are of special interest for NV behaviour analysis. In this particular case, there is clearly a prevailing wind condition, which is conditioned by the sea position (SW-NE direction) [10]. In any case, a local wind rose should be used to know the prevailing wind direction and frequency. In this case, the prevailing wind direction is east-south-east (ESE) in Valencia during spring, summer and autumn. Then, this wind condition should be analysed first in order to cover the most probable conditions. Further research should take into account all of the other wind conditions.

In conclusion, the warm season conditions should be firstly analysed in the particular case of the Valencian Community Mediterranean coast region to improve buildings energy efficiency by increasing NV use.

3.3. The CFD model

It has been introduced that innovative solutions and techniques should be used during the sustainable building design. In this case, CFD techniques are presented as a powerful tool to assist designers in the sustainable building design.

A general objective of the thesis is the CFD model creation, validation and analysis applied to NV in buildings. Particularly, 3 dimension models are used to simulate the wind behaviour in the area of interest. The areas of interest are further described in each annex of the present thesis.

Generally, the finite volume method (FVM) is used to solve the mass (1) and momentum (2) fluid dynamic equations in steady-state regime [11].

$$\frac{\partial \rho}{\partial t} + \nabla \rho \vec{v} = S_m \tag{1}$$

where ρ stands for the fluid density, t is the time, $\vec{v}$ is the velocity and S_m represents the mass contained in the control volume. The Reynolds Average Navier-Stokes (RANS) equations are solved in a 3D domain [8]. Navier-Stokes momentum equation is considered as (2). The simulations use the segregated model. Besides, turbulence is represented by the standard k-ε model [12]. The particular features of the turbulence model selection are described in the corresponding annexes.

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \rho (\vec{v}\vec{v}) = -\nabla p + \nabla \bar{\tau} + \rho \vec{g} + \vec{F} \tag{2}$$

where p stands for the static pressure, $\bar{\tau}$ the stress tensor defined by eq (3) and $\vec{g}$ and $\vec{F}$ represent the gravitational and outer forces respectively. μ is the eddy viscosity and I is the unit tensor.

$$\bar{\tau} = \mu \times \left[\left(\nabla \vec{v} + \nabla \vec{v}^{T} \right) - \frac{2}{3} \times \nabla \vec{v} I \right] \tag{3}$$

It is required the buoyancy effect inclusion in the momentum equations what is done through the gravity model selection. In order to avoid computational errors, an only mesh is defined for the complete volume represented. Thereby, outdoor and indoor volumes are computed together [13]. The numerical code solves the equations until the residuals are lower than 10^{-3} and then the solution is considered converged and the results can be analysed.

The FVM requires domain discretisation that it is made by a grid. The numerical results depend on grid quality so it should be checked the numerical results independence with reference to grid size. The grid size should be accurate enough for catching the important physical features in the area of concern. Consequently, the grid is tighter in those areas, especially in the indoor volumes. By contrast, the exterior volume does not require that accuracy [13]. The method consists in creating three models with three different mesh sizes. The models are solved by the numerical code and the results are compared to evaluate the solution grid independence. Quantitative and qualitative analyses are done to select the mesh that best achieves a balance between results accuracy and computational time. The particular results of each model and each mesh are detailed in the annexes.

Boundary conditions should be defined in each model. The annexes present the boundary conditions definition in detail. Within them, the most relevant is the definition of the air velocity inlet condition. The wind velocity boundary is defined using the Justus and Mikhail equation that defines the vertical velocity profile [14]. It is important to mention that wind fluctuating approach is not considered in the present research. Nevertheless further research should be done to take into account this feature.

Numerical results reliability should be ensured by the CFD model validation. In the first stages is not possible to validate the indoor models because the building is still not built.

Nevertheless the outdoor model can be validated. The CFD strategy aims to validate the outdoor computational model with full-scale wind measurements when the building is still not built. The validation is done by comparing the CFD results respect wind measurements [15]. The measurement points should be selected attending to the wind distribution that should be without the building, as the measurements are taken when the building is still not built. Once the building is constructed, the indoor CFD model can be validated as well as the NV behaviour and its influence in comfort conditions.

In general, different CFD simulations with different CFD models and meshes are performed in this research process. The details of each CFD models, meshes and simulations are indicated in each publication included in the annexes.

3.4. Façade opening position analysis by means of CFD

This section is based on the manuscript:

> *Mora-Pérez, M.; Guillen-Guillamón, I.; López-Patiño, G.; López-Jiménez, P.A. Natural Ventilation Building Design Approach in Mediterranean Regions—A Case Study at the Valencian Coastal Regional Scale (Spain). Sustainability 8, 855, 2016.*

The journal JCR impact factor is 1,343 in 2015 in the environmental sciences scope. The journal has the position 146 within 224 indexed journals. It belongs to the third quarter (Q3).

Paper Summary

The objective at this stage of the research is to show the design strategy followed to determine the most suitable façade opening configuration in a fixed building location in order to improve the NV behaviour in a building prior to its construction. It consists in analysing the relative building position regarding the surrounding buildings and the prevailing wind conditions. It is assessed how the prevailing wind direction could be modified when it would reach the future building.

The building outdoor opening position and orientation with regards to the prevailing wind direction strongly affects the NV performance of the building [16]. In this particular case, there is a building that may block winds coming from the prevailing direction. Consequently the wind flow might be a little bit disturbed. Therefore the opening size and position should be evaluated to ensure a well-distributed airflow throughout the building.

Design resources may be limited so it should be not possible to set many design alternatives. Therefore, an initial architectural alternatives selection should be done based on experience. In this particular case, it is suggested to include an opening in the ESE building face because it may allow the disturbed prevailing wind to flow through the building improving its NV behaviour. Moreover, a narrow vertical opening may allow wind to enter in the building improving the airflow distribution through the whole building. Then, three design alternatives will be compared; the option without the opening and two options with the opening located in the right or left side of the façade. Other features such as the building indoor distribution, location, shape and orientation are not considered because there are already defined.

The design strategy aims to test each time only a design alternative as in this case the additional opening position. Then, the effect of each alternative should be compared in order to select the solution that better behaves.

Afterwards, a CFD model is created to simulate the surrounding buildings that might create turbulence trails that may influence local wind flow distribution. In this case, it is important to maintain the surrounding definition in the CFD and only modify the tested building to include each opening position alternative.

The discussion compares three alternatives by means of a quantitative and qualitative analysis of the wind flows through the building and the energy saving potential analysis of each solution. Firstly, velocity streamlines are visualized in order to check which alternative improves the indoor air distribution (qualitative analysis). The areas with fully mixed air should be identified as well as dead zones, which should be avoided. It is shown that the option without the additional opening has many dead zones in the building. In contrast, the air is better mixed in the simulations that include the additional opening. Then, a quantitative analysis is done calculating the inlet and outlet air flow rates. The objective is to determine the indoor air replacement time in each case. The less time the better NV performance. Additionally, a parametric analysis of the results is done. The probability distribution of wind makes necessary the research to be done in a probabilistic approach in order to consider all wind values instead of only analysing the average value. Therefore, the wind measurement standard deviation is selected in order to take into account the whole feasible wind range. Then all feasible wind conditions are simulated in the architectural alternatives that are compared once again. The conclusion is that the new opening should be included and it should be positioned at the south-west side of the ESE façade.

Afterwards, the energy saving potential is assessed using the general ventilation heat transfer equation by comparing two proposed building designs. Insofar the calculation error is minimized because the comparison of the alternatives, it is concluded that the addition of the proposed opening could reduce up to 1.13 kWh/m^2/year the building energy consumption approximately (m^2 are the façade square meters in a façade with 5% openings). This means approximately a reduction of 434 grams of CO_2 per kWh/m^2 respect the initial building design. This reduction is due to the NV generated only by the prevailing wind conditions during the considered months (May-August) by the added 0.5 x 2 m lateral window in the 4 m height and 5 m width façade. Thus the energy value is based on the particular case study building with a relatively low relation of openings per square meter of façade (5%); it could be possible that this value could be increased with other building typologies, higher percentage of openings in the façade, indoor distributions, environmental conditions, etc. Therefore, it should be taken as a first relative reference and not as a fixed value. Future energy calculation should be extended to other wind directions to calculate the complete NV impact on reducing the CO_2 emissions.

Finally, the characteristics of the Valencian residential sector are analysed to estimate the NV design technique impact on new building constructions. It is concluded that the technique implementation could suppose a reduction up to 56.5 tons of CO_2 per year (this value is based on the 2015 new detached houses and the previous case study results references). Consequently the CO_2 potential reduction is very rough and should be taken as a possible reference value. Besides, the design strategy could be extended to other wind directions and other kind of residential buildings. Thus, the CO_2 saving potential could be even higher.

3.5. Ventilated façade performance assessment by means of CFD

This section is based on two manuscripts:

1)

Mora-Pérez, M.; López-Patiño, G.; Bengochea-Escribano, M.A.; López-Jiménez, P.A. Quantification of the efficiency of the ceramic ventilated façade (*Original title in spanish: Cuantificación de la eficiencia de la fachada cerámica ventilada*). *Boletín de la Sociedad Española de Cerámica y Vidrio*. Vol.50 nº2. 2011.

The journal JCR impact factor is 0,432 in 2015. The journal has the position 15 within 25 indexed journals. It belongs to the third quarter (Q3).

2)

Mora-Pérez, M.; López-Patiño, G.; López-Jiménez, P.A. Quantification of Ventilated Façade Effect Due to Convection in Buildings. Buoyancy and Wind

Driven Effect. Researches and Applications in Mechanical Engineering, Vol. 3 Iss. 1, March 2014.

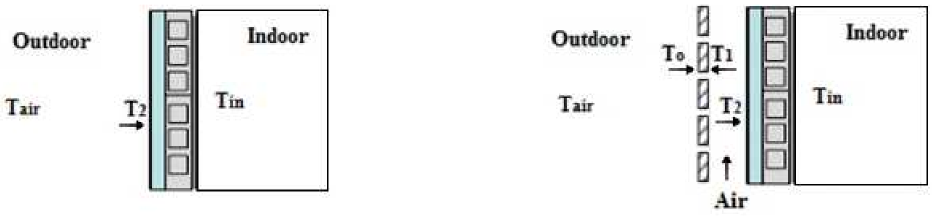
It is an international peer-reviewed journal (not yet indexed in JCR).

Paper 1 summary

At this stage the objective is to take a building architectural element such as the ventilated façade and apply the CFD technique to assess its operation under certain environmental conditions. The ventilated façade is selected from between all possible alternatives because it is a passive element that takes advantage of the air flow conditions to improve the energy performance of the building, especially in regions like the Valencian Community (Spain) in which summer over-heating becomes the major energy consuming source in buildings [10, 17]. The ventilated façade is made of an external ceramic layer that is separated from the building surface by a metallic structure that allows an air flow to circulate upwards through it. Annex 3 and 4 describe in detail the ventilated façade characteristics. The ventilated façade geometry is selected to make feasible a research by means of CFD i.e. a balance between the façade dimensions and the computational time is done.

The building energy improvement is done by the air movement in the façade layer due to wind and buoyancy forces. Consequently the different element temperatures should be taken into account in the simulation. Thereby the energy equation is included in the CFD numerical code equations.

The aim is to estimate and quantify the ventilated façade energy potential saving regarding its capacity for cooling the building under certain environmental conditions. The methodology compares the temperature in the external building surface with and without a ventilated façade (T_2). Figure 5 shows a section of the case a) without ventilated façade and case b) with ventilated façade. It is assumed that the environment air temperature is T_{air} = 25ºC and the temperature of all ceramic panels is $T_0 = T_1$ = 31ºC based on field measurements. Moreover the environmental radiation and the thermal inertia of the different materials are not considered in the simulation. The research compares the ventilated effect by comparing both cases; consequently the different building wall layers effect is omitted.



Case a) Case b)

Figure 5. Temperature nomenclature

The CFD results show whether air velocity in the ventilated chamber is increased while the temperature of the panel (T_1) is increased. Additionally, the temperature of the external surface of the building in contact with the ventilated chamber ($T_{2\_b)}$) is decreased more than 4.48K when T_1 is kept at 31ºC due to the air temperature and its movement in the chamber. If the ventilated façade had not been installed, this temperature would have been $T_{2\_a)}$ = 31ºC. The temperature decrease on the external face of the wall in contact with the ventilated chamber means a cooling energy reduction in the indoor building respect the building without the ventilated chamber ($T_{2\_b)} < T_{2\_a)}$). Generally, it means that the ventilated chamber has a cooling capacity due to the air movement and temperature through it. Moreover, the higher the panel temperature is (T_1), the higher the temperature difference between it and the external surface of the building in contact with the ventilated chamber ($T_1 – T_{2\_b)}$). Thereby, the passive cooling effect generated by the ventilated façade is more effective. Moreover the cooling effect of the air in the ventilated chamber has a linear behaviour respect the panel temperature increase for the temperature conditions simulated.

Finally, an energy balance comparison is done between the building with and without the ventilated façade, concluding that the ventilated façade could reduce up to 58.7% the heat transferred into the building through the façade for the tested environmental conditions. Thus, the CFD techniques are shown as a powerful tool to assess the behaviour of architectural elements in contact with fluids.

Paper 2 summary

The contribution analyses deeply the fluid mechanical behaviour in ventilated façades. The air flow is moved by either wind driven or buoyancy forces in the ventilated chamber [18]. The aim of the present contribution is to quantify the influence of buoyancy-driven ventilation on air velocity increase in ventilated chambers. This phenomenon is mathematically analysed using CFD techniques.

A CFD model is created to simulate air velocity in a ventilated façade taking into account wind velocity and other hydrodynamic features such as open joints, buoyancy and wind driven ventilation, etc. Thermal inertia of the different materials and radiation effect are not considered. The CFD model description is detailed in annex 4.

The research strategy consists on carrying out a first set of simulations that are taken as reference. In these reference simulations there are no buoyancy-driven effects (BDE) because all temperatures are uniform. The air movement in the ventilated chamber is only due to wind dynamic pressure. Then, different air velocities and temperatures are considered in order to analyse the relative weight of wind dynamic pressure over the buoyancy effect. On the one hand, wind speed is changed, keeping the same temperature difference between the internal face of the panel and wind. On the other hand, the temperature gradient between the panel and wind is changed for low wind speed in which BDE is significant.

The results show that as wind velocity is increased over 0.5 m/s, no air velocity changes in the ventilated gap due to BDE are observed. The velocity increase due to BDE is negligible with respect to the effect of wind-driven forces. In the particular case of 1.5 m/s wind velocity and 8K temperature gradient between wind and the indoor panel face, the air velocity increase due to BDE is negligible. Consequently, the behaviour of the air velocity in the ventilated façade mostly depends on outdoor weather conditions. On the other hand, as wind velocity is decreased under 0.5 m/s, the air velocity is increased due to the BDE having a higher specific weight in the total air velocity change. Furthermore, air velocity is proportionally increased as the temperature gradient determining the BDE is increased at low wind velocities ($v_{wind} = 0.2$ m/s).

The final effect is that the external façade face temperature is decreased due to ventilation at low wind velocities (≈ ≤ 0.2 m/s). In the particular case studied, the natural convection buoyancy-driven effect achieves a 35% of the temperature decrease and the other 65% is due to the wind-driven effect. Oppositely, the temperature reduction due to convective effect is negligible at high wind velocities (≈ ≥ 1.5 m/s). Then, ventilated façades design should be done mainly focusing on wind conditions.

Finally, it is concluded that BDE is negligible with respect to the wind-driven effect in summer at high wind velocities (≈ >1.5 m/s). Consequently BDE does not influence ventilated façade cooling effect. Moreover, at low wind velocities (≈ <0.5 m/s), the buoyancy-driven effect has a higher specific weight in the total air velocity change in the façade. In conclusion, the higher air velocity in the ventilated façade is accelerated, the higher cooling effect is achieved.

Finally, the independence of the presented numerical results from the geometry should be tested in the future, although the same tendencies for other geometries are expected. In any case, the current conclusions show which the main drivers of NV effect in the ventilated façades are. Further research will include how the geometry of the ventilated façade affects the particular numerical results.

3.6. Building location analysis by means of CFD

This section is based on the paper:

> Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation. *Advances in Engineering Software* 88, 73-82, 2015.

The journal JCR impact factor is 1,673 in 2015 in the engineering multidisciplinary scope. The journal has the position 24 within 85 indexed journals. It belongs to the second quarter (Q2).

Paper Summary

The awareness and understanding of building locations NV potential should be highly recommendable in order to maximize NV use in buildings [19]. The neighbouring buildings and its relative position towards the prevailing wind direction affects the new building energy behaviour. Consequently designers should select the most energy efficient building location, position and orientation, if possible. The objective at this stage of the research is to show a design strategy to assess and select the most suitable building location depending on the NV behaviour inside the building. It consists in evaluating two possible locations with different surroundings using CFD models. Indoor and outdoor wind behaviour information is provided by the CFD simulations, which are used to compare qualitatively and quantitatively the building NV behaviour in each location.

Two computational models are created, one for each possible location. The details of either each CFD model or location are described in the annex 5. Velocity streamlines are used to know the prevailing wind flow development that is influenced by the neighbouring buildings. It is observed that the prevailing wind direction is changed as well as the wind incidence angle within the building in the first location analysed. A qualitative analysis is done to detect dead zones and evaluate the air swept effect through the building. The simulations show that the wind reaches the building by the face in which the machine room is placed and then it enters in the building. It could be possible that the wind flow temperature raises a little bit before entering in the building, what it is not desirable during the warm season because additional cooling loads would be request to cool down the air in order to maintain indoor comfort conditions. Additionally, this extra load may decrease the building energy efficiency. Oppositely, it can be observed that the surrounding buildings of the second location do not disturb the prevailing wind direction. Moreover the air flow crosses the building from the south to the north creating an air swept effect through the complete building in the second location. Then, a quantitative analysis is done by analysing the time needed to completely replace the indoor air by outdoor air. The less time it is needed to passively renovate the indoor air, the more efficient natural ventilation is. The present case study shows that the building in location 1 needs 13.1% more time than location 2.

The present research stage provides the NV effect analysis according the initial observation of the neighbouring buildings effect on wind flows development around the case study building prior to construction. Particularly, this research stage is combined with the strategy presented in section 3.4 (façade opening position selection) in order to select the best façade opening configuration in the second location since the second location is finally selected.

It is concluded that wind flow development depends on neighbouring elements and could be slightly different than expected. Consequently sustainable building design decisions should be taken according to local wind studies or simulations and its quantitative and qualitative assessment. The research shows the huge CFD technique potential to assist sustainable designs considering not only indoor and façade building distribution, but also external elements.

3.7. Indoor comfort conditions analysis by means of CFD

This section is based on the paper:

> Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A. A CFD study for evaluating the effects of natural ventilation on indoor comfort conditions. AIMS Environmental Science, 4(2): 289-309, March 2017.

It is an international journal peer-reviewed indexed in the Emerging Sources Citation Index (ESCI - Web of Science).

Paper Summary

The implementation of NV techniques presents certain challenges, especially in the systems in which the wind driven effect ought to be maximized [20]. Accordingly, the main challenge of these designs is to minimize the draught risk and ensure acceptable comfort conditions while reducing the energy costs of the building, as it is in the present case. Furthermore, the objective at this stage of the research is to analyse the real NV effect on indoor comfort conditions. Comfort conditions are assessed through the indexes draught risk (DR), predicted mean vote (PMV) and predicted percentage of dissatisfied people (PPD) [21]; the particular equations of each index are described in annex 6. In addition, the potential energy saving improvement is evaluated.

The most relevant fact at this stage of the design process is that the full-scale building is already constructed. Therefore the CFD indoor models, which are used in the building design stage, can be successfully validated. This feature is important because it proves that the procedure followed to define and validate the initial CFD models is feasible and reliable. The validation is done through the comparison of the measured and calculated results that are available in annex 6. Then, the visual and numerical CFD model capabilities are used to evaluate the balance between NV performance and comfort conditions in the occupied spaces. The numerical formulas of the comfort indexes are programmed in the commercial CFD software used and then calculated in the volume of interest. It is depicted in the annex 6.

The strategy consists in checking whether similar comfort indexes that are reached with lower air temperatures could be achieved by the only effect of NV. Therefore, it should be assessed whether equivalent comfort conditions to lower air temperatures are reached with higher air temperatures and higher air velocities due to NV. Consequently the indoor target temperature would be increased maintaining the comfort conditions and improving the energy performance of the building. Three different wind temperatures and three different wind velocities are used to compare DR, PMV and PPD comfort indexes.

An indoor air temperature increase assessment is done in order to check whether comfort indexes belonging to lower indoor temperatures can be reached by increasing the air temperature and velocity. Nevertheless, velocity should not be increased as much as possible because of draught risk. DR is the most common reason for local discomfort that causes an undesired effect in the human body due to high velocities. Nevertheless, the air velocity increase may lead to a temperature increase maintaining similar comfortable feeling. It means that indoor temperature can be kept slightly higher than desired to increase energy efficiency in summer conditions. In this particular case, it is concluded that the occupied area has acceptable maximum mean air velocities and only higher velocities than expected (0.25 m/s) are achieved in the corridor. Nevertheless, special draught agreements are done in transit zones such as the corridor. It is concluded that DR has no critical values (DR<15%) with an acceptable distribution in the occupied zone.

Regarding PMV and PPD indexes; PMV is ranged between 0.2 and 1 (it means between an almost neutral and a slightly warm indoor condition) for all tested conditions, what is acceptable. In this conditions and during the warm season, an air velocity increase will be beneficial because it will decrease the warm discomfort for people feeling warm in their body. Consequently, the PMV index is not so much modified when increasing the indoor temperature if the air velocity is increased accordingly. However, higher air velocities may increase the predicted dissatisfied people, which will be more spread along the occupied volume. The simulation results show that PPD index belonging to a lower indoor air temperature is ≈ 60% maintained with higher indoor temperatures and air velocities.

The conclusion is that the NV strategy could manage to increase the indoor air temperature by 1ºC, while maintaining no draught risk and 60% of the initial percentage of people likely to feel uncomfortable. Thus, energy savings are achieved due to the indoor air temperature increase. Otherwise additional mechanical system should be necessary to maintain the comfort conditions if the indoor air temperature is increased. The energy needs of the building are then reduced. Thereby, a proper NV focus during the initial design stage could improve buildings energy efficiency without compromising indoor comfort conditions.

CHAPTER 4: CONCLUSIONS AND CONTRIBUTIONS

The present thesis describes a sustainable building design strategy based on CFD techniques in order to improve NV use in buildings. The design strategy consists in simulating different architectural solutions in order to select the most suitable alternative prior to building construction.

CFD is presented as a powerful technique to assist sustainable building designers, not only to increase the building energy efficiency performance but also to ensure indoor comfort conditions and add value to society. CFD techniques are applied in different scenarios in order to take advantage of environmental wind flow hydrodynamic conditions to improve buildings energy performance by reducing its cooling demand during the warm season. Passive design features and elements are further analysed and selected. The research provides qualitative and qualitative criteria to select the most suitable building design alternative. Furthermore the strategies are conducted through an original case study building in order to ensure the design strategy reliability.

Additionally, NV use potential is assessed in a particular region: the Valencian Community coastal region. The conclusion is that the potential CO_2 reductions are considerable. Therefore, the design approach can be applied in regions with similar environmental conditions around the World. In this frame, new design strategies developments as well as the addition to the literature of successfully developed new buildings create added value for society.

The particular results are detailed in the corresponding annexes. The main general conclusions of the complete research that are independent of the case study are summarized as follow:

- The building adaptation to the environmental conditions of each region in order to maximize the use of the available sustainable energy resources can reduce substantially CO_2 emissions since the residential sector is one of the major potential sectors in which energy efficiency can be improved.
- The here described design strategy is based on comparative calculation between CFD results. Comparative results are mostly recommended because they reduce the remaining uncertainties, especially between CFD models. Moreover, comparative calculation between CFD results and full-scale measurements are used to validate the CFD models. The comparative calculations are based on:
    - Qualitative analysis. NV is analysed by velocity vectors, streamlines and its visualization. Dead zones should be avoided. The building should be swept and the air should be completely mixed in order to maximize NV use. Moreover air paths should be avoided to pass through heat sources such as machinery rooms before entering in the building during the warm season.
    - Quantitative analysis. NV is measured by the time that indoor air needs to be exchanged by outdoor air. The design alternative that requires less time should be the best option to improve NV use. Furthermore maximum air velocities should be considered in order to ensure indoor comfort conditions.
    - Parametric analysis. It is proposed to take into account not only the most probable wind modulus, but also the other wind modulus possibilities since wind is defined in probabilistic terms.
- Building neighbouring elements should be analysed in order to increase building energy behaviour by NV techniques. Furthermore, CFD simulations should be used because many times wind results can be quite different than expected. Thereby, CFD simulations become a virtual laboratory in order to assist designers to take decisions.
- The CFD techniques use allows selecting the most efficient architectural alternatives in the building design stage. Moreover, the technique permits the

optimization of the surrounding resources use to reduce the building energy demand along the whole buildings life time.

- CFD techniques should be used to select different architectural solutions as well as to know the behaviour of particular elements in order to quantify and optimise the use of natural resources available in each region. The techniques are especially suitable during the initial design stage when there is no access to full-scale measurements because the building is still not built. The proposed architectural solutions analysed are focused on:
    - Building location. The indoor wind flow evolution through the building is used to assess NV in two different locations. The location in which NV behaves better is then selected.
    - Façade openings position. The proper design of outdoor opening shape and position could originate well naturally ventilated buildings in which energy demand should be reduced. The façade openings position in which NV behaves better is then selected.
    - Ventilated façades are a suitable architectural element to reduce building energy demand during the warm season. Additionally, wind driven forces have major impact than buoyancy-driven forces at high wind velocities (≈ >1.5 m/s). Moreover, at low wind velocities (≈ <0.5 m/s), the buoyancy-driven effect has a higher specific weight in the total air velocity change although the driven effect is still higher. Therefore, ventilated façades should be designed based on wind driven forces.
- Indoor comfort conditions are assessed through CFD techniques by means of simulations and validation of comfort indexes (DR, PMV and PPD). The conclusion is that the NV design strategy applied in the building could manage to increase the indoor air temperature maintaining no draught risk and almost the percentage of people likely to feel uncomfortable. In the particular case study, the air temperature can be increased by 1ºC almost maintaining initial comfort conditions. Otherwise, additional mechanical system should be necessary to

maintain the comfort conditions if the indoor air temperature is increased. Thereby, a proper NV focus during the initial design stage could improve the building energy efficiency without compromising indoor comfort conditions.

Additionally, the following general guideline is provided in order to assist building designers in the use of the presented design approach.

1. .Analysis of the particular environmental conditions of the region in which the building will be constructed. It is highly recommended the wind rose analysis in order to prioritize the wind directions considered.
2. Creation of the CFD model including the particular features of the neighbouring volume near the future building. Afterwards, validation of the outdoor CFD model.
3. Use the CFD models to analyse which could be the best architectural solution according to the feature that must be analysed to determine its influence on the NV behaviour of the building: building location influence, façade opening configuration (as it is done in the present research), comfort conditions, indoor volumes configuration, distribution, etc. The analysis should include the indoor comfort conditions analysis in order to ensure that the future indoor conditions will accomplish the comfort conditions Standards.
4. Take the most suitable building construction decision depending on the results provided by the CFD simulations.
5. Proceed with the building construction.
6. Validation of the indoor CFD model after the building construction in order to close the design loop.

In any case, the specific design approach features used in the presented case study should be followed in detail according to the annexes of the present thesis.

Recommendations for future research

There are many features that can be done to extend and continue the current research.

Firstly, the present research is focused on the most probable environmental conditions in the Valencian Community Region: the warm season and the prevailing wind direction. Future research should include the analysis of building NV behaviour during the whole year. Environmental conditions during the cold season and all wind conditions should be simulated as well. Moreover, it should be included the analysis of air fluctuation and thermal behaviour (buoyancy effect, thermal inertia, radiation, etc.) effect on complete building energy performance and indoor comfort conditions.

Secondly, it is highly recommendable to extend the comfort conditions analysis by means of the adaptative comfort model.

Thirdly, the analysis of NV potential in different European regions and the limitations for different building typologies and shapes are also recommended to be done in the future.

To conclude, the ultimate aim of the research is to achieve more environmentally friendly buildings and add a reliable NV behaviour analysis by CFD techniques to the literature.

CHAPTER 5: REFERENCES

[1] UN Document: Gathering a Body of Global Agreements. NGO Committee on Education of the Conference of NGOs from the United Nations. 1987-1988 http://www.un-documents.net/wced-ocf.htm

[2] Hamdy, M.; Hasan, A.; siren, K. Applying a multi-objective optimization approach for design of low emission cost-effective dwellings. Build. Environ. 2011, 46, 109-123.

[3] Linden, P.F. The fluid mechanics of natural ventilation. Ann. Rev. Fluid Mech. 2009, 31, 201-208.

[4] Chen Q. Ventilation performance prediction for buildings: a method overview and recent applications. Build Environ 2009;44:848–58.

[5] Yuguo L, Heiselberg P. Analysis methods for natural and hybrid ventilation: a critical literature review and recent developments. Int J Ventilation 2003;1(4):3–20.

[6] Martin L, James A, Heiselberg P, Yuguo L, Stathopoulos T. Achieving natural and hybrid ventilation in practice. Int J Ventilation 2006;5(1):115–30.

[7] Khan N, Yuehong S, Riffat SaffaB. A review on wind driven ventilation techniques. Energy Build 2008;40(8):1586–604.

[8] Ray SD, Gong N-W, Glicksman LR, Paradiso JA. Experimental characterization of full-scale naturally ventilated atrium and validation of CFD simulations.Energy Build 2014;69:285–91.

[9] Word Commission on Environment and Development (WCED). Report of the World Commission on Environment and Development: Our Common Future. 1987.

[10] Agencia Estatal de Meteorología de España (AEMET). Available online: http://www.aemet.es (accessed on January 2017).

[11] CD-Adapco. Star CCM+ User’s Manual. Available from: www.cd-adapco.com.

[12] Shao J, Liu J, Zhao J. Evaluation of various non-linear k-epsilon models for predicting wind flow around an isolated high-rise building within the surface boundary layer. Build Environ 2012;57:145–55.

[13] Franke, J.; Hellsten, A.; Schlünzen, H.; Carissimo, B. Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment; COST Action: Brussels, Belgium, 2007.

[14] Justus, C.G.; Mikhail, A. Height variation of wind speed and wind distributions statistics. Geophys. Res. Lett.1976, 3, 261–264.

[15] Hajdukievwicz M, Geron M, Keane MM (2013) Formal calibration methodology for CFD models of naturally ventilated indoor environments. Build Environ 59: 290-302.

[16] Lomas JK. Architectural design of an advanced naturally ventilated building form. Energy Build 2007;39:166–81.

[17] Instituto Valenciano de la Edificación (IVE). Generalitat Valenciana. Available online: http://www.ive.es.

[18] Linden PF. The fluid mechanics of natural ventilation. Annu Rev Fluid Mech 2009:201–8.

[19] Suresh BS, Srikanth M, Robert FB. Passive building energy savings: a review of building envelope components. Renew Sustain Energy Rev 2011; 15: 3617–31.

[20] Visagavel K, Srinivasan PSS. Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening using CFD. Sol Energy 2009;81(1):2–5.

[21] ISO Standard 7730 (2005) Ergonomics of the thermal environment—Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria.

ANNEXES

Annex I

López-Jiménez, P.A., Mora-Pérez, M., La Ferla, G., Roset-Calzada, J.

Increasing the value of buildings through environmental design.

COST Action TU1104 – Smart Energy Regions – Cost and Value. ISBN: 978-1-899895-22-9. February 2016. International institutional publisher.

Annex II

Mora-Pérez, M.; Guillen-Guillamón, I.; López-Patiño, G.; López-Jiménez, P.A.

Natural Ventilation Building Design Approach in Mediterranean Regions—A Case Study at the Valencian Coastal Regional Scale (Spain).

Sustainability 8, 855, 2016. JCR impact factor 1,343 (Q3).

Annex III

Mora-Pérez, M.; López-Patiño, G.; Bengochea-Escribano, M.A.; López-Jiménez, P.A.

Cuantificación de la eficiencia de la fachada cerámica ventilada.

Boletín de la Sociedad Española de Cerámica y Vidrio. Vol.50 nº2. 2011. JCR impact factor 0,432 (Q4).

Annex IV

Mora-Pérez, M.; López-Patiño, G.; López-Jiménez, P.A.

Quantification of Ventilated Façade Effect Due to Convection in Buildings. Buoyancy and Wind Driven Effect.

Researches and Applications in Mechanical Engineering, Vol. 3 Iss. 1, March 2014. International publisher peer-reviewed.

Annex V

Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A.

Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation.

Advances in Engineering Software 88, 73-82, 2015. JCR impact factor 1,673 (Q2).

Annex VI

Mora-Pérez, M.; Guillen-Guillamón, I.; López-Jiménez, P.A.

A CFD study for evaluating the effects of natural ventilation on indoor comfort conditions.

AIMS Environmental Science, 4(2): 289-309, March 2017. Emerging Sources Citation Index (ESCI - Web of Science), extended JCR 2016.

Annex I

Increasing the Value of Buildings through Environmental Design

Abstract: The environmental design of the built environment can be defined as a design strategy focused on reducing the depletion of resources such as energy, water, and raw materials. Sustainability in new constructions and refurbishment has achieved paramount importance in the last decades for all the agents involved in building management such as urban planners, policymakers, developers and designers as well as citizens, due to social, economic and environmental implications.

The application of environmental design to new and existing buildings becomes a tool to increase the value in two ways: on one hand, the use of materials and techniques with smaller environmental impact can make the building more attractive for particularly conscious consumers. On the other hand, the use of sustainable strategies for heating, cooling and ventilation can make the building less energy consuming. The purpose of this paper is to analyse how environmental design can positively affect the cost and final value of a building.

Introduction

Comfort achievement in building is the final objective for many designers. Maintaining comfort conditions uses resources and energy, thus efforts must be done during the design phase to decrease the operational costs for maintaining the performance of the building. Environmental design is focused on reducing the use of resources while providing final products of high quality. This is achieved by optimizing the potential for ‘passive’ strategies which make use of the environmental conditions [1].

The environmental conditions in which human beings are in a state of comfort are limited within a small range of temperature, humidity and air speed. These conditions are mostly imposed by our physical constitution, but are also affected by cultural standards.

As comfort conditions do not always exist in nature, men have developed clever strategies to generate barriers that protect them from adverse weather and make them feel comfortable. These strategies include the development of buildings appropriate for the local climatic conditions [2]. Some years ago in the work of Garg [3] it was concluded that two thirds of the cases of discomfort could be solved by using simple passive techniques based on thermo-physical and geometric properties of buildings.

Responding to local climatic conditions, environmental designers will try to achieve indoor comfort conditions with the least possible expenditure of energy [4]. These principles have guided the design of traditional buildings, which take advantage of local conditions through the layout and shape of the building, and in recent times have inspired the concept of ‘passive’ architecture [5].

Nowadays, comfort conditions in buildings are generally achieved with large amounts of energy, as little attention is paid to the resouces needed. As buildings are responsible for 33% of the total energy consumption in the world [6], buildings are major contributors to problem of climate change and, more generally, environmental pollution. This situation clearly needs to change.

As indicated by El Dean El-Alfy [7], sustainable development meets the needs of the present generation without compromising the ability of future generations meet their own. ‘Green’ buildings refers to buildings which are environmentally responsible and resource-efficient throughout their life-cycle, from siting to design, construction, operation, maintenance, renovation, and demolition [8]. Thus these buildings attempt to meet the needs of society whilst reducing their impact in social, economical and environmental terms.

From an economical perspective, there are benefits in the improvement of energy efficiency buildings and resource use in buildings. The implementation of sustainability can be a powerful tool to save on energy bills, to reduce energy dependence and to increase competitiveness.

Environmetal design has many benefits; among others, it can reduce annual utility expenses and maintenance costs [9]. In order for this design approch to be effective, significant decisions regarding technology are taken during the design phase and have an effect on the final performance of the building.

Nowadays, several assessment methodologies exist to evaluate the sustainability of buildings. These can be categorized into three groups [10]:

- Those based on the evaluation of actions and associated impacts, such as LEED V3 [11] and BREEAM [12].
- Those based on the concept of efficiency such as the Japanese CASBEE [13].
- Those based on a tree structure with different categories and criteria, in order to be adapted to each country particularities. In this case we can find the SB tool [14].

Increasing building value through environmental design

‘Value engineering’ should be considered as a philosophy to optimize the value of an item fulfilling the objectives of its purpose. In our case, this involves many aspects of the design of buildings without compromising their final quality. During the design phase, engineers and architects must select and finalize materials, and components of the building. Environmental design includes the sustainability assessment of construction products, which is becoming easier to ceonduct through Environmental Product Declarations (EPD).

When considering the sustainability of the built environment, the focus quickly moves to energy retrofit projects, since existing buildings have high environmental impact. Economical considerations are also involved, as existing buildings often present high operational energy costs as well as a large potential for energy savings.

Economic benefits of environmetal design

The economic benefits of environmetal design include lower energy and water consumption, smaller construction waste, lower operations and maintenance costs, lower environmental impact, and increased comfort, health and productivity. Unfortunately environmental design can require higher investments during design and construction phases. However, this situation is changing, and operational savings do not have to come at the expense of higher initial costs.

Some environmental design features have higher initial costs, but payback periods are often short and the life-cycle cost typically lower than the cost of conventional buildings. Apart from those direct savings related to energy consumption, there are other potential economic benefits that can increase in the value of the building if the correct indicators are shown:

- Increase in health and comfort of the building occupants. This can reduce levels of absenteeism and increase the productivity of workers. For instance, it has been estimated that improving occupants’ productivity in commercial buildings, considering Indoor Environmental Quality (IEQ) aspects, in the US could bring economic gains between $20-$160 billion in 1996 [15].
- Longer building lifetime and less investment in retrofitting and maintenance.
- Higher community acceptance and support.
- Reduced costs from air pollution at the regional scale.

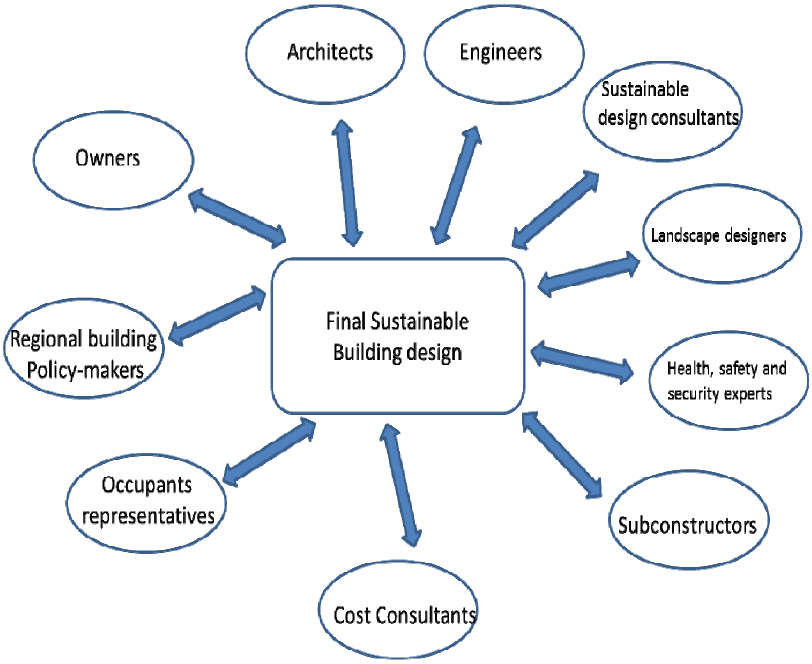


Figure 1. Agents involved in sustainable design

Initial cost of environmental design

A project should include environmental design in its conceptual phase in order to realize the full benefits. The design team should be composed by all the agents involved in the final product, in order to increase the synergy of the solution. Environmental design requires close cooperation of all these agents (Figure 1) who form an integrated design group. The team should assess the sustainability of products and components used in the building in order to meet the specific sustainable requirements of the project.

Furthermore, the goal of the design team is to develop innovative solutions without increasing the budget of the building. Most of the times, environment-friendly products are more expensive than conventional ones. Nevertheless these additional costs imply higher energy savings during the life of the building, and therefore if the economical savings overtake the initial expenses, the investment is profitable.

Significant decisions must be taken in order of not to increase the initial and final cost of the building:

- Eliminate unnecessary elements in the building. Designers have to consider whether some elements can be avoided, like internal doors, ornamental features, etc. This will decrease the use of materials, make the building lighter and decrease the initial cost.
- Use recycled materials and modular solutions.
- Choose a correct location for the building in order to decrease the initial need for site infrastructure. Some particular locations increase very much the initial cost because the waste disposal or de conditioning of the site costs are very high. This must be avoided by choosing a better location.
- Have a bioclimatic approach to study how to achieve high comfort levels for the occupants, adaptation geometry, orientation and construction techniques to the climate of the site [16].

Environmental design aims to create buildings which are more comfortable and healthier than conventional buildings without implying an increase in costs by supporting comfort conditions with minimum energy demand.

Cost savings across the life of green buildings

The benefits of green buildings should be considered throughout their life-cycle and not just in comparison to the upfront costs, because savings resulting from investment in environmental design usually exceed the additional upfront costs.

Just as much as there are design decisions that can reduce the initial cost of a building, there are equally complementary design solutions that can reduce the operational cost of a building. The aim of these efforts is to decrease the energy cost across all the life-cycle of the building. For example:

- Optimize site and orientation. An appropriate choice of site will decrease the energy cost across the life-cycle. Solar radiation, natural ventilation and shading can decrease the use of energy used to achieve the comfort of the occupants.
- Choose the best room distribution considering the future use the building.
- Install adequate thermal insulation. A well-insulated envelope limits heat losses and therefore less energy will be needed to reach the thermal comfort conditions.

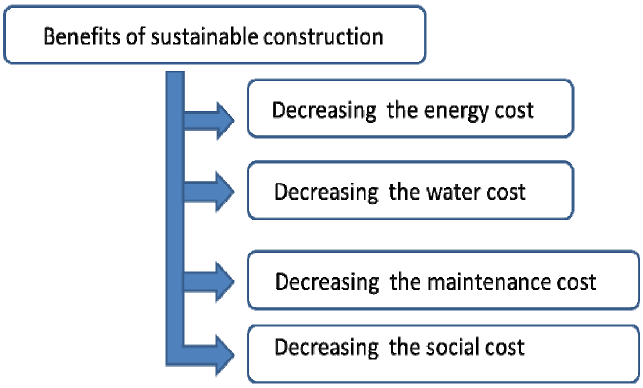


Figure 2. Benefits of environmental design.

The benefits of environmental design include some elements that are relatively easy to quantify (Figure 2), such as energy and water savings, as well as those that are less easily quantified, such as the decreases in maintenance and material costs, as well as other indirect social and environmental benefits.

Medium term energy savings: decreasing the energy cost

Nowadays, many commercially available technologies can help designers to effectively minimize a building's energy costs. However, these technologies should be integrated in the fist stages of the design process.

A complete strategy should be adopted in order to decrease the medium term energy costs. The aspects to be considered are referred in Table 1. First, a high percentage of the total building energy demand is due to heat losses through the building envelope. Appropriate envelope insulation will limit heat losses. Nowadays many different materials are being developed to meet the need for energy efficiency, environmental design and cheaper costs. Additionally, adequate envelope design can maximize the use of natural ventilation techniques which help reducing the heat gains, such as ventilated façades, atria and effective distribution of windows.

Table 1. Energy implications of sustainable building in the medium term.

Item	Measure	Action
Building envelope	Window distribution	Optimize solar radiation and natural ventilation
	Wall insulation	Sustainable materials. Ventilated façades
	Efficient Windows	Automatic performance avoiding thermal bridges
	Thermal bridge	Improve continuity between insulation layers to avoid heat loses
Mechanical Systems	Main system controller	Energy management. Choose the best way to produce/consume energy. Smart-consume or energy-saving
	Heating, Ventilation and Air Condition systems (HVAC)	High efficient air conditioning systems: ventilation, underfloor heating-cooling with heat pumps.
	Domestic Hot Water	Produce it by renewable energies such solar thermal collectors and heat pumps. Recapture energy from waste hot water.
	Water heater	Efficient condenser in the roofs. Layout distribution for decreasing energy losses

Item	Measure	Action
	Low pressure ducts	Enlarging the duct sizes for saving energy
Lighting	Increase day lighting.	Add Skylights
	Reduce lighting intensity	Appropriate Lighting power
	Perimeter automatic daylighting controls	Daylight sensors

A ventilated façade generally consists in a continuous layer placed over the building wall leaving a naturally ventilated cavity. Depending on outdoor conditions, a cooling effect can be produced by the ascending flow of air, which is induced by the chimney effect, reducing the heat gains [17].

An atrium is a building central space mainly designed to expose indoor spaces to daylight and to maximize direct solar gains. An appropriate atrium design can also help reduce heat gains by increasing natural ventilation [18].

HVAC systems are used to create comfortable indoor conditions in buildings. An efficient HVAC system can also reduce the amount of energy needed to meet the demand for heating and cooling the building.

Currently, air- and ground-source heat pumps are one of the most advanced technologies available for heating/cooling and domestic hot water (DHW). Ground-source heat pumps collect energy stored in the earth and use it to heat water. The energy stored in the groung as is an extremely reliable and constant energy source. The heat pump uses some amount of electrical energy to accomplish the work of transferring heat from the original source to a medium (usually water) with very high efficiency, as for each kW of electric energy used by the system; a higher quantity of heat is extracted form the source. Heat pumps emit no harmful substances and use very small amount of electricity. This technology can be used in combination with solar thermal heating and condensing gas boilers, though it needs to be managed by an intelligent control to guarantee the lowest use of energy and the highest level of comfort. The system must be able to choose the best energy source to decrease the operational cost in each moment: smart energy supply.

Regarding lighting, building openings can be designed to increase the penetration of natural light and therefore reduce the artificial lighting demand where possible. Energy-saving light bulbs can be installed and steps have to be taken to ensure that lights are turned off in unused areas.

To provide a general quantification of energy costs is difficult, but literature indicates that energy saving induced from eco-feedback system can range from 5% to 55%, [19-24].

Water heating represents also an important amount of the energy demand across the building life-cycle. The use of roof condensers and a rational layout of the hot water distribution system (minimizing the distance between heater and consumption points) can decrease significantly the energy lost in the system.

Medium term water savings: decreasing the water cost

Nowadays, several techniques can be used to reduce the water consumed in buildings. These technical devices decrease the medium term use of fresh water and not necessary increase the cost of the design project: ultra low-flow showerheads, faucet aerators, or dual-flush toilets. In certain application, the re-use of non potable or regenerated water can be proposed.

Environmental design will also be focused on the necessity to improve the efficiency of water uses. This can be done implementing new water reuse systems and better controls on water losses [25]. Green building water conservation strategies can be considered into four categories [26]:

- Efficiency of potable water use through better design/technology.
- Capture of grey water – non-faecal waste water from bathroom sinks, bathtubs, showers, washing machines, etc. – and use for irrigation.
- On-site storm water capture for use or groundwater recharge.
- Recycled/reclaimed water use.

Facilities repair: decreasing the maintenance costs

Environmental design is intended to increase durability and easier maintenance. The accessibility to services areas or the use of durable materials will decrease maintenance and repair costs. Seasonal maintenance strategies will promote proper use of facilities getting an efficient use of resources.

Furthermore, environmental design potentially improves efficiency and convenient collection of recyclable materials, such as glass, paper, plastic or others. This affects the environmental value of the building by reducing annual disposal costs for the occupants.

Indirect benefits of environmental design: social cost savings

Environmental design has additional benefits related to social and life quality aspects. It is difficult to quantify their economical effects in a single indicator, but there is no doubt that these aspects increase the value of the final building [27]. For instance, a lower absenteeism and improved productivity is related to these types of buildings.

The social response to some of the features of green buildings can be an increase in people's satisfaction, reduction in mistakes, reduced absenteeism and increased productivity, thus reducing labour costs [28].

However, economical benefits are not the main motivating factor everyone. The cost-effectiveness of green buildings makes environmental design a pragmatic way to ensure the protection of the planet's resources. Furthermore, buildings that are constructed or retrofitted according to environmental design usually provide high-quality indoor environment, thereby decreasing the risk of illnesses in the occupants due to indoor pollution.

The retrofit of buildings occupied by households in conditions of fuel poverty can result in substantial energy savings. A retrofit based on environmental design can tackle fuel poverty problem with high cost-effectiveness and generate additional benefits [29]. For example in Spain domestic building retrofits generate near 17 full-time workplaces per million Euro invested, or 47 full-time workplaces per 1,000 square meters retrofitted domestic area [30].

Other social, economical and environmental benefits are linked with the refurbishments of existing buildings, such as the reuse of materials (decreasing the overall environmental impact), possible reductions in transport costs, reduced landfill disposal, local economic development, retention of community infrastructure and neighbourhood renewal and management [31].

There are several certification programmes such as Green Globes and the U.S. Green Building Council's LEED: Leadership in Energy and Environmental Design Green Building Rating System. These certifications aim to certify the 'performance' of green buildings, or how much 'sustainably designed' is the building, in order for society to take it into account.

Conclusions

This chapter provided an overview of the value added to green buildings through environmental design. The main points arising from this reflection can be summarised as follows:

- There is a common perception that green buildings are more expensive than conventional buildings. This might have been the case in the past but the present situation is much more favourable.
- Green buildings increase the energy savings across their life-cycle. The overinvestment due to environmental design, if quickly recovered, can generate earnings.
- Environmental design encourage scientist to investigate and create new materials, building solutions and HVAC systems more environment-friendly.
- There are several social benefits associated with improved health and enhanced building occupants performance.

References

[1] Desideri, U.; Proietti, S. Sdringola,P.; Taticchi,P.; Carbone, P.; Tonelli, F. Integrated approach to a multifunctional complex: Sustainable design, building solutions and certifications, Management of Environmental Quality: An International Journal, 2010 Vol. 21 Iss: 5, pp.659 – 679

[2] Ralegaonkar, R.V.; Gupta, R. Review of intelligent building construction: A passive solar architecture approach, Renewable and Sustainable Energy Reviews 2010, Vol. 14, . 8, 2010, Pp 2238-2242.

[3] Garg. N.K. Passive options for thermal comfort in building envelopes—an assessment. Solar Energy 1991, 47 (6) (1991). Pp. 437–441

[4] Aksoy, U. T.; Inalli, M. Impacts of some building passive design parameters on heating demand for a cold region. Building and Environment 2006. 41(12). Pp 1742–1754.

[5] Parasonis, J.; Keizikas, A. ; Endriukaitytė, A.; Kalibatienė. D. Architectural solutions to increase the energy efficiency of buildings. Journal of Civil Engineering and Management 2012, 18 (1), pp. 71–80.

[6] Urge-Vorsatz D.; Petrichenko, K; Staniec, M.; Eom, J. Energy use in buildings in a long-term perspective, Current Opinion in Environmental Sustainability, Energy Systems 2013. Volume 5, Issue 2, Pp 141-151.

[7] El Dean El-Alfy, A. Design of sustainable buildings through Value Engineering . Journal of Building Appraisal 2010, 6, Pp 69–79.

[8] Ji, Y. and Plainiotis, S. Design for Sustainability. Beijing: China Architecture and Building press 2006. ISBN 7-112-08390-7

[9] Zhou, L.; Lowe. D.J. Economic challenges of sustainable construction. London: The RICS Foundation 2003. Pp. 113–126

[10] Macías, M.; García Navarro, J. Verde, a methodology and tool for sustainable building assessment. Informes de la construcción 2010, Vol 62.

[11] U.S. Green Building Council (USGBC). LEED. Consulted online in October 2014 on: http://www.usgbc.org/leed

[12] BREEAM. Value of green buildings. Consulted online in October 2014 on: www.casbee.jp.

[13] CASBEE. Comprehensive Assessment for Sytstem for Built Environmental Efficiency. Consulted online in October 2014 on: http://www.ibec.or.jp/CASBEE/english/index.htm

[14] IISBE 2015. International Initiative for Sustainable Built Environment. Consulted online in October 2014 on: http://www.iisbe.org/

[15] Jin, Q.; Overend, M.; Thompson P. Towards productivity indicators for performance-based façade design in commercial buildings. Building and Environment 2012, 57 (2012), pp.271-281.

[16] Barajas, L. M.; Roset Calzada, J.; La Ferla, G. Thermal evaluation of buildings. A convenient tool. JIDA’15, III Workshop on educational innovation in architecture. Barcelona, 25th-29th May 2015, pp. 285-298.

[17] Giancola E, Sanjuan C, Blanco E, Heras M.R. Experimental assessment and modeling of the performance of an open joint ventilated façade during actual operating conditions in Mediterranean climate. Energy and Buildings 2012.

[18] Moosavi L, Mahyuddin N, Ab Ghafar N, Ismail M.A. Thermal performance of atria: an overview of natural ventilation effective designs. Renewable and Sustainable Energy Reviews 2014.

[19] Pisello, A.L.; Asdrubali,F. Human-based en-ergy retrofits in residential buildings: A cost-effective alternative to traditional physical strategies, Applied Energy 2014, Volume 133. Pp 224-235.

[20] Azar, E.; Menassa, C. Agent-based modelling of occupants and their impact on energy use in commercial. Buildings, Journal of Computing in Civil Engineering 2012. 26. Pp. 506–518.

[21] Fabi, V.; Andersen, R.V.; Corgnati, S.P.; Olesen. B.W. A methodology for modelling energy-related human behaviour: application to window opening behaviour in residential buildings. J. Build Simul 2013, 6 (4) pp. 415–427.

[22] Chen, J.; Jain, R.K.; Taylor. J.E. Block configuration modeling: a novel simulation model to emulate building occupant peer networks and their impact on building energy consumption. J. Appl Energy 2013, 10. Pp. 358–368.

[23] Seligman, C.; Darley, J.M.; Becker. L.J. Behavioral approaches to residential energy conservation. Energy Build 1978, 1. Pp. 325–337.

[24] Yang, L. ; Yan, H.; Lam J.C. Thermal comfort and building energy consumption implications – a re-view. J. Appl Energy 2014, 115. Pp. 164–173.

[25] Matos, C.; Teixeira, C. A.; Duarte, A.A.L.S; Bentes, I. Domestic water uses: Characterization of daily cycles in the north region of Portugal, Science of The Total Environment 2013, Vol. 458–460, Pages 444-450.

[26] Kats,E.; Alevantis, L.; Berman,A.; Mills, E.; Perlman, J. The Costs and Financial Benefits of Green Buildings. A Report to California’s Sustainable Building Task Force 2003. California. USA.

[27] Frontczak M, Schiavon S, Goins J, Arens E, Zhang H, Wargocki P. Quantitative relationships between occupant satisfaction and satisfaction aspects of indoor environmental quality and building design. Indoor Air 2012;22:119e31.

[28] Haynes, BP. The impact of office comfort on productivity. .J Facil Manag 2006; 6(1):37e51.

[29] Urge-Vorsatz, D.; Tirado Herrero. S. Building synergies between climate change mitigation and energy poverty alleviation. Energy Policy 2012, 49. Pp. 83–90.

[30] Tirado Herrero., S. López Fernández, J.L., Martín García, P. Pobreza energética en España, Potencial de generación de empleo derivado de la rehabilitación energética de viviendas. Asociación de Ciencias Ambientales, Madrid, 2012. Spain.

[31] Power,A. Does demolition or refurbishment of old and inefficient homes help to increase our environmental, social and economic viability?, Energy Policy 2008, Volume 36, Issue 12, December 2008. Pp 4487-4501.

Annex II

Natural Ventilation Building Design Approach in Mediterranean Regions—A Case Study at the Valencian Coastal Regional Scale (Spain)

Abstract: Environmental awareness has led to an increased concern about low carbon technologies implementation. Among these technologies, the following research is focused on the natural ventilation effect evaluation in buildings prior to its construction. The aim is to select the most suitable architectural solution to ensure comfortable indoor environment in the most efficient way in the early building design stage. The design approach takes into account the wind conditions in the region and the building surroundings to evaluate the façade opening distribution impact on natural ventilation performance. The design approach is based on computational fluid dynamics (CFD). In this article, a case study located in the Valencian Community (Spain) is depicted. The Valencian Community coastal climatic conditions are evaluated to assess the low carbon technology energy saving potential. Moreover, the main drivers and barriers involved in the design approach implementation in the region are discussed. The conclusions show that the natural ventilation design approach can improve up to 9.7% the building energy performance respect an initial building design, in which natural ventilation has not been considered. The results contribute to an assessment of the complete low carbon technology effect in the region.

1. Introduction

Over the last 15 years, low carbon technologies to achieve sustainable buildings have become more popular, mainly because of the aim to reduce the greenhouse effect and to save energy and resources. The EU is working to achieve the reduction of greenhouse gas emission by 20% compared with the emission levels of 1990 by 2020. The built environment contributes with almost 35% of the total European CO_2 emissions [1]. Residential buildings represent 77% of the total built environment energy use [1]. Consequently the residential sector represents an important percentage in which CO_2 emissions could be reduced. Therefore, the authorities should push the implementation of low carbon technologies in the residential built environment to meet the targets for sustainable development. Unfortunately, there are still barriers in the national laws that make it difficult.

The present sustainable building design approach is based on natural ventilation (NV). NV is a passive ventilation method based on wind and/or buoyancy pressure differences to refresh air from indoor spaces with outdoor air [2]. The lower energy cost in comparison with mechanical ventilation and the improved indoor environment air quality are the main benefits of NV [3]. Furthermore, many parameters should be analysed carefully to achieve these benefits. NV requires an appropriate understanding of building pressurization, façade design [4], wind patterns and local climate conditions. Accurately predicting and evaluating the NV performance, before and after building construction, is essential to reduce building energy use since the early design stage.

The energy reduction as a result of NV should be considered from the early building design stage in which building location, orientation and envelope have primary impact on building energy performance [5]. Last but not least, wind flow also influences building ventilation, infiltration rates and the associated heat losses or gains. Therefore, building envelope bears much importance in the buildings energy behaviour. The façade performance should be improved according to the surrounding environment conditions and the neighbouring constructions that may modify the local flow causing unexpected flow effects. However, the fact remains that building designers often do not consider it due to the lack of tools and expertise for evaluating and implementing it. Thus, this lack of knowledge in the field may result in poorly designed, constructed and operated naturally ventilated buildings.

Many methods are used to predict ventilation [6]. Among them, computational fluid dynamics (CFD) has been the most popular method for predicting NV nowadays [7–10], especially in studies that cannot be done in laboratories. Thus, CFD empowers building designers to optimize the building solutions by simulation instead of trial-and-error techniques that require much effort, time and resources. Moreover, several standards and guidelines [11–15] have been developed to assist designers in the CFD model definition. The current research is therefore based on CFD to assist building designers to optimize NV building behaviour.

The present design approach is suitable to be used in a regional scale, thereby becoming it in a smart energy region. In any case the “smart energy region” definition is not simple. The definition involves many issues, i.e.,: energy, technology, social, economic and environmental aspects of territory, etc. in a regional scale [16]. Furthermore, some strategies can be common in a regional scale and their global analysis permits designers to have tools for taking decisions when regional conditions are similar.

In this context, the NV design approach is presented through a residential house—case study in the Mediterranean region. Although the design approach is applied in a residential house, it can be used in the whole built environment. The case study analyses the repercussion of distinct façade opening configurations on the indoor airflow distribution to achieve an air sweeping effect to improve the NV building behaviour. The particular Mediterranean coast environmental conditions are analysed. Finally, the energy saving potential of the low carbon design approach used in the future new buildings is evaluated over the complete region.

2. Case Study Description

2.1. Region Description

The research is focused on the Valencian Community Mediterranean coast (Spain). The Valencian Community is located along the Mediterranean Sea coast in the east of the Iberian Peninsula. The most important cities are Valencia, Alicante and Castellon. The Valencian Community has a large mountain range and a thin coastal strip along the whole region. The Iberian mountain range is placed in the inland part of the region. This part is craggy with mountain ranges higher than 1000 m high and mountain peaks higher than 1300 m (*Penyagolosa* 1813 m, *Aitana* 1558 m, *Montcabrer* 1390 m). The temperatures in the inland part are lower than comfort conditions in winter and higher in summer. Oppositely, the rather thin coastal strip is a very plain terrain and the temperatures are more averaged during the whole year due to the Mediterranean Sea effect. Accordingly most of the population lives along the thin coastal strip.

Climatic conditions in Valencian Community coastal zones are also common and particularly interesting to implement construction strategies based on NV. On the one hand, temperatures are averaged during spring and autumn (temperature is near 20 °C). The mean temperatures are little bit higher than 25 °C during July and August and a little bit lower than 15 °C during winter (see Figure 1) [17].

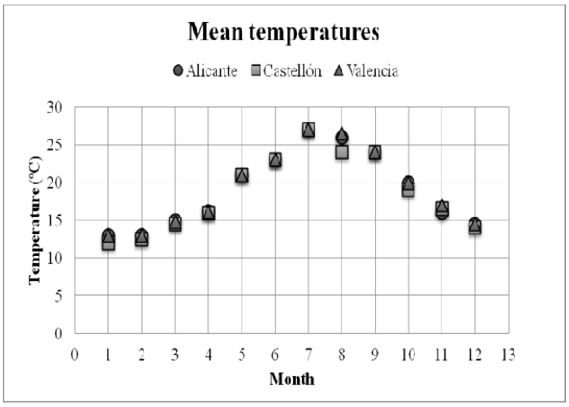


Figure 1. Mean temperature in the three more important cities in Valencian coastal region (IVE).

On the other hand, the sea location has a paramount influence in wind directions. The wind main direction determination is important for the efficient NV implementation as a low carbon technology. In this case, the predominant winds are in the main direction SW-NE in all the coastal regional zones [18]. Consequently, façades opening orientation and distribution analysis will be strongly recommended in the building design stage to ensure and increase NV effect.

The Valencian Community has a population of near 5,000,000 people distributed as shown in Figure 2 [17]. Moreover, the coastal strip is a national and international touristic area, so the population is hugely increased during vacation. The 20% of the residential houses are considered as secondary dwellings [17]. Thus, the potential CO_2 savings in the residential sector acquires a relevant weight during holiday periods (from March to October). Therefore, the design of the buildings placed within the coastal strip has a special interest during spring, summer and autumn time, when the occupancy is high and the indoor temperature might rise above the comfort limit.

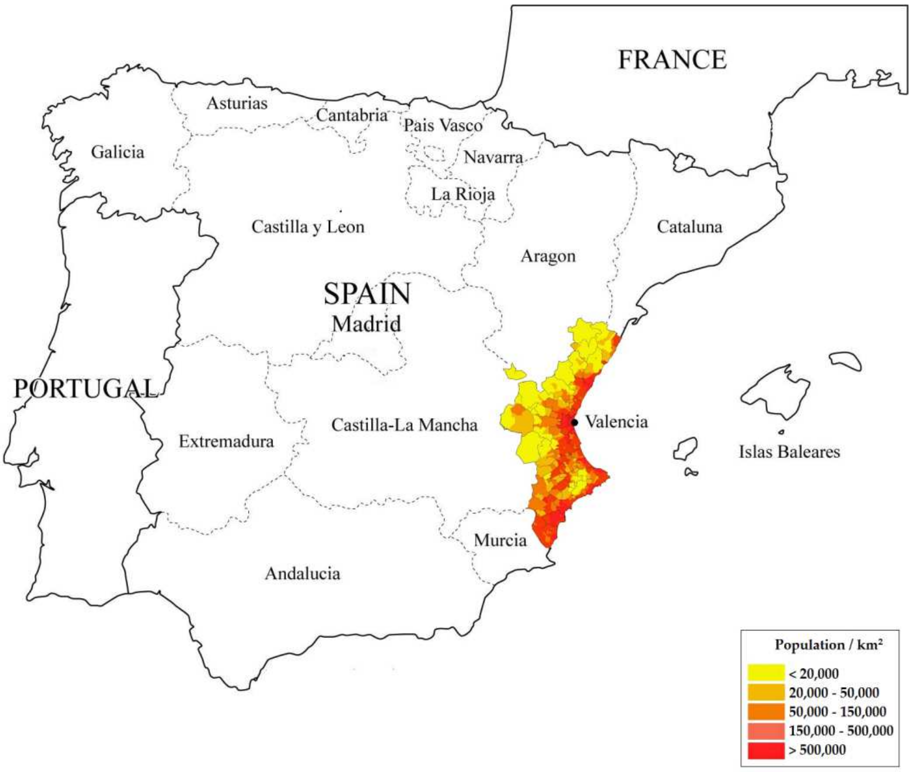


Figure 2. Valencian Community location and population distribution (IVE).

It should be remarked that the complete region is a very high sun-heated area (average of 8.3 h/day) so the building heat gains due to sun complicate the energy efficiency during the cooling period. Figure 3 shows the sun hours along the year [19].

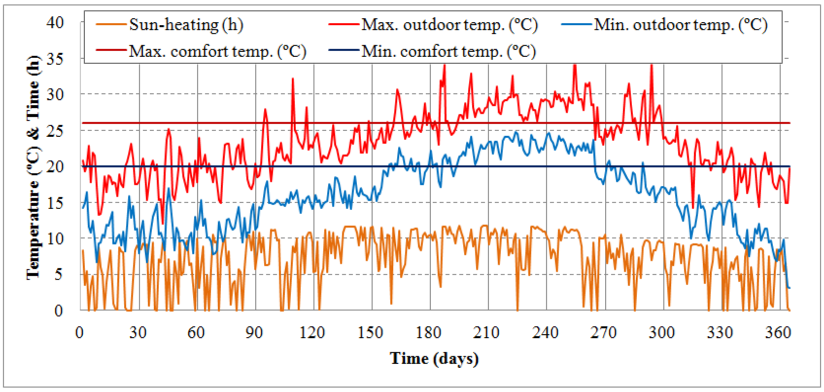


Figure 3. Valencian Community climatic data (AEMET).

Although the sun-heated period is long, the outdoor temperatures are most of the time within the comfort range [19] as shown in Figure 3. These conditions are particularly suitable to maximize the NV use to improve buildings energy efficiency in the region. Thus, a good building design should ensure achieving indoor comfort conditions while having low CO_2 emissions related to the building energy consumption.

The dwellings are growing continuously in the Valencian region. Figure 4 [17] shows the new residential building construction evolution during the last five years. Although there has been a decrease in the number of buildings during some years due to extraordinary economic conditions, the new buildings have been increased until near 1700 constructed buildings during 2015.

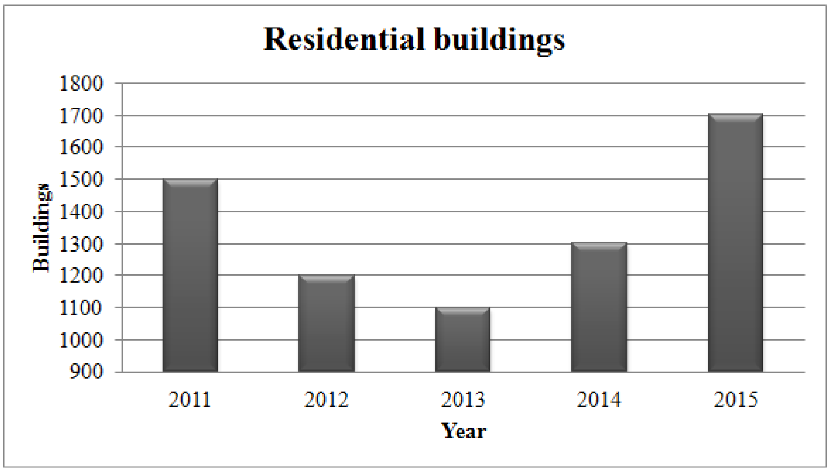


Figure 4. Residential building construction evolution in the Valencian region (IVE).

Multi dwelling, single family houses, detached and semi-detached houses are included in the data shown in Figure 4. Likewise, buildings can be classified into three categories: blocks, semidetached and detached houses. Seventy percent of new houses built during 2015 are detached houses (see Figure 5) [17], the most important new buildings when compared to blocks, row and semidetached houses. Consequently, the case study building selected is a detached house to cover the most probable building typology.

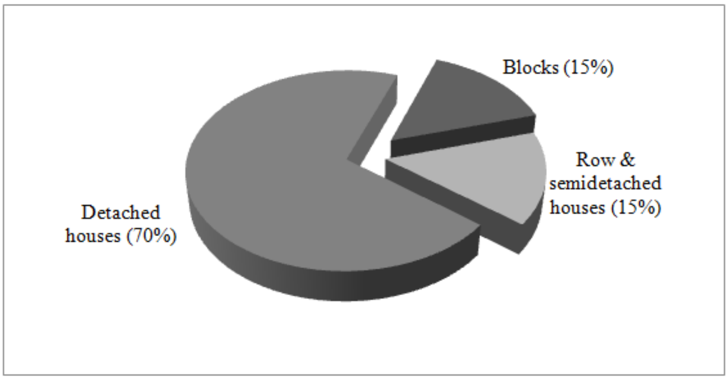


Figure 5. Residential building distribution in the Valencian region (IVE).

2.2. Drivers and Barriers to Natural Ventilation Implementation

The building laws are not too permissive regarding NV in Spain, as the installation of mechanical ventilation is mandatory in most of buildings. The ventilation applicability in buildings is regulated through the rules laid down in the “Código Técnico de la Edificación (CTE)” (building technical rules) [20] and the “Reglamento de Instalaciones Térmicas en los Edificios (RITE)” (thermal building installations regulations) [21].

The residential dwellings, among others, need to comply with the regulation “DB HS3 Calidad de aire interior” (indoor air quality regulation) included in the CTE. A minimum air flow change is required, depending on building dimensions and occupancy. Therefore only mechanical and hybrid ventilation are currently accepted. Oppositely, the RITE allows NV use in some critical scenarios as fire smoke evacuation and/or due to too high CO_2 concentration. In this case, NV systems could work by opening or closing the NV duct gates. In any case, the normal conditions ventilation through building openings or windows is not actually allowed in the Spanish regulation. Conversely, countries such as Denmark use the NV techniques although having a colder climate than Spain [22].

The only NV use is not allowed in order to guarantee the minimum ventilation rate always, even under thermal inversion conditions. The minimum ventilation rate is calculated depending on the outdoor and the target indoor air quality requirement [21]. The problem arises when the bad indoor air quality allowed requires the air to be filtered by a high efficient filter. Consequently the NV use becomes more difficult due to the additional filter pressure loss.

In the end, although the applicable regulation is not too permissive, NV use is allowed if the minimum ventilation rate, the thermal comfort and the indoor air quality are ensured. Hence there should be no problem to use only NV if the building designs are properly done to ensure it. In this sense, the Spanish regulation suggests CFD use to assist building designers to ensure it. Accordingly the building energy efficiency behaviour is increased and the building operation CO_2 emissions are reduced.

2.3. Case Study Building Description

The proposed design approach is conducted through a case study building. The case study building is carefully selected, considering the most common residential houses in the Valencian Community region (see Figure 5). The case study building represents a typical 70 m^2 modular familiar dwelling. The dwelling is a one-floor house (14 m × 5 m), 4 m high located in the Polytechnic University of Valencia, east of Spain. The house is at 39°28′50″ N 0°20′43″ W, close to the Mediterranean Sea. The internal building layout is divided into three rooms connected by a short corridor. The building has some passive components to avoid solar gains like a pergola at the entry and shadows in the windows. Figure 6 shows a render of the prototype building design.

Figure 6. Building prototype render.

The urban place where the house is going to be built is flat with height difference lower than 5 m. 15 m high constructions are build in the adjacent environment. Figure 7a provides a plan view of the location where the building will be placed, and Figure 7b shows a view from the third floor level with a render of the building in the design location (west-east view). In Figure 7, an arrow and a point indicate the position from which the view is shown.

(**a**) (**b**)

Figure 7. View of the building location. (**a**) aerial plan view; (**b**) west-east perspective view.

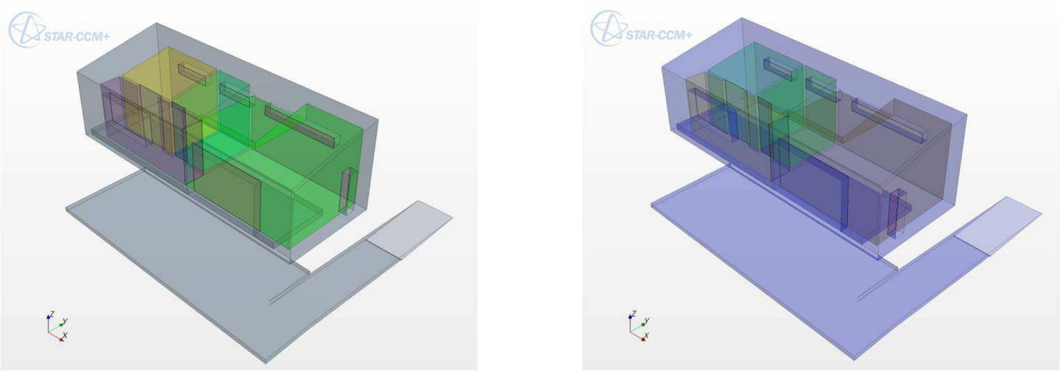
Considering the surroundings where the building is going to be placed, 14 m tall buildings may block winds coming from the south, east, north and west directions (see Figures 7 and 8). According to the relative building position respect to the surroundings and the prevailing wind direction in Valencia, east-south-east (ESE) wind should reach the building already relatively disturbed by the presence of surrounding buildings.

Figure 8. West-east view of the future building location.

The main wind direction strongly affects window orientation for NV design. Therefore the position and size of the openings has to be analysed to provide strong and well-distributed airflow throughout the building. Other aspects are assumed to be constant and already defined (building shape, location, orientation and internal distribution).

The building has six initial outdoor ventilation openings. Four of these openings are windows placed in the upper part of the north façade (0.5 m × 2 m). The south façade has a window (1.5 m × 3.7 m) and a door (2.5 m × 6 m). To improve NV performance, the ESE prevailing wind suggests including an opening in the east building face. This opening allows the prevailing wind flow to go through the east building face, improving the total air change rate (ACR) of the building. Furthermore, a vertical and narrow window may provide a well-distributed airflow throughout the whole building.

The optimal position of a vertical window in the east-lateral face is analysed. Figure 9 graphs the window position alternatives. Three models are compared: Case A has no lateral window, as shown in Figure 6, and Case B and Case C have a vertical lateral window, 0.5 × 1.5 m, in opposite sides of the east façade, as shown in Figure 9.



(**a**) (**b**)

Figure 9. Study cases. (**a**) case B, right window; (**b**) case C, left window.

Although three building opening alternatives are analysed in the current research, the design approach could be extended to other opening position alternatives.

3. Case Study Computational Fluid Dynamics Model

3.1. Geometry Description

The geometry represents the computational domain built area. The geometry should include all things that influence the building NV behaviour. In this case, wind-driven flows allow buildings to improve their NV behaviour. Knowledge of the dominant winds and the effect of nearby buildings on the dwelling are extremely important. The presence of other buildings can produce turbulence trails that may influence the velocity profiles near the building. Consequently, simulating the macro-scale model including surrounding buildings is necessary (see Figures 7 and 8).

The computational model includes both, the case study dwelling and the macro-scale in which the house is going to be placed. Therefore all the effects are considered minimizing the computational error [23]. The entire computational domain is designed following the recommendations given in COST Action 732 [23]. There is a main region in which the wind effect should be studied carefully. This region is modelled in detail according to the design drawings. The neighbouring buildings are modeled explicitly only by its main contour. Because the main objective is the NV behaviour, only the buildings internal and external shapes are considered. Figure 10 shows the four simulated indoor spaces and their layout.

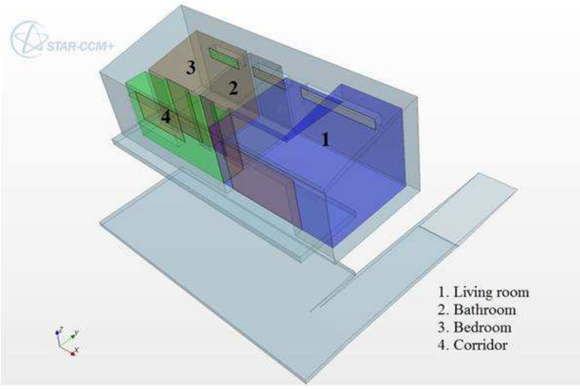
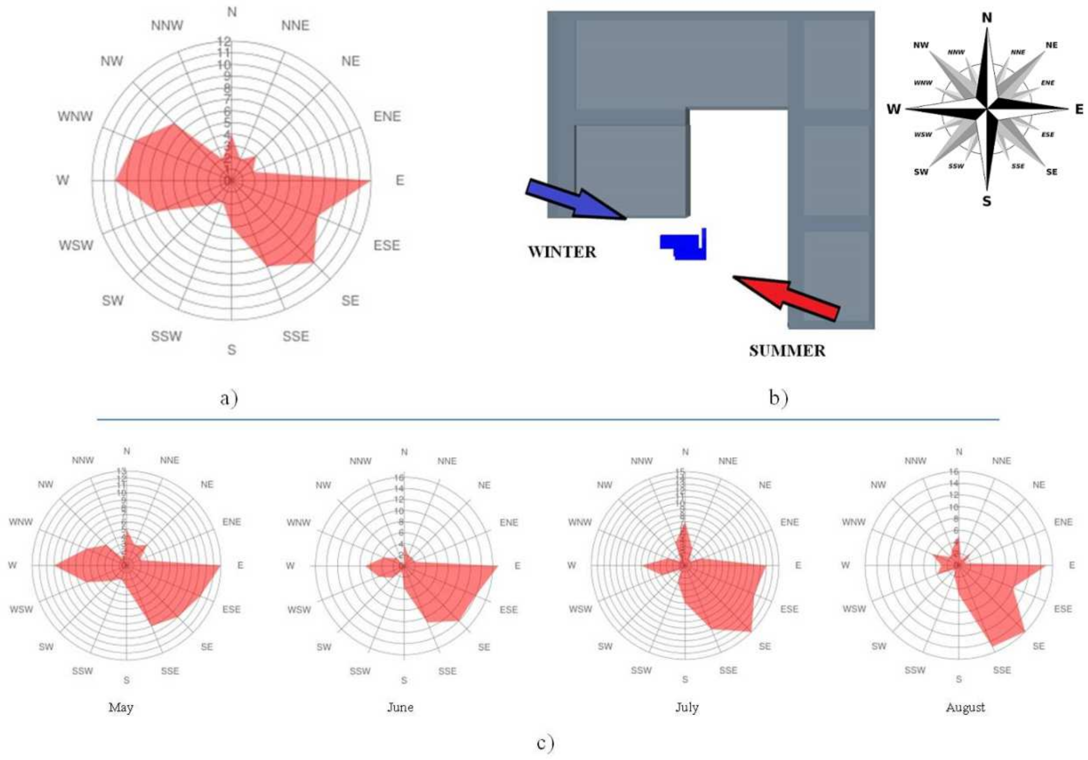


Figure 10. Design building indoor layout.

3.2. Boundary Conditions

Boundary conditions are the surfaces that define the computational domain. Boundary conditions depict the impact of the surroundings that are not included in the model. In this case, wind data is the main boundary condition set in the model. To define the velocity inlet boundary condition, the first stage is to evaluate the wind-driven flows in the most probable condition. A wind rose provides the prevailing wind direction for each particular location [24].

The wind measurements and statistics data are measured by the AEMET Valencia Meteorology Station [25]. The prevailing wind in Valencia in spring, autumn and especially summer is ESE, therefore this situation is used for design, as the most probable one. The frequency of different winds is shown in the wind rose as probabilistic data (see Figure 11). Therefore, the wind boundary condition will be based on this prevailing wind in summer. The logged wind velocities are between 0.0 and 15.8 m/s. These wind velocities are approximately normally distributed, the wind mean velocity is 3.05 m/s and the standard deviation is 1.47 m/s at a height of 10 m during summer period, as analysed from similar studies in regional scale [19,25].



a)

b)

May

June

July

August

c)

Figure 11. Wind rose at Valencia Meteorology Station. (**a**) wind distribution during all year; (**b**) wind distribution where the case study building is located; (**c**) wind distribution at Valencia in summer (windfinder).

Wind data is adapted to be simulated for height [26]. The wind resource should be extrapolated from the available height to a shear model [27]. Justus and Mikhail [28] is used to define the vertical velocity profile using Expression (1).

$$c(z) = c_a \left(\frac{z}{z_a} \right)^n \tag{1}$$

in which $c(z)$ represents the speed at height z (measured in m) and c_a is the average speed at z_a height (measured in m/s). The “n” exponent is defined by Expression (2).

$$n = \frac{0.37 - 0.088 \times \ln(c_a)}{1 - 0.088 \times \ln(z_a / 10)} \tag{2}$$

The exponential velocity profile is set in the ESE direction as inlet velocity in the simulation. The outlet boundary conditions is set as zero relative static pressure based on previous researches [29–31]. The top and lateral model walls are set as symmetry faces to enforce parallel flow. The model lateral walls and ceiling are defined as smooth surfaces. The commercial code Star CCM+ is used as a computational fluid dynamic software to perform the simulations. The conservation of mass and momentum fluid dynamic laws are solved by the numerical code. The Reynolds Averaged Navier-Stokes (RANS) equations are calculated in the target region considering the turbulent approach [32]. The reviewed literature shows the standard k-ε model as the most suitable model for this application [33].

3.3. Validation with Full-Scale Measurements

CFD models must be validated to ensure their reliable results. The CFD validation can be done using several methods [15,34]. This particular case does not allow full-scale indoor measurements to be done, as the building only exists in drawings, and it can still be modified. However, outdoor wind measurements can be performed as well to validate the CFD model. The measurement strategy aims to measure the outdoor conditions by means of installing sensors in some strategic points, as described in the referred literature [10,15,34–36]. Future works will focus on a more detailed validation process, with inside measurements comparisons, when the building is finally constructed.

The outdoor wind velocities are measured using a network of air-speed sensors (HD403TS). The hotwire sensors measure air velocities up to 5 m/s, with an accuracy of ±0.03 m/s. The sensors are located in three horizontal layers (1 m, 2 m, and 3 m) to observe the wind velocity profile in the area of interest. The measurement is done in front of the future east building face where the flow should be similar before and after building construction. Figure 12 shows the corner taken as a reference and the position of the sensor with respect to the future building.

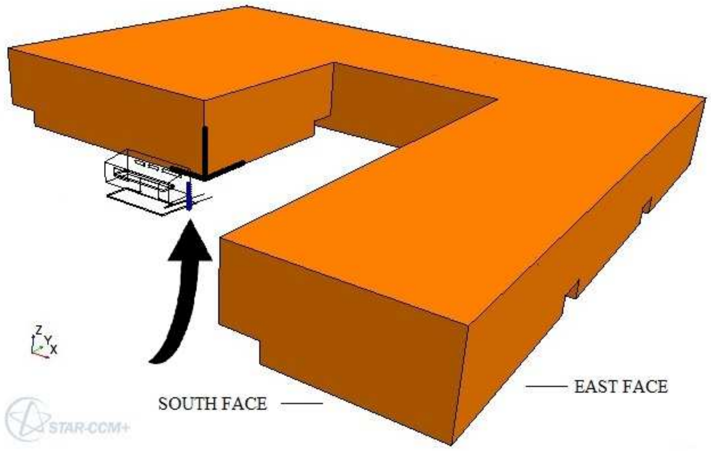


Figure 12. Position of the air-speed transmitters with respect to the surrounding building.

To gather data to validate the numerical model, the measurements are taken on a common ESE windy day. The field wind measurement results mean average over 10 min to 30 min time intervals for a whole day. Longer intervals are not feasible because the outdoor conditions are considered steady for less than 30 min [37]. Then, the steadiest 30 min wind measurements (2.82 m/s ± 0.23 m/s) are selected to validate the numerical results. Table 1 shows the comparison in each position.

Table 1. Full-scale and numerical results comparison.

Position (mm)	**Velocity Sensor (m/s)**	**Numerical Model (m/s)**
(6,596.7; −10,842.3; 1)	1.59 ± 0.12	1.585654
(6,596.7; 10,842.3; 2)	1.39 ± 0.19	1.340761
(6,596.7; −10,842.3; 3)	1.99 ± 0.15	1.986461

The outdoor measurements are in the same range as the CFD predictions. In conclusion, the CFD model is regarded as a reliable model for the design stage building analysis.

4. Results and Discussion

The prevailing wind influence, the façade opening location and the neighbouring buildings influence on the building NV behaviour are assessed. The simulations (see Figure 9) are conducted at three wind speeds to cover the complete wind range. Cases A (no window in the east face), B (right window in the east face) and C (left window in the east face) are compared to determine the architectural solution that best improves the NV behaviour in the building.

The discussion is focused on three issues: the quantitative analysis of the wind flows through the building, the NV behaviour, and the energy saving potential. The velocity inlet boundary condition influence on the CFD results is also discussed.

4.1. Quantitative Analysis of Results

Velocity streamlines are studied to determine the flow distribution through the building. The surrounding buildings change the wind flow incidence angle and turbulences, so the computed streamlines could be quite different than expected. The turbulent nature of the flow is important for NV as a more turbulent flow ensures better replacement. In this case study, the main wind flow direction is modified by the surrounding buildings reaching the building with a south-southeast (SSE) to north-northwest (NNW) direction instead of the typical ESE direction. Focusing on the main room, case A has some dead zones in the north corners of the rooms. In contrast, cases B and C obtain fully mixed air. Figure 13 shows the velocity streamlines passing through the building. The east-facing window originates a low pressure at this opening that helps to mix and drive the air through the dwelling. Consequently, the east face opening improves the indoor air distribution.

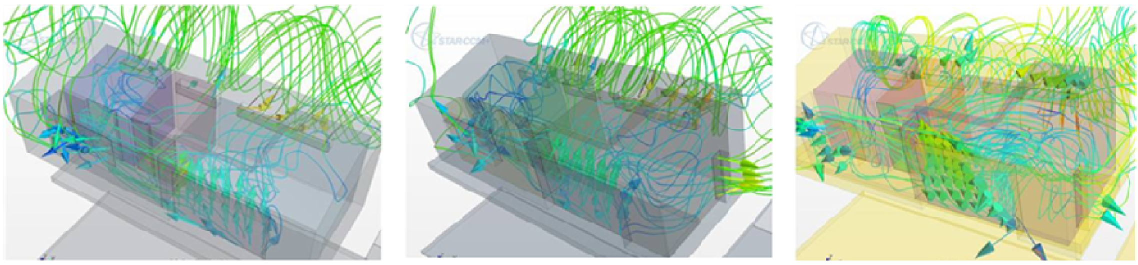

(**a**) (**b**) (**c**)

Figure 13. Main room velocity streamlines visualization. (**a**) Initial case; (**b**) case B, right window; (**c**) case C, left window.

Natural ventilation will be more effective with less time needed for the indoor air to be refreshed. The inlet and outlet air flow rates are used to calculate the effectiveness of wind-driven flows for NV. The objective is to determine how much time it takes for the total indoor air to be replaced using wind-driven flows.

A relative coordinate system is set in each building opening. Air that enters the building (positive velocity value) is taken as an inlet flow and air that leaves the building (negative value) is taken as an outlet flow. The time needed (*t*) to replace all of the internal building air is calculated by dividing the total control volume and the density by the whole mass flow rate, as shown in Expression (3).

$$t = \frac{V \times \rho}{f} \tag{3}$$

where ρ is the fluid density (kg/m^3), f is the mass flow rate (kg/s) and V is the total volume of the building (m^3).

Table 2 shows the time needed to replace the indoor air comparison of the three cases (the wind speed considered is the wind mean velocity 3.05 m/s). Case B (east face right opening) and case C (east face left opening) are 7.60% and 9.77% better than case A (no east face opening) respectively. Case C performs almost 2% better than case B regarding natural ventilation. Therefore, the east face left opening achieves better results than the east face right opening for the prevailing wind condition.

Table 2. Time needed to replace the indoor air improvement.

Case B vs. A	**Case C vs. A**	**Case B vs. C**
+7.60%	+9.77%	−1.97%

4.2. Parametric Analysis of Results

The CFD results accuracy depends on the influence of the input boundary conditions. Therefore, a parametric analysis must be performed, especially when the most important input boundary condition is a probabilistic data that has a standard deviation of approximately 50% of its mean value (see Section 3.2). Because wind is defined in terms of probabilistic data, NV should also be studied under a probabilistically approach and not only examining the mean values of wind velocity modulus and direction but also all of the other probable wind values. Consequently, the complete feasible wind range is selected taking into account the standard deviation of the prevailing nominal wind measurement.

Compared to the nominal wind velocity (v = 3 m/s), when the mean inlet velocity is increased to 4.5 m/s, case B and C still achieves better performance than cases A (+5.07% and +7.63% respectively). However, the relative improvement respect the nominal wind velocity is lower (+5.07% vs. +7.60% and +7.63% vs. +9.77% respectively). When the average wind velocity is decreased to 1.6 m/s, the relative improvement is lower than the nominal wind velocity (+2.38% vs. +7.60% and +2.04% vs. +9.77%). Case B and C behave very similarly with only a 0.33% difference. In conclusion, case C has a relative better performance vs. case B and A. Consequently the design recommendation is to include the proposed window in the left side of the east face of the house. Table 3 shows the NV behaviour comparison of the different cases when the wind modulus is modified.

Table 3. Natural ventilation performance comparison.

Average Wind Velocity	**Case B vs. A**	**Case C vs. A**	**Case B vs. C**
4.5 m/s	+5.07%	+7.63%	−2.38%
3.0 m/s	+7.60%	+9.77%	−1.97%
1.6 m/s	+2.38%	+2.04%	+0.33%

4.3. Energy Saving Potential

The energy saving potential is calculated according to standard 13790, calculation of energy use for space heating and cooling for residential buildings [38]. The Spanish regulations CTE [20] and RITE [21] take the standard 13790 [38] as reference. The monthly quasi-steady-state calculation method is used. Expression (4) shows the general ventilation heat transfer equation used in the Standard 13790, Section 9 [38].

$$Q_{ve} = H_{ve,adj} \times \left(\theta_{\mathrm{int},set,C,z} - \theta_e\right) \times t \tag{4}$$

where $H_{ve,adj}$ is the ventilation global heat transfer coefficient, $\theta_{int,set,C}$ is the indoor temperature, θ_e is the outdoor temperature and t is the time period used for the calculation. The method provided by the standard is particularly suitable for comparison between proposed building designs, especially when the objective is to calculate the various design options influence on the energy use. Insofar as the design options are compared, the calculation error is minimized. The method accuracy depends on the quality of the input data. Particular attention should be paid in the air change rate determination. The present NV approach is focused on the air change rate estimation by

CFD. Accordingly, the method set in the standard 13790 is strongly recommended to be used in the NV design approach.

The supply temperature is the value of the external environment temperature. The cooling demand is calculated during the months in which the average outside temperature is below the maximum comfortable indoor allowed temperature in the warm season and above the minimum required temperature in the cold season according to the UNE-CR 1752 [39] (see Figure 3). Additionally, the wind direction data is taken into account so that is considered only the time in which the prevailing direction (ESE) is flowing. Case A and case C are compared to calculate the relative improvement achieved. The results show that the proposed opening addition (case C) could reduce up to 1.13 kWh/m^2 the energy consumption of the building per annum. It is equivalent approximately to a reduction of 434 grams of CO_2 equivalent per kWh/m^2 of electricity [40] respect the initial building design per annum. The energy behaviour of the complete building has been approximately improved in 4.12%. The here described research is only focused on the prevailing wind conditions. In any case the calculation could be extended to the other wind directions to calculate the complete NV impact on reducing the CO_2 emissions.

The characteristics of the Valencian residential sector have been analysed to calculate approximately the NV design approach impact on the new building construction. The CO_2 emissions potential reduction could be up to 56.5 tons of CO_2 per year, taking into consideration the 2015 new detached houses data (see Section 2). However, the CO_2 potential reduction is very rough since the calculation is only based on the case study presented in this research. The NV design approach potential CO_2 savings is higher because it can be extended to the complete wind directions. Moreover, although the case study is focused on a one-floor modular detached dwelling, the design approach can be extrapolated to other kind of residential buildings. This research is challenging in terms of modelling time and resources. Consequently, the results are just illustrative to quantify the possible NV design approach CO_2 reduction potential applied onto a large-scale scenario.

5. Conclusions and Future Work

The paper shows a design approach for selecting building architectural alternatives to improve NV behaviour using CFD. It has been demonstrated that it is feasible to make comparative results of different façade opening alternatives in an initial design stage to make better NV design decisions to improve the building energy efficiency and reduce the CO_2 emissions.

The residential sector is being shown as one of the major potential sectors to improve energy efficiency and reduce CO_2 emissions. Therefore, the building should be adapted to the region's particular conditions to make as much profit as possible of sustainable energy resources. In this case, the research is focused on the Valencian Community region that is located in the Mediterranean coast of Spain. The local climate and the wind conditions are particular due to the Mediterranean Sea regulatory thermal effect and the topography, respectively.

With the NV design approach since the first building design stage there is a potential saving of 56.5 tons of CO_2 per annum taking into account the new building construction in the Valencian Community. Unfortunately, the main barriers for implementing natural ventilation in external comfortable days come from regional legislation, as only mechanical and hybrid ventilation are generally accepted.

Nevertheless, NV should be considered as a reasonable tool for decreasing CO_2 emissions in Mediterranean regional scale. The impact of the predominant winds and the façades opening position on the buildings in NV performance should be analysed using CFD techniques. The design approach allows selecting the most efficient building alternatives using simulation instead of trial-and-error experimental techniques in the building design stage. Well-designed outdoor openings shape and location can provide naturally ventilated and healthy indoor conditions while significantly reducing the energy consumed by buildings. The study approves the inclusion of a new opening in the east side of the building. This new opening improves the building NV behaviour up to 9.7% (see Table 3).

In future works, the design approach should be extended to the complete wind rose conditions range for the case study building during the whole year. The NV performance assumptions will be validated in the constructed case study building in the future. Moreover, the modular case study results should be adapted to the other building typologies in order to improve the design approach scope.

Acknowledgments: This work has been made possible within the framework of the research project “E3 EDIFICACIÓN ECO EFICIENTE, cofounded by CDTI: Proyectos Tecnológicos de Empresas”.

References

[1] Hamdy, M.; Hasan, A.; Siren, K. Applying a multi-objective optimization approach for design of low-emission cost-effective dwellings. *Build. Environ.* 2011, *46*, 109–123.

[2] Linden, P.F. The fluid mechanics of natural ventilation. *Annu. Rev. Fluid Mech.* 2009, *31*, 201–208.

[3] Luo, Z.; Zhao, J.; Gao, J.; He, L. Estimating natural-ventilation potential considering both thermal comfort and IAQ issues. *Build. Environ.* 2007, *42*, 2289–2298.

[4] Suresh, B.S.; Srikanth, M.; Robert, F.B. Passive building energy savings: A review of building envelope components. *Renew. Sustain. Energy Rev.* 2011, *15*, 3617–3631.

[5] Jin, Q.; Overend, M.; Thompson, P. Towards productivity indicators for performance-based façade design in commercial buildings. *Build. Environ.* 2012, *57*, 271–281.

[6] Li, Y.; Heiselberg, P. Analysis methods for natural and hybrid ventilation—A critical literature review and recent developments. *Int. J. Vent.* 2003, *4*, 3–20.

[7] Martin, L.; James, A.; Heiselberg, P.; Li, Y.; Stathopoulos, T. Achieving natural and hybrid ventilation in practice. *Int. J. Vent.* 2006, *5*, 115–130.

[8] Khan, N.; Su, Y.; Riffat, S.B. A review on wind driven ventilation techniques. *Energy Build.* 2008, *40*, 1586–1604.

[9] Mora-Pérez, M.; López-Patiño, G.; López-Jiménez, P.A. Quantification of ventilated façade effect due to convection in buildings—Buoyancy and wind driven effect. *Res. Appl. Mech. Eng.* 2014, *3*, 1–11.

[10] Ray, S.D.; Gong, N.W.; Glicksman, L.R.; Paradiso, J.A. Experimental characterization of full-scale naturally ventilated atrium and validation of CFD simulations. *Energy Build.* 2014, *69*, 285–291.

[11] Chen, Q.; Zhai, Z. The use of computational fluid dynamics tools for indoor environmental design. In *Advanced Building Simulation*; Taylor & Francis: Oxfordshire, UK, 2003; pp. 119–140.

[12] Tominaga, Y.; Mochida, A.; Yoshie, R.; Kataoka, H.; Nozu, T.; Yoshikawa, M. AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. *J. Wind Eng. Ind. Aerodyn.* 2008, *96*, 1749–1761.

[13] Norris, S.E. Appropriate boundary conditions for computational wind engineering in models revisited. In Proceedings of the Fifth International Symposium on Computational Wind Engineering (CWE2010), Chapel Hill, NC, USA, 23–27 May 2010; pp. 23–27.

[14] Hajdukievwicz, M.; Geron, M.; Keane, M.M. Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room. *Build. Environ.* 2013, *70*, 73–89.

[15] Hajdukievwicz, M.; Geron, M.; Keane, M.M. Formal calibration methodology for CFD models of naturally ventilated indoor environments. *Build. Environ.* 2013, *59*, 290–302.

[16] Marsa-Maestre, I.; Lopez-Carmona, M.A.; Velasco, J.R.; Navarro, A. Mobile agents for service personalization in smart environments. *J. Netw.* 2008, *3*, 30–41.

[17] Instituto Valenciano de la Edificación (IVE). Generalitat Valenciana. Available online: http://www.ive.es (accessed on 8th February 2016).

[18] Pérez-Cueva, A. *Atlas Climático de la Comunidad Valenciana*; Generalitat Valenciana: Valencia, Spain, 1993.

[19] Agencia Estatal de Meteorología de España (AEMET). Available online: http://www.aemet.es (accessed on 8th February 2016).

[20] Código Técnico de la Edificación (CTE). Ministerio de Fomento, Gobierno de España. Available online: http://www.codigotecnico.org/ (accessed on 15th April 2015).

[21] Reglamento de Instalaciones Térmicas en los Edificios (RITE). Ministerio de Industria, Energía y Turismo, Gobierno de España. Available online: http://www.minetur.gob.es/energia/desarrollo/EficienciaEnergetica/RITE/Paginas/InstalacionesTermicas.aspx (accessed on 15th April 2015).

[22] Oropeza-Perez, I.; Ostergaard, P.A. Potential of natural ventilation in temperate countries—A case study of Denmark. *Appl. Energy* 2014, *114*, 520–530.

[23] Franke, J.; Hellsten, A.; Schlünzen, H.; Carissimo, B. *Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment*; COST Action: Brussels, Belgium, 2007.

[24] Reboussin, D. *Wind Rose*; University of Florida: Gainesville, FL, USA, 2005.

[25] Windfinder. Available online: http://es.windfinder.com/windstats/windstatistic_valencia.htm (accessed on 15th April 2015)

[26] Joseph, H.; Willian, D.; Warren, H. *Wind Energy Resource Assessment Using Wind Atlas and Meteorological Data*; Stiver School of Engineering, University of Guelph: Guelph, Ontario, USA, 2008.

[27] Giovanni, G.; Sauro, S. Methods to extrapolate wind resource to the turbine hub height based on power laws: A 1-h wind speed vs. Weibull distribution extrapolation comparison. *Renew. Energy* 2012, *43*, 183–200.

[28] Justus, C.G.; Mikhail, A. Height variation of wind speed and wind distributions statistics. *Geophys. Res. Lett.* 1976, *3*, 261–264.

[29] Senthooran, S.; Lee, D.D.; Parameswaran, S. A computational model to calculate the flow-induced pressure fluctuations on buildings. *J. Wind Eng. Ind. Aerodyn.* 2004, *92*, 1131–1145.

[30] Hu, C.H., Wang, F. Using a CFD approach for the study of street-level winds in a built-up area. *Build. Environ.* 2005, *40*, 617–631.

[31] Evola, G.; Popov, V. Computational analysis of wind driven natural ventilation in buildings. *Energy Build.* 2006, *38*, 491–501.

[32] Wang, L.; Wong, N.H. The impacts of ventilation strategies and façade on indoor thermal environment for naturally ventilated residential buildings in Singapore. *Build. Environ.* 2007, *42*, 4006–4015.

[33] Shao, J.; Liu, J.; Zhao, J. Evaluation of various non-linear k-epsilon models for predicting wind flow around an isolated high-rise building within the surface boundary layer. *Build. Environ.* 2012, *57*, 145–155.

[34] Chen, Q. Ventilation performance prediction for buildings: A method overview and recent applications. *Build. Environ.* 2009, *44*, 848–858.

[35] Van Hooff T, Blocken B. On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium. *Comput. Fluids* 2010, *39*, 1146–1155.

[36] James Lo, L.; Novoselac, A. Cross ventilation with small openings: Measurements in a multi-zone test building. *Build. Environ.* 2012, *57*, 377–386.

[37] Franke, J.; Hirsch, C.; Jensen, A.G.; Krus, H.W.; Schatzmann, M.; Westbury, P.S.; Miles, S.D.; Wisse, J.A.; Wright, N.G. Recommendations on the use of CFD in wind engineering. In Proceedings of the International Conference Urban Wind Engineering and Building Aerodynamics 2004, Rhode-Saint-Genèse, Belgium, 5–7 May *2004*.

[38] European Technical Standard EN ISO 13790. *Calculation of Energy Use for Space Heating and Cooling*, 1st ed.; CEN: Brussels, Belgium, 2008. (http://www.iso.org)

[39] Comité técnico AEN/CTN 100. UNE-CR 1752:2008. *Ventilation for Buildings—Design Criteria for the Indoor Environment*; Aenor, Madrid, Spain, 2008. (http://www.aenor.es)

[40] Instituto para la Diversificación y Ahorro de Energía (IDAE). *Factores de Emisión de* CO_2 *y Coeficientes de Paso a Energía Primaría de Diferentes Fuentes de Energía Final Consumidas en el Sector de Edificios en España*, Version 20/07/2014.; Ministerio de Industria, Energía y Turismo y Ministerio de Fomento: Madrid, Spain, 2014.

(http://www.minetur.gob.es/energia/desarrollo/EficienciaEnergetica/RITE/Reconocidos/.) (accessed on 8th February 2016)

Annex III

Quantification of ventilated façade efficiency by using computational fluid mechanics techniques

Abstract. In many countries, summer over-heating becomes the major energy consuming source in buildings. Ventilated façades could become a tool for sustainable design, especially in bioclimatic building design. A ventilated façade is a multi-layer structural solution based on the combination of ceramic panels with air chambers in order to improve energy savings under some climatic conditions, especially during the warm season. The objective of this paper is to quantify the ventilated façade thermal behaviour contribution to the energy efficiency of buildings. The energy improvement is done by the air movement due to convection in the façade air layer. The convective air movement is linked to the heat exchange through the façade panels. The quantification is mathematically modelled by Computational Fluid Dynamics (CFD) using a commercial code: STAR CCM+. The proposed methodology allows an estimation of the energy potential saving of the ventilated façade regarding its capacity for cooling the building under certain conditions.

Keywords: Ceramic ventilated façade, computational fluid dynamics.

1. Introduction

The current energy scenario is increasingly focused on sustainable design strategies. Engineers and architects are being encouragement to propose constructive strategies that take advantage of available resources and reduce energy consumption in buildings to improve its energy performance. In this sense, the ventilated ceramic façade is becoming one of the most promising strategies. This façade typology has a prominent role in the techniques of maximizing the building envelope usage taking into account the convective ventilation principles. Moreover, it has a great potential in the future sustainability strategies applied in edification.

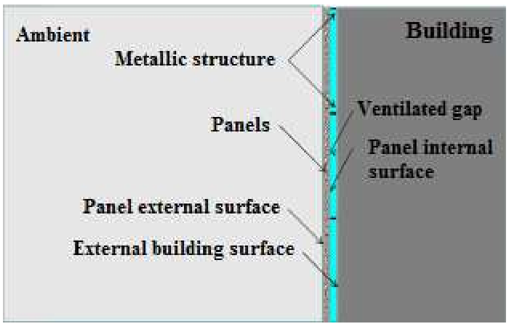


Figure 1. Ventilated façade lateral view

The ventilated façade is a complex multilayer structure system that allows an external ceramic panel installation. The panel is installed far enough from the building's external wall in order to allow air to move between the panel and the wall. The main objective is to avoid heat entering the building and to remove heat from the indoor volume (see Figure 1). The ventilated façade is a useful tool for the bioclimatic architecture.

However, although the usefulness of the ceramic panel ventilated façade is known empirically, the quantification of its effect on the building is still in the development phase. The present research proposes a parameterization strategy of the façade efficiency by analysing the temperature variation that occurs in the space where the air is moving between the ceramic panel and the building. The temperature variation can be quantified and then, the cooling power reduction of the indoor environment in contact with the façade can be estimated.

Initially, ventilated ceramic façades were designed to protect buildings against a combined action of rain and wind. Moreover, the aim is to take advantage of the aesthetic and cooling convection benefits of the flow within the ventilated façade air layer. The different material layers of a ventilated façade are:

Externally placed ceramic panels: this layer defines the esthetical appearance of the building. It has a fundamental role from the structural and esthetical point of view. It also affects the air convection in the interior, since its thermal transmissivity will influence the temperature and heat transferred.

- Air layer: the thickness of this layer should be at least 3 cm (according to sanitary regulations in Spain), with a maximum of 10 cm. This layer aim is to allow the circulation of an airflow in order to remove moisture and evacuate heat.
- Metal structure: it is traditionally made of aluminium and it is designed to hold and give mechanical resistance and stability to the ceramic material separated from the wall at the proper distance.
- Besides the esthetical possibilities, the proper colocation of these materials should improve thermal building conditions mainly in summer as well as in winter. [1].

The air movement dynamics as a consequence of the convective effect in the chamber has to be analysed, especially when considering the cooling effect of the building face in contact with the air chamber. This air movement dynamics will improve the cooling effect, avoiding some heating energy to be transferred into the building. The chamber air movement analysis requires the deep knowledge of the fluid flow mechanics equations. This is achieved by the use of computational fluid dynamic (CFD) techniques. The technique allows the movement and energy conservation equations solution for the fluid behaviour in the chamber. Additionally, it is used for the visualization and quantification of the fluid behaviour in the region of interest. The CFD techniques have been widely used to solve the fluid movement equations in some media using numerical methods with great success, as contrasted in [2-4] during the last decades.

2. Mathematical model of the air movement in the ventilated façade chamber

2.1. Modelling geometry

The air movement in the air chamber of the ventilated façade and the evolution of its temperatures should be known through the mathematical model development. The particular case simulates a 6 m wide and 7 m high building with a 40 mm air chamber thickness. The aim is to know the effect of the air entering the chamber from the front of the building by means of the flow equations solution.

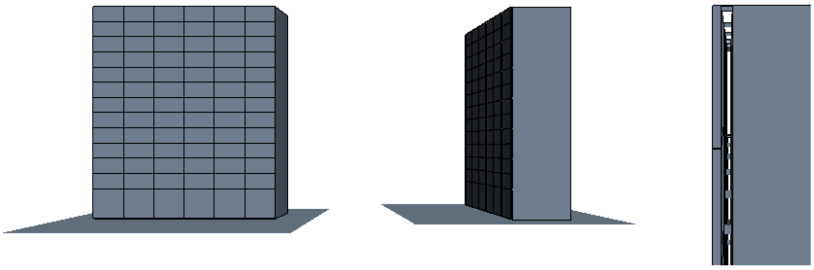


Figure 2. Façade model for simulating the air in the ventilated chamber

Figure 3 shows the 40mm width ventilated chamber that is simulated in this case.

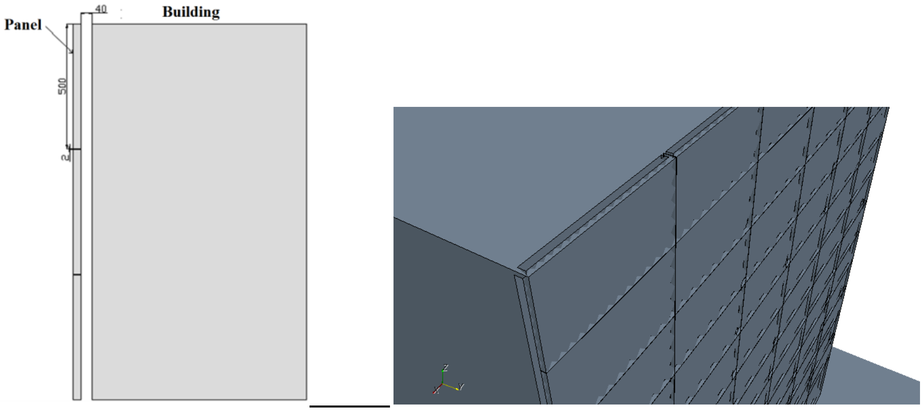


Figure 3. Façade geometric detail for the CFD model

2.2. Ventilated façade mathematical model.

CFD has been widely used since 1970 for the air movement simulation through buildings. However, this technique has not been used for modelling ventilated façades. The present paper aims to present the CFD as a powerful tool to quantify the capacity that the convective flow created in the ventilated chamber has to improve the energy savings regarding air conditioning in buildings.

A commercial software (STARCCM+) is used for modelling the fluid movement and for representing the temperature variation of the air that is moved within the ventilated façade. The numerical code solves the fluid mechanics equations in a defined domain. The domain represents the fluid volume that should be simulated. Then, boundary conditions define the domain that is divided into small cells creating a mesh. The objective is to converge the solution and to calculate and allow the visualization of the temperature, pressure and velocity flow profiles in the studied region.

The finite volume method (FVM) is used to solve the fluid mechanics equations by the numerical code. The FVM allows the differential equations to be discretised and numerically solved in a given domain. Control volumes are built around each point of the domain mesh so that each control volume does not overlap with the neighbouring points. Therefore the final volume is equal to the sum of all control volumes. The differential equation to be solved is integrated on each control volume, which results in a discretised version of this equation. The equation is then solved numerically.

The modelling method assumes that the total flow that enters or leaves a cell in the mesh through its contours is maintained. This feature means that the net flow through the entrances and exits is unique, although flow sources and sinks can also be modelled. Moreover, the numerical code can show the complex differential equation solution that represents the flow in each cell. However, an average value of the calculated property is associated to each point, then the differential equation that represents it is integrated by the code. Thus, this procedure creates a set of linear equations. The equation system is solved to obtain the properties distribution in all the nodes. In this case, velocities, pressures and temperatures are assessed.

The advantage of using these models lies in the fact that they will reproduce real problems of fluid mechanics difficult to be known otherwise. Actually, the visualization of hydrodynamic aspects difficult to be measured or represented in real cases such as stream lines is allowed. Moreover, these features are often important for the understanding of the phenomenon being studied.

The FVM is used to solve the dynamic fluid equations in the mesh generated by the commercial code. The turbulent model used is the conventional k-ε. The model shows good convergence with 600 iterations.

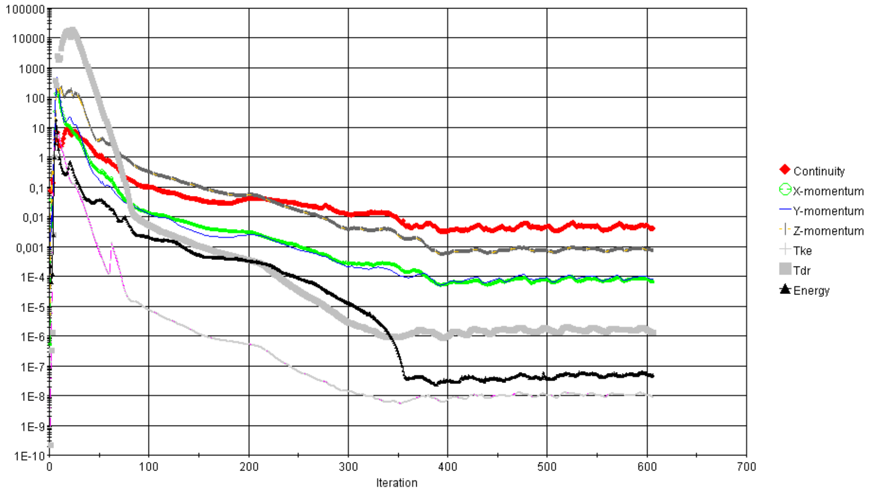


Figura 4. Residuals convergention of the model used by the code

The Boussinesq approximation for buoyancy-driven flows is considered. The flow is considered incompressible i.e. the fluid density is constant. Nevertheless the only density variation considered is due to the gravitational force, which is included in the equation of momentum conservation.

The model is considered steady, turbulent and Newtonian with steady physical properties. The averaged variables and steady-state equation for the mass conservation (continuity) (1), momentum (2) and energy (3) can be expressed as follows [5]:

$$\frac{\partial}{\partial x_j} u_j = 0 \tag{1}$$

$$\rho \frac{\partial}{\partial x_j} u_i u_j = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \mu \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{\partial}{\partial x_j} \rho (\overline{u'_i u'_j}) + F_i \tag{2}$$

$$\rho C_p \frac{\partial}{\partial x_j} T u_j = \frac{\partial}{\partial x_j} \left(-k \frac{\partial T}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \rho C_p \left(\overline{u'_j T'} \right) \tag{3}$$

where u$_i$, T and p are averaged values and u'_j and T' are velocity fluctuant components and instantaneous temperatures respectively. The average and fluctuating velocities are defined as shown in (4), where υ_i represents an instant value.

$$u_i = \frac{1}{\Delta t} \int_t^{t+\Delta t} \upsilon_i dt \qquad u'_i = \upsilon_i - u_i \tag{4}$$

The following expressions within bracket $(\overline{u'_i u'_j})$ and $\rho C_p \left(\overline{u'_j T'} \right)$ represents both fluctuant average quantities. It represents the Reynolds stress and the additional flow due to turbulent phenomena. F$_i$ is the vertical strength component due to density differences between flows at different temperatures. Moreover Reynold stress is related to average values by means of the Boussinesq hypothesis:

$$\rho (\overline{u'_i u'_j}) = \frac{2}{3} \rho k_e \delta_{i,j} - \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \tag{5}$$

where k_e represents the kinetic turbulent energy defined as:

$$k_e = \frac{1}{2} (\overline{u'_i u'_j}) \tag{6}$$

The heat flow due to equation (3) turbulence is also represented by the turbulent viscosity in equation (7)

$$\rho C_p \left(\overline{u'_j T'} \right) = k_t \left(\frac{\partial T_i}{\partial x_i} \right) \tag{7}$$

where μ_t y k_t are flow properties connected by the Prandtl number. The reviewed references proposes a Prandtl number of 0.9 [6].

In this case, all conditions and properties are defined using the commercial software STAR-CCM+. The corresponding equations are solved by the commercial numerical code. Moreover, the results are presented taking advantage of the post-processing techniques of the commercial software. The segregated resolution model is used. It solves the flow equations (one for each component of the velocity and pressure) in a decoupled method. The connection between continuity and momentum conservation equations is done by a predictor-corrector approximation in the finite volumes defined for the equation numerical solution. This model should be used in non-compressive fluids as in the present case. Opposite to the segregated method is the coupled resolution method in which the conservation equations are solved simultaneously, without using approximations. Therefore this method requires the solution of large systems of simultaneously coupled equations.

The gravity model is mandatory in this case as it permits the inclusion of the flotation phenomena in the vertical fluid movement. The Boussinesq approximation is used in this case.

Turbulence should be properly selected with a suitable enclosure approach. Thereby the most suitable enclosure is the k-ε. This is a two equation model in which two parameters responsible for closing the equations should be defined: the turbulent kinetic energy and the dissipation of the turbulent kinetic energy. The reviewed literature and the convergence degree reached by the turbulence residuals in the simulation justify the use of this turbulence model respect other models considered. The physics of the problem allows also its use.

Several models can be taken into account when dealing with the closure problem [7-10]:

- One equation models. In this case a single differential equation is solve for *k* i.e. the Spalart-Allmaras method.
- Two equation models. In addition to the above described model for k, a further equation must be solved to provide the turbulent kinetic energy dissipation rate ε, and the specific dissipation rate ω = ε / k, which is the variable that determines the turbulence scale [9]. k-ε models and their variances are suitable to represent this case. In particular, one of the advantages of the k-ω model versus k-ε is a best behaviour of the simulations in the boundary layer, even under adverse pressure gradients. However, the k-ε model presents an acceptable convergence capability especially with not too high pressure variations.
- Reynolds stress models (RSM). There are models known also as DSM (differential, second-moment turbulence models). These models have a higher accuracy level regarding their closure solution. In this paper, the averaged turbulent equations are not considered as done with the previous ones with one or two equations with new unknowns. The model is more complex since it is intended to establish a differential equation for each of the Reynolds stress: production, transportation, diffusion and rotation features are modelled. Thus, these models could initially be more suitable for processes in which rotary features, strong curvature of flow lines or secondary and anisotropic flows are done. The computational cost is the main disadvantage in comparison with the traditional model. Seven equations should be solved in three-dimensional models while five equations should be solved simultaneously in two-

dimensional [11-13]. In this case, the flow is directional so this closure solution is discarded.

Consequently, the k-ε model considers a turbulent viscosity distribution obtained as:

$$\mu_t = \rho C_\mu \left(\frac{k_e}{\varepsilon} \right) \tag{8}$$

where $C_\mu = 0{,}09$ [14].

The kinetic turbulen energy k_e and its dissipation ε are known when their classic equations are solved [14].

$$\frac{\partial}{\partial x_j} \rho u_j k_e = \frac{\partial}{\partial x_j} \frac{\mu_t}{\sigma_k} \frac{\partial k_e}{\partial x_j} + \left(-\rho(\overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} + \beta g_i \frac{\mu_t}{\mathrm{Pr}_t} \frac{\partial T}{\partial x_i} - \rho \varepsilon \right) \tag{9}$$

$$\frac{\partial}{\partial x_j} \rho u_j \varepsilon = \frac{\partial}{\partial x_j} \frac{\mu_t}{\sigma_\varepsilon} \frac{\partial \varepsilon}{\partial x_j} - C_{\varepsilon 1} \rho(\overline{u'_i u'_j} \frac{\partial u_j}{\partial x_i} \frac{\varepsilon}{k_e} - C_{\varepsilon 2} \rho \frac{\varepsilon^2}{k_e} \tag{10}$$

The values used for the constant parameters are the ones proposed in the literature:

$C_{\varepsilon 1} = 1{,}44$; $C_{\varepsilon 2} = 1{,}92$; $\sigma_k = 1{,}0$; $\sigma_\varepsilon = 1{,}3$

2.3 Applied methodology for the mathematical model creation

The objective of the present research is to quantify the ventilated façade effect on the cooling energy saving potential of the indoor room next to the ventilated façade.

Thus, a heat analysis is done in the volumes near the façade. The aim is to compare the temperature differences of the indoor wall in two cases, with and without a ventilated façade installed in the other side of the wall. Figure 4 shows the geometry without ventilated façade that is used in the simulation. The insulation material effect has not been considered since the research is based on a comparison and the effect of the insulation material should be the same in both cases (with and without ventilated façade). Moreover it will be assessed whether the indoor insulation material could be removed thanks to the contribution of the ventilated façade to energy efficiency.

2.3.1. Simplifications

The objective of this contribution is to show the CFD capability for modelling ventilated façade features in order to provide a tool to understand the role played by the ceramic ventilated façade in the possible energy improvements and cooling savings in the indoor. Some simplifications are made when comparing the ventilated and non-ventilated façade:

a) The temperature of the environment air and the external surface of the ventilated façade panel are considered the same. This temperature may depend on the surface material and features such as convection, radiation, the colour of the surface, the geographical position and the orientation respect the sun, the wind, the humidity or the possibility to be wet by the rain, etc. These aspects are simplified and assumed constant in the simulation.

b) The thermal inertia of the different material layers has not been considered. Thermal inertia is the physical property that indicates the speed at which the material absorbs and keeps energy from the environment. It depends on the mass, the specific heat, the thermal conductivity coefficient, etc. The parameters of the indoor wall are not modified neither in the model with and without ventilated façade because the wall remains the same. Although it is known that thermal inertia is important in different external enclosures, the present research does also not consider the ceramic façade thermal inertia.

c) The simulation is a steady state simulation and the temperature boundary conditions are selected according to measurement in the ventilated façade during the warm season. The year seasons and the changes motivated by the day/night have not been considered. Moreover the steady state condition does not allow the thermal inertia consideration. Actually this simplification is important since it will strongly affect the savings of the ventilated façade.

d) It is considered that the solar effect is able to maintain constant the temperature of the outdoor ceramic panel surface for the complete height.

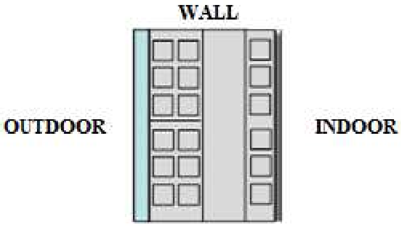


Figure 4. Wall considered in the modellation

Figure 5 represents the temperature nomenclature considered in each surface. Case a) is the reference case without ventilated façade and case b) includes the external panels that makes the ventilated façade. Firstly, the air temperature (T_{air}) is set at 298K (25 °C). The ceramic panel internal surface temperature (T_1) is measured in field test and set at 304 K (31ºC). The indoor temperature (Tin) of the indoor surface in direct contact with the façade is set at 296 K (23ºC). Then, the temperature of the outdoor building surface (T_2) is calculated by the CFD software.

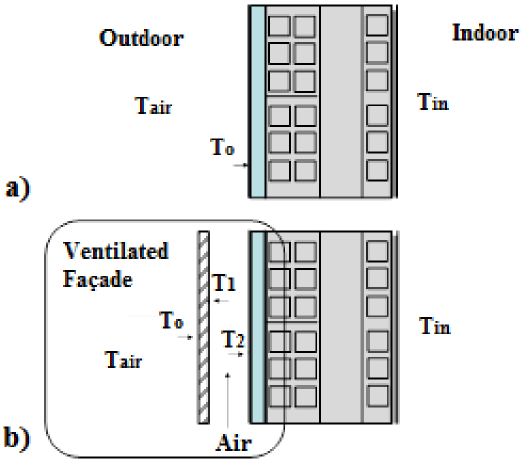


Figure 5. Temperatures considered in the model

2.4. Simulation results: velocity and temperature fields

The CFD software calculates the air velocity in the ventilated façade chamber taking into account the wind speed near the building envelope and the convective effect due to temperature differences. The wind direction is perpendicular to the façade and the air velocities in the chamber are calculated by the numerical code. Figure 6 shows the velocity profile in a section centred on the third column of panels from the left when $T_1 = 304K$. It can be observed that the velocity vectors tend to be increased (get darken in the figure) as velocity vectors goes towards the top of the building.

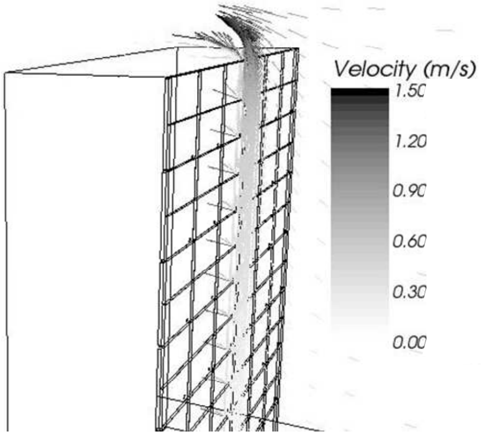
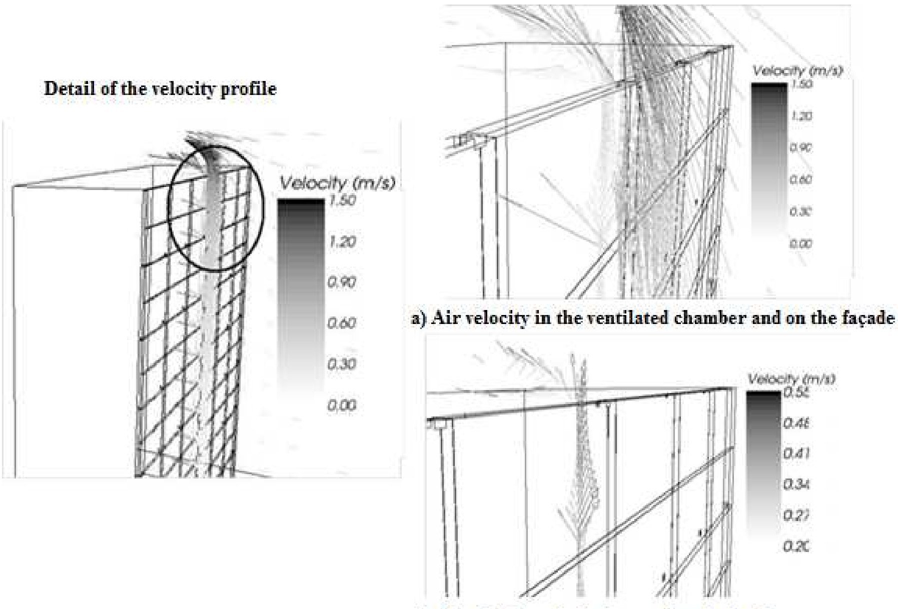


Figure 6. Velocity profile shown in a transversal section (m/s)

Wind can enter the ventilated façade through the slots existing between the panels and from the low part of the façade. The slots between the panels are 2 mm width. The wind does not enter through the lateral openings because the wind is defined perpendicular to the façade panels.

Figure 7 shows in more detail the velocity profile in a central area of the ventilated façade, where the laterals have no influence. Figure 7 a) shows the velocity in a region near the façade in which the wind enters the chamber through the slots. It also shows the velocity profile in the chamber between the panels of the ventilated façade. Figure 7 b) shows only the velocity field in the ventilated chamber. The air direction is mainly upwards.



a) Air velocity in the ventilated chamber and on the façade

b) Air velocity only in the ventilated chamber

Figure 7. Detail of the velocity profile

These velocities depend on temperature T_1. When T_1 is increased, the convective effect also increases the air velocity in the ventilated chamber. Figure 8 shows how the air velocity modulus is increased while increasing T_1 (it is represented in a central line of the ventilated façade chamber section).

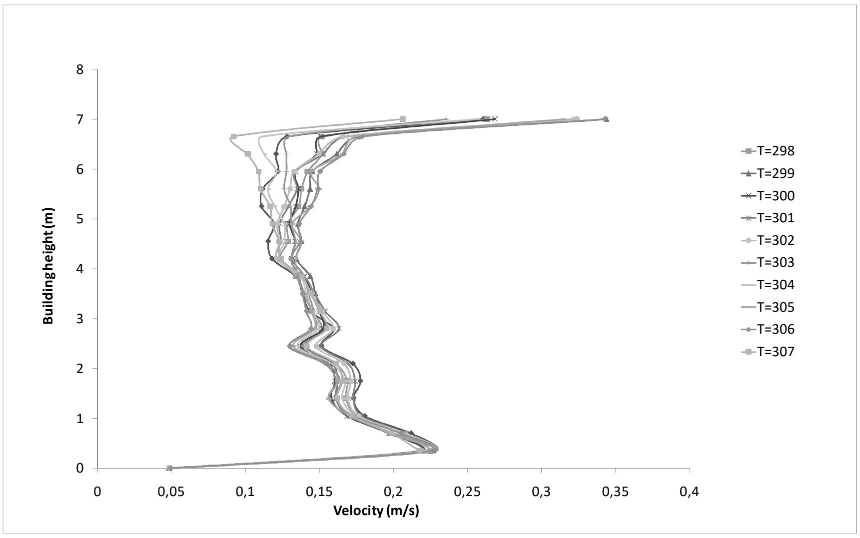


Figure 8. Velocity modulus for different T_1.

T_2 is calculated by the numerical code considering the set temperatures and the simulated wind perpendicular to the façade. Temperature T_2 directly depends on the ventilated façade cooling capacity efficiency. Moreover, T_2 has direct influence on the indoor temperature. This temperature is show in a frontal building view in figure 9.

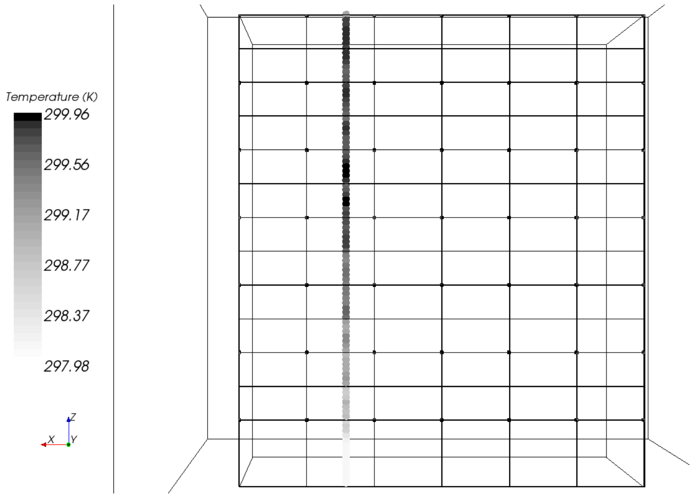


Figure 9. T_2 temperature profile in the external building wall protected by the ceramic panels (not shown)

The results show that the temperature in the ventilated chamber is decreased while the outdoor ceramic panel face temperature is kept constant: $T_1 = 304$ K. The average T_2 decrease is 4.482K (in the ventilated chamber), considering that temperature increases as soon as the air goes upwards within the façade, as shown in Figure 10.

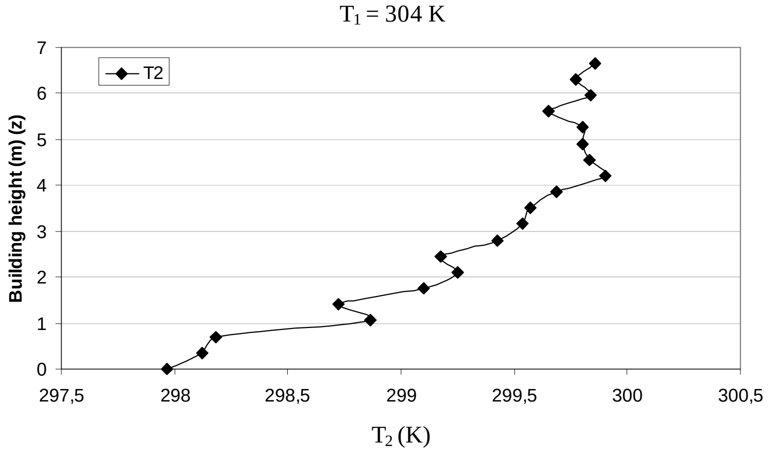


Figure 10. T_2 vertical profile of the case study building

The temperature decrease on the external face of the wall in contact with the ventilated chamber (T_2) means a cooling energy reduction in the indoor building.

The research developed by Ciampi [15] concludes that the outside air temperature is one of the parameters that more influence the ventilated façade efficiency. In summer conditions, the energy savings will increase considerably according to solar radiation increase. Therefore, not only the convective air movement will be increased but also the cooling capacity of the air through the ventilated chamber. This feature is simulated with the CFD because it is considered important. Figure 11 shows the temperature variation in the ventilated chamber (difference between T_1 and T_2) when T_1 is increased as result of the external ceramic panel heating. T_1 is fixed in each simulation and it is set at different temperature levels and T_2 is calculated for each supposed T_1. Figure 11 shows that when the outside panel temperature is increased, the ventilated chamber temperature difference (T_1-T_2) is higher. Thereby, the passive cooling effect generated by the ventilated façade is more effective.

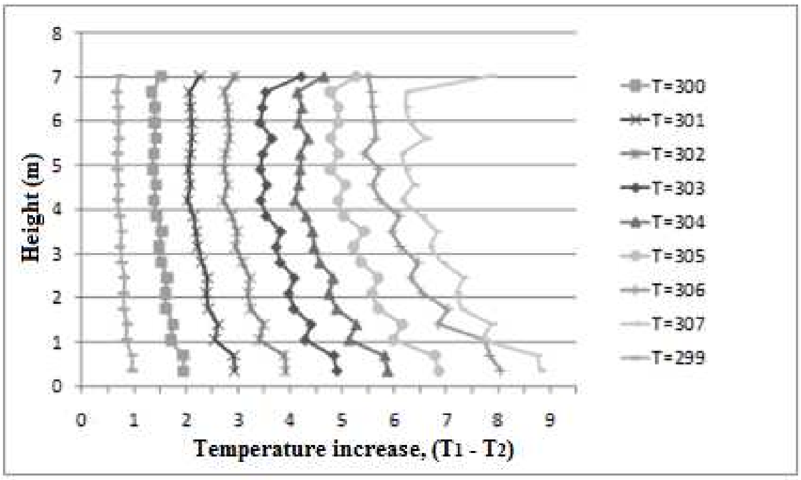


Figure 11. Ventilated gap temperature variation simulated by the CFD when T_1 is changed

It is observed that the cooling effect of the air in the ventilated façade chamber has a linear behaviour respect the temperature increase of the ventilated panels due to the environmental conditions simulated. Figure 12 shows the average temperature decrease in the ventilated chamber while the external panel temperature is increased similarly than in the figure 11.

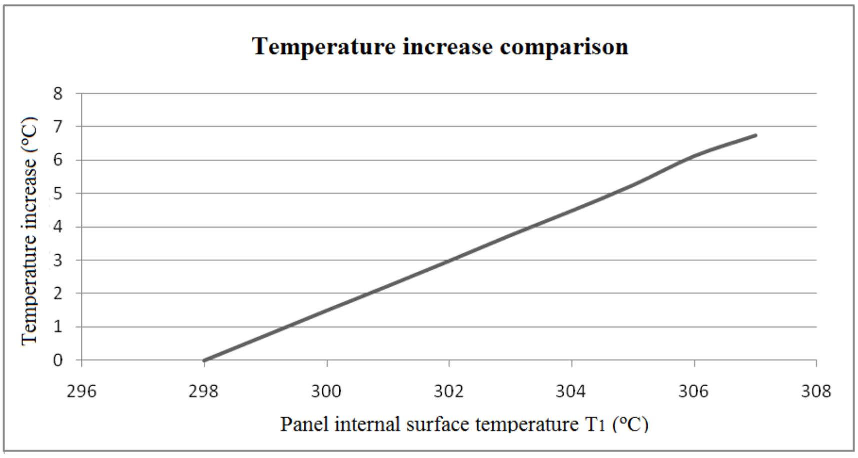


Figure 12. Average temperature variation simulated by the CFD when T_1 is modified

In the same way that T1 can be modified by environmental conditions such as radiation and then the indoor temperature is affected, the air velocity in the ventilated chamber can be modified by the heating or cooling convective effect and achieve a similar effect on the indoor temperatures. This feature is shown in the figures that represent the velocity profiles previously described.

2.5. Energetic analysis

The ceramic panel installation in order to create a ventilated façade should have, from the energy efficiency point of view, the main objective to reduce the cooling energy required to climate the indoor environment, as proposed in the present contribution. This issue has been recently studied and reflected in the literature [16, 17].

The energy analysis proposed requires thus knowledge of the mechanism of heat transmission through walls. The U-value knowledge is required for modelling the heat transfer. The information regarding the transmission coefficient of every layer that composes the façade is required. However, when all this information is not available; a comparative analysis could be done. This comparative analysis is proposed in order to estimate and demonstrate how the CFD technique could help designers.

The heat transmitted through the building wall that is in contact with the ventilated façade could be known using the computational model and considering that the indoor temperature should be kept at 296K (23ºC). The reviewed literature proposes values between 0.55 and 1.42 for the U-value [18]. Table 1 shows the thermal material conductivity coefficients “k” considered and the indoor and outdoor convection coefficient h = 10 W/m^{2}K. The complete heat transmittance U-value for the enclosure is calculated following expression (11). The literature default values are used for the coefficients of the different materials that could compose the façade [19].

$$U = \frac{1}{(\frac{1}{h_i} + \sum\left(\frac{e_i}{k_{i+}}\right) + \frac{1}{h_e})} \tag{11}$$

Table 1. *Considered values for the enclosure heat transfer coefficient calculation*

Global heat transmission coefficient value

h_indoor	10	W/m^2·K
(e/k)	0.71	m^2·K/W
h_outdoor	10	W/m^2·K
U	**1.10**	**W/m^2·K**

Default façade building materials

	e (m)	k (W/m·K)	e/k (m^2·K/W)
Cement grouting	0.02	1,2	0.017
Double hollow brick	0.12	0.42	0.29
Non-ventilated air chamber	0.04	0.19	0.21
Single hollow brick	0.05	0.42	0.12
Plaster cast	0.02	0.26	0.08
		TOTAL	0.7084

The global heat transmission coefficient is then equal to $1.1\frac{W}{m^2K}$.

The wall in which the façade is not installed temperature is 305K (32ºC) based on the measurements done. The heat transferred through the wall is:

Case a) non-ventilated façade:

$$q = U_{wall} \cdot (\Delta T) = 1.1 \cdot (T_0 - T_{hab}) \frac{W}{m^2} \qquad (12)$$

Case b) ventilated façade:

$$q = U_{wall} \cdot (\Delta T) = 1.1 \cdot (T_2 - T_{hab}) \frac{W}{m^2} \qquad (13)$$

A simplified analysis of the previous simulation shows the difference of heat transferred from outside to indoor between the case with and without the ventilated façade considering the same external conditions for the simulation.

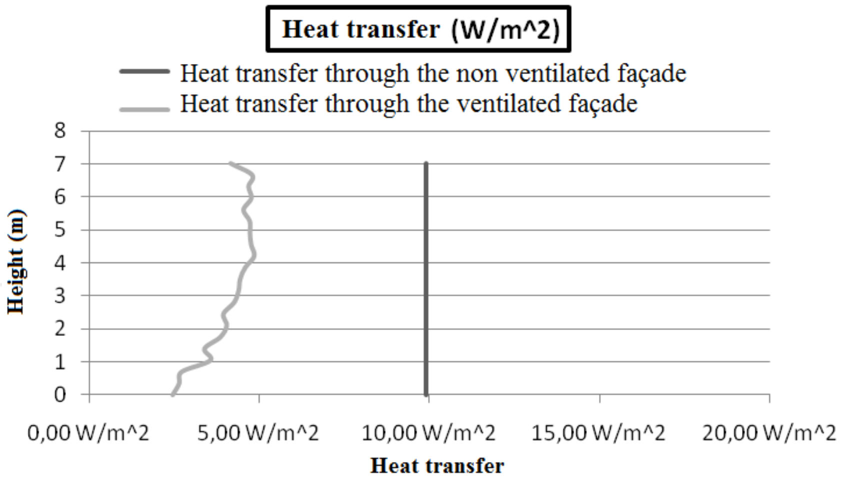


Figure 13. Heat transfer through the façade to maintain the indoor temperature at 296K.

Figure 13 represents the energy required to maintain the indoor temperature at 296K. The analysis of this figure shows that 9.9 W/m^2 are transferred to keep the indoor conditions in the case without the ventilated façade. However, the heat transfer when the ventilated façade is installed is lower, 4.09 W/m^2 for the conditions considered. Consequently there is a cooling effect on the wall that achieve a reduction of 58.7% on the energy required to keep the indoor temperature cold when installing the ventilated façade in comparison with the simple façade.

This energy saving obviously depends on the external thermal conditions. As shown here, the energy-saving effect of the ventilated façade will be lower as soon as the panel temperature will be decreased in comparison with the case without ventilated façade. The ventilated façade will not be necessary when the outdoor and the required indoor temperature will be the same. The analysis do not considers the walls and materials thermal inertia as it is a steady state analysis. Nevertheless the thermal inertia effect will be important in the analysis of average daily and seasonal savings.

The depicted results are aligned with the latest ceramic ventilation façade researches. The Ciampi research [15] presents a comparison between the effect of the air movement in the ventilated façade and the additional traditional insulation layer that should be added to the non-ventilated façade to achieve the same effect in an analytical way. The present research proposes the use of the computational fluid dynamics technic to estimate the energy saving of the ceramic ventilated façade in order to complement the Ciampi research [15].

3. Conclusions

The quantification of the ceramic ventilated façade effect in a building is complex. The literature does not provide direct parameters to quantify the efficiency of the natural ventilated façade. The present contribution proposes a strategy to quantify this effect considering some boundary condition and using the computational fluid dynamics simulation (CFD).

The fluid mechanic equations that describe the air movement in the ventilated façade chamber are solved through computational software. A case study is depicted considering suitable Mediterranean conditions and field measurements in summer conditions. The air velocity and temperature in the ventilated chamber and the resulting temperature in the external building wall are calculated by the software. Consequently, the cooling effect that could produce the ventilated façade installation in the building is assessed. The temperature profile obtained by the simulation is used to quantify the cooling energy savings. The case with and without the ventilated façade are compared and the results is that the ventilated façade installation could save up to 58.7% of the energy initially required to maintain the initial indoor temperature. This percentage is reached under certain external temperature condition assumption in steady state.

The computational fluid dynamics technique is presented as a powerful tool to represent the behaviour of the building envelope when a ceramic ventilated façade is installed besides the current methodology proposal to know the air temperature of the ventilated chamber. The air velocity and temperature profile have a fundamental role in the behaviour of the ventilated façade and it is very challenging and almost impossible its calculation without the computational assistance.

Therefore, a more complex mathematical modelling is required to represent the heat transfer through the building wall and the ceramic panels. There are many parameters that influence the ventilated façade energy efficiency such as the panel separation, the velocity and temperature of the entering air, solar radiation and the temperatures of the solid panels, the heat transmission coefficients of the solid elements, etc. However, the presented computational modelling will allow the variation of these parameters to determine the optimal working solutions, especially if the radiation effect on the solid elements, the thermal inertia and a non-steady state will be integrated in the simulation.

In this regard, the present contribution shows computational fluid dynamics techniques as an adequate and complete technique to evaluate the ventilated façade performance. This will allow knowing the best conditions for the ventilated façades selection and installation in order to optimize its behaviour from an energy point of view even before its installation, in the future. Moreover the present research should be complemented by the determination of real results in order to validate the numerical simulations in the future.

Acknowledgements

The research has been conducted within the frame "computational fluid dynamics techniques applied to environmental flows analysis” at the Polytechnic University of Valencia.

References

[1] Balocco, C. A simple model to study ventilated façades energy performance. Energy and Buildings 2002, Vol 34, N 4. Pp 469-475.

[2] Linden, P.F. The fluid mechanics of natural ventilation. Ann. Rev. Fluid Mech. 1999, 31, 201-238.

[3] El Sadi, H.; Haghighat; and Ali Fallahi. CFD Analysis of Turbulent Natural Ventilation in Double-Skin Façade: Thermal Mass and Energy Efficiency. Journal of Energy Engineering 2010, 136, 68-75.

[4] Todorovic M, Ecim O, Marjanovic A and Randjelovic I: Natural and Mixed Ventilation Design via CFD and Architectural Modelling, International Journal of Ventilation 2007, 5, (4), 447-458.

[5] Tennekes H, Lumley JL. A first course in turbulence. Cambridge. MA: The MIT Press 1972.

[6] White FM. Viscous fluid flow. New York: McGraw-Hill, Inc., 1991.

[7] Gualtieri, C., López Jiménez, P.A., and Mora Rodríguez J.J., A comparison among turbulence modelling approaches in the simulation of a square dead zone. XXXIII IAHR Congress, Vancouver, Canada, 2009.

[8] Gualtieri, C., López Jiménez, P.A., and Mora Rodríguez J.J. Modelling turbulence and solute transport in a square cavity. First European IAHR Congress. Edimburgo, 2010.

[9] Wilcox, D.C., Turbulence modeling for CFD, DCW Industries, 2002.

[10] Hanjalić, K., Closure models for incompressible turbulent flows, Lecture Notes at Von Kármán Institute, pp.75, 2004.

[11] Sarkar, S., and Balakrishnan, L. "Application of a Reynolds-stress turbulence model to the compressible shear layer", ICASE Report 90-18, NASA CR 182002. 1990.

[12] Speziale, C.G., Sarkar, S., and Gatski, T.B. "Modelling the pressure-strain correlation of turbulence: an invariant dynamical systems approach", J. Fluid Mech.1991, 227, pp. 245-272.

[13] STAR-CCM+ 3.04.009, User's Guide, CD-Adapco, USA, 2010.

[14] Launder BE, Spalding DB. Lectures in mathematical models of turbulence. New York: Academic, 1972.

[15] Ciampi, M.; Leccese, F.; Tuoni, G. Ventilated façades energy performance in summer cooling of buildings. Solar Energy 2003, N. 75. Pp 491–502.

[16] Gang, G. A parametric study of Tromble walls for passive cooling of buildings, Energy and Buildings 1998, Vol 27, N 1. Pp 37-43.

[17] Balocco, C. A non dimensional analysis of a ventilated double façade energy performance. Energy and Buildings 2003, Vol 36, N 1. Pp 35-40.

[18] Yilmaz, Z. Evaluation of energy efficient design strategies for different climatic zones: Comparison of thermal performance of buildings in temperate-humid and hot-dry climate. Energy and Buildings 2007, Vol 39, N 3. Pp 306-316.

[19] CTE. Código Técnico de la Edificación. Documento Básico. Ahorro de energía. 2010.http://www.codigotecnico.org/cte/export/sites/default/web/galerias/archivos/DB_HE_abril_2009.pdf

Annex IV

Quantification of Ventilated Façade Effect due to Convection in Buildings.

Buoyancy and Wind Driven Effect.

Abstract: The external layer in a building has a paramount role under the building's energy behaviour point of view. The ventilated façade is a passive system installed on buildings to improve the global energy behaviour. The ventilated façade performance is described. Ventilated façade is mainly based on convection and radiation. This contribution focuses on the convective effect due to buoyancy driven respect to wind forces in the ventilated gap and their influence in the building thermal behaviour. To do so, several computational fluid dynamics models are undertaken with 1.5m/s, 0.5m/s and 0.2m/s wind velocities and a wind temperature of 298K and 300K. In the modelled conditions, the temperature of the external face of the façade was decreased due to the ventilated façade effect at 0.2m/s. 35% of the temperature reduction was due to natural convection buoyancy-driven effect and the rest due to wind-driven effect. With higher wind velocities (1.5 m/s) the temperature reduction due to the convective effect was negligible. According to these simple trials it can be concluded that the buoyancy driven forces have influence only below a certain threshold of wind velocity. The paper helps to better understand the behaviour of the ventilated façade installed in a building and allow designers to quantify the influence of the façade on the global building energy balance.

Keywords: Convection; Natural ventilation (NV); Ventilated façade (VF); Computational fluid dynamics (CFD)

1. Introduction

Government bodies, architects and engineers are becoming increasingly aware of the importance to reduce energy usage and CO_2 emissions, resulting in an increasing number of new solutions applied to improve the energy behaviour of the buildings. Thermal behaviour is one of the most important to be considered. The main responsible of the heat exchange between the indoor and outdoor environment of a building is the building façade. It is the responsible for the performance of several physical, dynamic and thermal parameters of the whole system [1]. Thermal comportment of the façade is crucial to understand the energy behaviour and to reduce the energy waste of the buildings. Up to now, many techniques have been used to reduce this heat exchange through the façade. Initially, the façade thermal performance was improved by increasing thickness of insulating materials: as higher was the indoor-outdoor temperature difference; greater was the coating material thickness. Secondly, the quality of the coating material was improved with better isolation materials. Nowadays much sophisticated techniques are being studied and already used such as ventilated façades (VF), which have many energy implications [2].

Particularly, a ventilated façade is a system used to improve the thermal behaviour of the buildings. This system allows designers to improve the energy behaviour in both, new building design and rehabilitation in the existing ones. The ventilated façade system is generally made up of an external cladding layer attached to the last continuous insulating external layer of the building by means of a mechanical structure (generally made of aluminium profiles). The cladding layer is made of glass, marble, ceramic, etc. and it defines the external appearance of the building. An air cavity of at least 3 cm wide is thus created between the insulation and the cladding layer. The mechanical structure must allow an upward a continued air flow in the façade. Thus, this air cavity is naturally ventilated as a result of solar radiation on the slabs and the ensuing convention within the cavity. There are two different ventilated façades depending on the way that the different slabs are installed. On the one hand the continuous VF (close joints) in which the upward flow is completely continuous, homogeneous and symmetrical along the wall [3]. On the other hand the façades with open joints between each cladding, commonly known as OJVF (Open Join Ventilated Façade), which turn the flow discontinuous, inhomogeneous and much more complex [4].

Numerical models of air movement have been developped in VF and other building structures, such as solar chimneys or atriums by means of CFD techniques [5]. As instance, in the work of Seytier and Naraghy [6], the radiative and convective effect in the buoyancy along a solar chimney is studied, successfully validating the computational fluid dynamics (CFD) uses for convection simulation. The influence of convection in buildings façades is a permanent topic of interest in research [7-10] always concluding the importance of the façade in the reduction of energy needs in the whole building, and the importance of an accuracy model for the deep knowledge of the façade performance.

Natural convection is one of the main principles for VF [11]. Heat transfer is classified into three main mechanisms: thermal conduction, radiation and convection. The fluid flow can be forced by external mechanisms or by buoyancy forces: natural convection. Natural ventilation (NV) is explained by two phenomena: wind driven ventilation and buoyancy-driven ventilation. Wind driven ventilation occurs due to the pressure difference in the façade surfaces produced by wind forces. Buoyancy-driven ventilation occurs as a result of the directional buoyancy forces that results from temperature differences between the interior and exterior [12]. This paper is focused in the thermal convection performance of an open join ventilated façade.

This effect has been partialli investigated. Mingotti described the fluid mechanics of the natural ventilation of a narrow-cavity double-skin façade [13]. Increasing the height of the façade, the buoyancy effect could be improved leading to a faster flow. This faster flow will improve the energy performance of the VF. The research presented by Gratia concluded than the ventilation of the façade is driven primarily by wind on the upper floors, where buoyancy heads are small, but by buoyancy on the lower floors, where buoyancy heads are large [14].

Different modelling approaches are used in predicting building ventilation including analytical, empirical, experimental and CFD models [15]. Recent studies have been done to improve the experimental data available in the literature: Lomas K.J. et al. studied a hybrid advanced naturally ventilated system in a new building [16], Giancola carried out an experimental assessment and modelling of the performance of an OJVF [17], Sanjuan developed and experimental validation of a computational model for OJVF [18] and González adjusted and energy computational model to analyze its energy performance in a building [19]. In those, the fluid and thermal performance of OJVF has been investigated using both experimental and numerical methods. These studies confirm CFD as a reliable tool to model the fluid behaviour in this particular application. Furthermore, CFD simulation techniques enable designers to understand the behaviour of the systems and to predict whether it will work as expected or not. Thus the designer can optimize their constructive solutions in real scale models in an efficient computational way and not by expensive trial-and-error methodologies. The advantage of using these models resides in the fact that they can reproduce real problems of Fluid Mechanics to any degree of complexity.

In this contribution, the objective is to quantify the influence of the buoyancy-driven ventilation to increase the air velocity in the air due to the directional buoyancy force. This phenomenon is mathematically modelled by CFD techniques applied in a ventilated façade.

2. Methodology and general objective

This research aims to quantify the convective flow effect due to buoyancy-driven forces in the air gap of an Open Join Ventilated Façade. The wind velocity and the temperature differences between the façade faces are the parameters that need to be defined, modified and then compared.

First stage on the CFD analysis method consisted of defining a simplified OJVF geometry in which the fluid phenomenon was studied. The model was composed of a geometry, boundary conditions and mesh definition. The geometry and the mesh definition remained constant in all simulations.

Figure 1 shows the control volume where the research was focused. It is composed by the indoor face of the cladding material, the ventilated gap and the external building's face. The volume was especially chosen to determine the temperature effect of the air flow through the cavity in the building external wall.

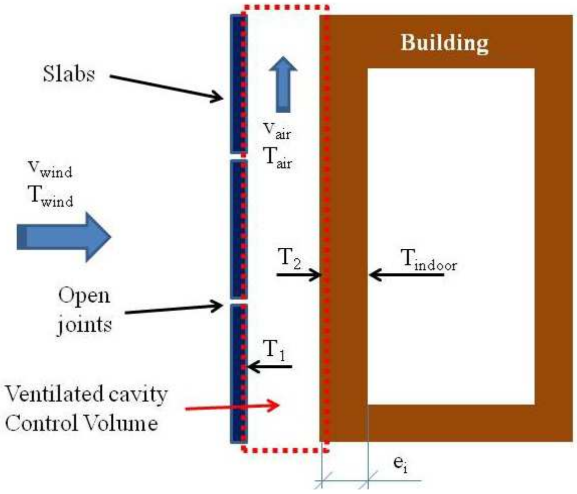


Figure 1. Control volume and temperature definition

The CFD model simulated the air velocity in the gap taking into account the wind velocity and all the hydrodynamic effects: the narrow apertures of the OJVF, the friction forces, the wind driven flows, the buoyancy natural ventilation, etc. Wind velocity (v_{wind}), temperature (T_{wind}) and the temperature of the internal face of the cladding layer (T_1) were set at different values specially chosen to quantify the buoyancy-driven effect

(BDE) in the ventilated gap. The external façade layer temperature T_2 was provided by final CFD simulation.

A first set of six simulations has been used as a reference (Non-BDE simulations) to analyze the thermal and fluid performance of the air in the ventilated façade. The set includes three pairs of simulations in which wind velocity was decreased: 1.5m/s, 0.5m/s and 0.2m/s respectively. Wind and the indoor cladding face temperatures were set at 298K and 300K in each pair of simulations; $T_1=T_{wind}=298K$ and $T_1=T_{wind}=300K$ respectively. Then, air velocity in the ventilated layer and temperature in the external façade (T_2) were calculated. These simulations were used as reference because there was no air movement due to the BDE because there wasn't any temperature gradient.

A second set of three simulations has been done to analyze the BDE in the air through the ventilated gap in comparison with the first set of simulations. The wind velocity was set at 1.5m/s, 0.5m/s and 0.2m/s in each simulation. Wind temperature was set at 298K and the indoor cladding face temperature was set at 300K in each simulation. Then, air velocity in the ventilated layer and temperature in the external façade (T_2) were calculated.

A third set of simulations has been carried out to notice how the increase of the temperature difference affects the BDE in the air through the ventilated gap with high and low wind velocities (1.5m/s and 0.2m/s respectively). The wind temperature was set at 298K and the indoor cladding face temperature was set at 302K, 304K and 306K for low velocity (0.2m/s) and at 306K for high velocity (1.5m/s). Then, air velocity in the ventilated layer was calculated. These data of velocity and temperature of wind correspond to measurements performed in Valencia (Spain) in June. The temperature gradients have been carefully chosen according to Mora-Pérez et al. [20].

To simulate the wind velocity modulus it is necessary to set and to adapt wind data for height with the correct approach profile of mean wind-speed. In general, in a big vertical scale, the potential law for velocity modulus is accepted as vertical velocity profile [21]. Nevertheless, in this particular case, in order to make a simplification and considering that the present research describes a very low structure compared with the atmospheric boundary layers, constant values for wind velocity have been considered to perform the numerical model.

The first sets of simulations (used as reference) were compared with the second set to analyze the influence of the wind in the natural ventilated flow in terms of convection. In the first simulations the BDE didn`t exist because there wasn`t any temperature gradient. Moreover, the second set of simulations was set at 2K temperature gradient (T_1-T_{wind}). This temperature gradient produced a BDE that was quantified by comparison respect the case without it. Absolute results in each simulation have no sense in this research. Additionally, the air velocity and the temperature of the air in the ventilated gap and the building`s façade temperature (T_2) were analysed.

Finally, the third set of simulations compared at lower and higher wind flows how the temperature gradient affected the buoyancy-driven forces. The temperature gradient was set at 4K, 6K and 8K at low velocity (0.2m/s) and 8K at high velocity (1.5m/s). In addition, the air velocity in the ventilated gap was analysed.

3. Modelling strategy

3.1. CFD solver applied to ventilation in the façade gap

The depicted methodology is an investigation with CFD and its application research in building systems. The literature is profuse in documents based on research applications of CFD, including experimental validations. Marioscini investigated experimentally and numerically the thermal comportment of a real OJVF in winter conditions [22]. Wang modelled and validated the impacts of ventilation strategies and façades on indoor thermal environment for naturally ventilated residential buildings [23]. Omar recommended CFD as a reliable method to study systems that have no access to laboratory or full-scale testing facilities [24]. This reference found agreement between experimental and numerical approach although the good agreement was restricted in the majority of the cases to conditions of calm wind. The agreement with experimental results strongly supported the use of CFD for studying the fluid behaviour in a ventilated façade.

CFD allows designers to understand the fluid behaviour in different systems and to take better design decisions. Compared to other references like Giancola [17] and Sanjuan [18], who contrasted the results of the CFD simulation with real experimental results; this contribution assumes that CFD simulations are right to represent the fluid behaviour. CFD is used as a design tool as Kang et al. [25] used the methodology to improve NV in a large factory building and Mora-Pérez et al. to quantify the efficiency of the ceramic ventilated façade [20].

3.2. Mathematical model

CFD is a detailed modelling technique based on mathematical models which discretises the space in small cells where mass and momentum conservation equations are solved by the code. These equations are solved in a geometrical domain defined by boundary conditions and taking into account turbulent phenomena. The continuity or mass conservation equation solved by the software used is expression (1).

$$\frac{\partial \rho}{\partial t} + \nabla \rho \vec{v} = S_m \tag{1}$$

Where ρ is the fluid density, $\vec{v}$ is velocity and S_m represents the mass contained in the control volume. The momentum equation is considered in equation (2).

$$\frac{\partial (\rho \vec{v})}{\partial t} + \nabla \rho (\vec{v}\vec{v}) = -\nabla p + \nabla \bar{\tau} + \rho \vec{g} + \vec{F} \tag{2}$$

Where p is the static pressure, $\bar{\tau}$ the stress tensor defined in expression (3) and $\vec{g}$ and $\vec{F}$ the gravitational and outer forces respectively. μ is the eddy viscosity and I is the unit tensor.

$$\bar{\tau} = \mu\left[\left(\nabla\vec{v} + \nabla\vec{v}^{T}\right) - \frac{2}{3}\nabla\vec{v}I\right] \tag{3}$$

A general purpose CFD software package has been used: STAR-CCM+. The equations solved by STAR are discretized according to Finite Volume method (FVM). In FVM the solution domain is subdivided into a finite number of small cells called Control Volumes (CVs). Usually CVs are defined by a suitable grid and computational node is assigned to the CV center. All variations of FVM share them same discretization principals. They are different in relations between various locations within integration volume. The integral form of Navier Stokes equations are applied to each CV, as well as the solution domain as a whole. Summing all the equations for all CVs is obtained the global conservation equation since surface integrals over inner CV faces cancel out. The result is a set of linear algebraic equations with the total number of unknowns equal to the number of cells in the grid. Appliying this discretization to equation (2) the following discrete equation for velocity is obtained [26], where v_g is velocity grid, "f" refers to each face in the discretization method, and "a" is the face area vector.

$$\begin{aligned}\frac{d}{dt}(\rho v V)_0 + \sum_f \left[v\rho\left(v - v_g\right)a\right]_f &= \sum_f \left(pI \cdot a\right)_f \\ &\quad + \tau \cdot a + (\rho + +F)a_f\end{aligned} \tag{4}$$

4. Simplifications

Several simplifications were asumed to reduce the computational time:

- The modelling is considered steady.

- No radiation effect is considered in the model, as the study is focused on the convective effect.
- The study is focused in the ventilated air gap between the internal sheet of the cladding layer and the external face of the building, where the convective effect takes place.
- The capacity of the cladding material to accumulate heat is not taken into account. The study is focused in the fluid region.

4.1. Modelled geometry

The geometry modelled is a simplification of a OJVF installed in a building exposed to wind. The geometry consists of two columns with 14 slabs separated by horizontal and vertical joints 2mm wide.

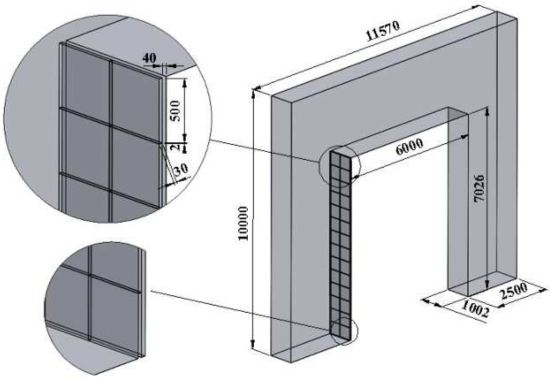


Figure 2. Building and panel dimensions (mm)

Each column represents half piece of the cladding material which made the external ventilated layer. Each slab is 0.5m. high, 0.03m thick and 1m. wide. The ventilated gap simulated is 40mm. thick and 1.002m. wide. The ventilated façade is installed on a building in a wind tunnel. The building is 6m. deep. The wind tunnel is 10 m. high, 1.002m. wide and 11.57m. deep. A detail of the air gap and the dimensions of the building model are shown in Figure 2.

4.2. Boundary conditions and physical description

The simulation has been performed under steady conditions in a 3D model, with constant density fluid flow. Turbulence effects have been included using the K-Epsilon model and segregated flow. The gravity model was used as it permits the inclusion of the buoyancy source terms in the momentum equations when using the segregated flow model.

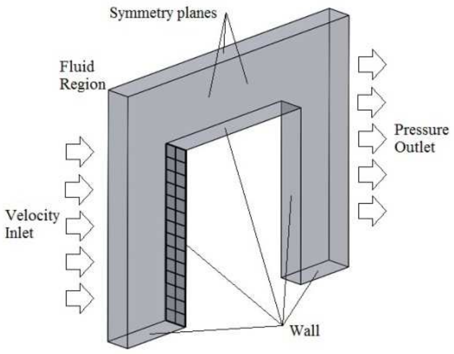


Figure 3. Boundary conditions for CFD model

The whole domain was defined as fluid region (air) with boundaries. The boundaries settings were specially chosen to simulate the BDE in a building with an OJVF in a wind tunnel. The wind tunnel was defined by three symmetry planes (both laterals and the top face), a wall (bottom face), a velocity inlet in front façade and a pressure outlet at the end of the wind tunnel. The cladding and building's faces were defined as walls with a roughness of 2.5E-7m. (Table 1)

Table 1. Boundary conditions specifications

Type	**Surface (Wind tunnel)**	**Properties**
Velocity Inlet	The front face	Constant velocity. 1.5m/s, 0.5m/s and 0.2m/s perpendicular to the building
Pressure outlet	The back face	Constant athmospheric pressure
Wall	All building and façade faces and the bottom face	Constant Roughness height = 2.5E-7 m.
Symmetry Plane	The upper and lateral faces	

4.3.CFD mesh and convergence

The simulation was done with a numerical method which is solved by the finite volume technique. The fluid model was solved by calculating the flow-equations on the nodes within the cells. The accuracy of the result depends on the definition of the nodes. A structured mesh was created with a refined grid near the joints. In addition, the mesh near the wall was set fine enough to allow the enhanced wall treatment to solve the near wall region all the way to the sub laminar region.

A mesh analysis was done to find out the optimum between the smallest number of nodes and the accuracy of the results. A grid-independence study, including the number of nodes and the size of the enlarged domain was performed in order to assess the validity of the numerical computational procedure.

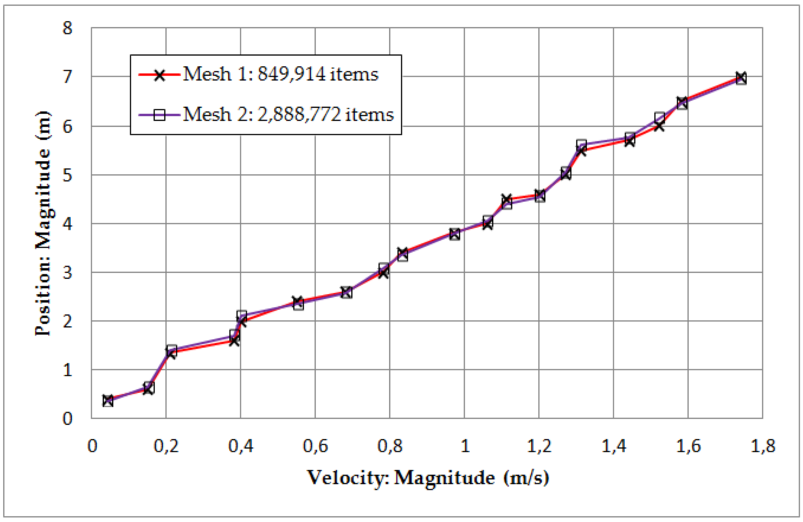


Figure 4. Velocity comparison between meshes

Two meshes were modelled with a computer Intel® Core™ i5 CPU M520 @2.4GHz. M1 with 849,914 items (M1: accuracy of 2 mm. and CPUTime 716.5 s) and M2 with 2,888,772 items; 1 mm. accuracy and CPUTime 2226.16s). Velocity magnitude in a line in the ventilated façade was compared. Figure 4 shows that the velocity difference between models was less than 0.04%. Furthermore the M1 model lasted 3 times more than the M2 model. Therefore the first mesh was selected. Figure 5.1 shows a detail of the mesh selected and Figure 5.2 shows the CFD residuals calculation.

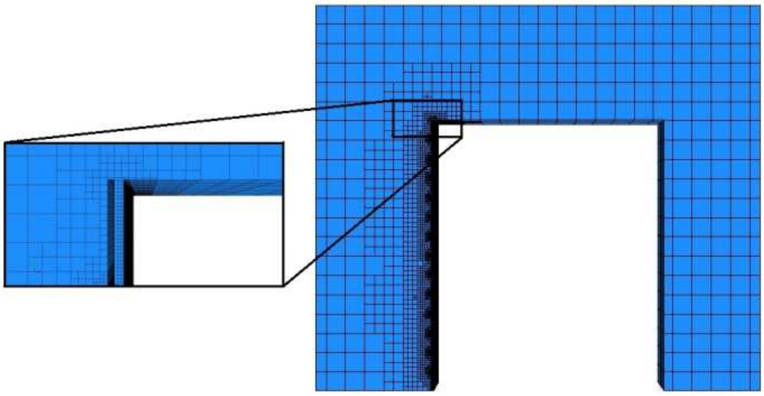


Figure 5.1. Detail of the selected mesh (M1)

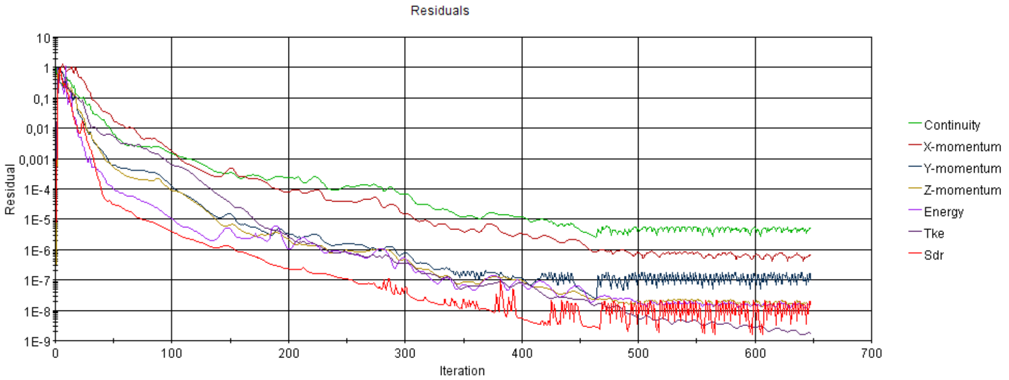


Figure 5.2. Residuals of the CFD calculus for M1 model

Wall treatment is necessary for model up proper boundary conditions. A numerical verification was made to check that the model was all right. In this case the high-y+ wall treatment used implies the wall-function-type approach, in which it is assumed that the near-wall cell lies within the logarithmic region of the boundary layer. A good rule is that the wall-cell centroid should be situated in the logarithmic region of the boundary layer (y+>30) [26]. In this case, this requirement was accomplished for all walls.

4.4. CFD post-processing

The last step in the CFD modelling strategy consists on obtaining the calculated parameters useful for the analysis. Post-processing allows designers to obtain XY plots (velocity, pressure, temperature, etc.), 2D and 3D vector representations, streamlines, numerical data, etc. In this case, the parameters used were air velocity in the ventilated air gap and the temperature of the external face of the building (T_2). The velocity of the air was determined in the centre of the ventilated gap in the middle of the panel. The temperature of the face was determined in the middle of the panel at the buildings face side.

5. Analysis of results

5.1. Analysis on air gap velocity

Several set of simulations were done to analyze the influence on the ventilated gap of both the buoyancy and the dynamic pressure effect due to wind. The dynamic pressure acting on the façade is generated when is converted the kinetic energy of the wind into pressure energy due to the braking of the wind on the fixed surface of the building.

To analyze the relative weight of wind dynamic pressure over the buoyancy effect, different situations have been considered. On one side, wind speed has been changed, keeping the same temperature difference between the internal face of the cladding and wind. On the other side, for a low wind speed, in which the buoyancy effect is significant, it has been changed the temperature gradient between the cladding and wind.

Then, a first set of simulations have been performed without considering the buoyancy effect (Non-BDE simulations). The air into the gap should not be heated. To get this the wind (T_{wind}) and the internal face of the cladding (T1) temperatures were the same. The air movement in the chamber was only due to wind dynamic pressure (Figure 6 lines a-b, d-e, g-h).

Table 2. Temperatures and velocities considered

	Simulation								
Case	a	b	c	d	e	f	g	h	i
BDE	No	No	Yes	No	No	Yes	No	No	Yes
T_{wind} (ºK)	298	300	298	298	300	298	298	300	298
T_1(ºK)	298	300	300	298	300	300	298	300	300
v_{wind} (m/s)	1,5	1,5	1,5	0,5	0,5	0,5	0,2	0,2	0,2

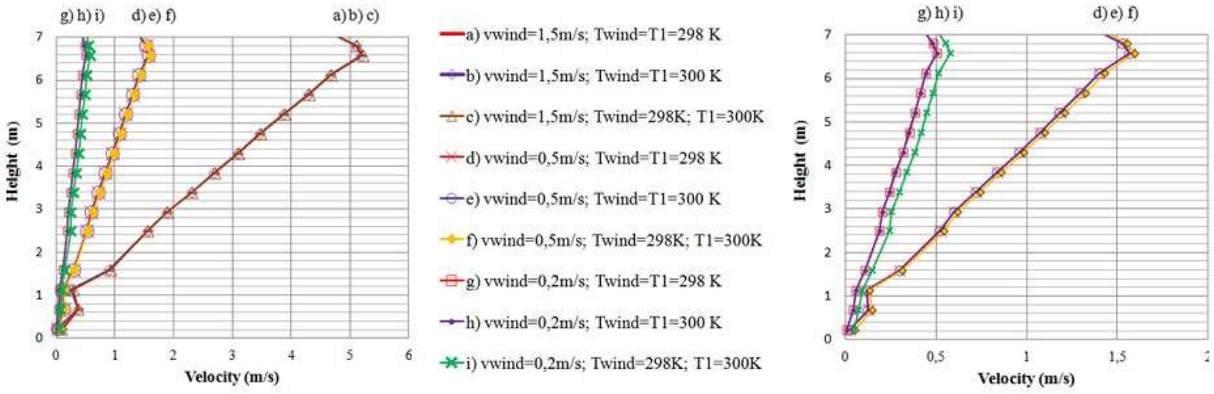


Figure 6. Air velocity in the ventilated gap

Figure 6 shows the evolution of wind speed inside the gap (v_{air}) for the simulations listed in Table 2.

It is noted that the higher the wind speed (v_{wind}), and thus the dynamic pressure on the façade, the higher the velocity of air inside of the gap (v_{air}). Furthermore, at medium and high wind speeds, the buoyancy effect is not appreciable compared with the dynamic pressure effect. By contrast, at low wind speeds (simulations g, h, i) there is a certain influence of the buoyancy effect with respect to the dynamic pressure effect, due to the small value of the latter. As the façade is opened in the lower part, the wind driven effect causes a higher effect in this range of wind velocities due to the great dynamic pressure generated by the wind in comparison with the buoyancy effect.

The second and third set of simulations considered the buoyancy effect (BDE simulations). The internal face of the cladding temperature was greater than that of the wind, as there was a progressive transmission of heat to it. Air movement in the gap was due to dynamic pressure and buoyancy effect.

Table 3. Temperatures and velocities considered

	Simulation						
Case	c	f	i	k	m	n	p
BDE considered	Yes	Yes	Yes	Yes	Yes	Yes	Yes
T_{wind} (ºK)	298	298	298	298	298	298	298
T_1(ºK)	300	300	300	302	304	306	306
v_{wind} (m/s)	1,5	0,5	0,2	0,2	0,2	0,2	1,5

The temperature gradient between the wind and the internal face of the cladding material (T_1-T_{wind}) was increased from 2K to 4K, 6K and 8K (third set of simulation) with a low wind velocity (0.2m/s).

Figure 7 shows the influence of the buoyancy effect when the wind dynamic pressure is reduced. The greater the temperature gradient between the air and the cladding, the greater the buoyancy effect and the air velocity inside the gap.

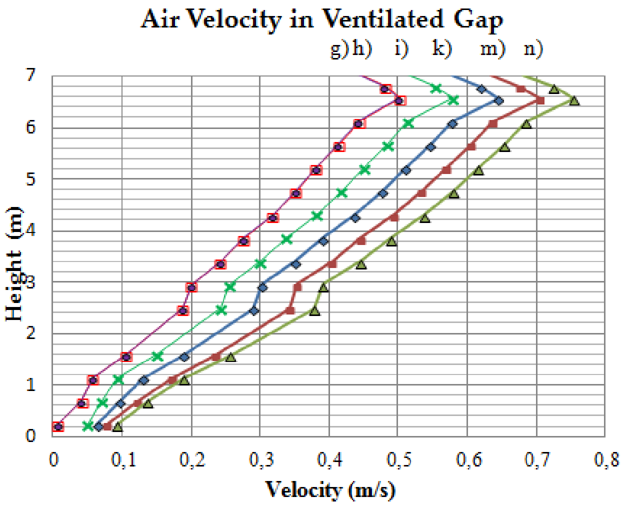


Figure 7. Air velocity in the ventilated gap at 0.2m/s wind velocity

Finally, to verify the reduced influence of the buoyancy effect at high dynamic pressure, a new simulation has been done (simulation p). It has been considered the major buoyancy effect (high gradient of temperature, T_{wind} = 298°K, T_1 = 306°K) for the dynamic pressure corresponding to v_{wind} = 1.5 m/s. In figure 8 the evolution of velocity in the gap together with the simulations a, b, c, (Table 1) are shown. There was no difference observed in the evolution of v_{air} velocities. Then, the air velocity increase due to the BDE produced by increasing the temperature gradient at high velocities was negligible.

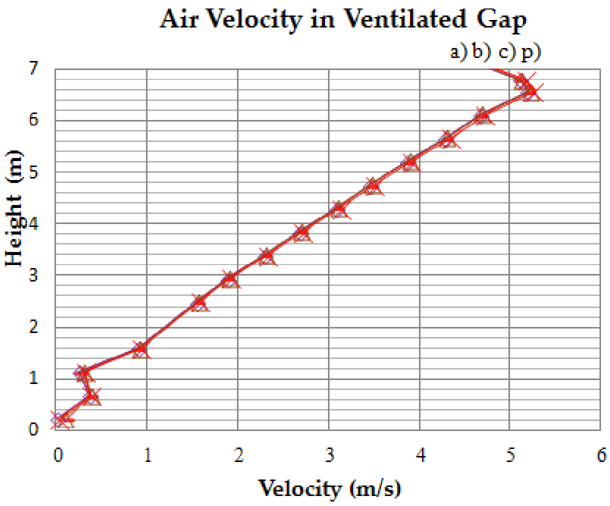


Figure 8. Air velocity in the ventilated gap at 1.5m/s wind velocity

5.2. Analysis on façade temperature (T_2): heat flow to the building

An analysis has been done to the temperature on the outer side of the façade (T_2), which was in contact with the gap, for the same cases listed in Table 1.

The objective is to verify how it will be affected the temperature T_2, and therefore the thermal comfort inside the building, by varying the dynamic pressure, due to the air velocity, and the buoyancy effect.

Figure 9 shows results from the different simulations. For those cases in which there was no considered the BDE, the temperature of the façade was the same as the wind, the inner cladding and the air in the ventilation gap. For those cases in which BDE was considered, the air temperature inside the gap (T_{air}) was generally higher than the temperature on the façade (T_2). There was a mismatch in this behavior, due to edge effect, on the top of the gap.

By heat transfer, the greater the flow velocity of the fluid through the gap, the lower the temperature reached. This is the reason why the air temperature in the case c is lower than for the case i (Figure 9).

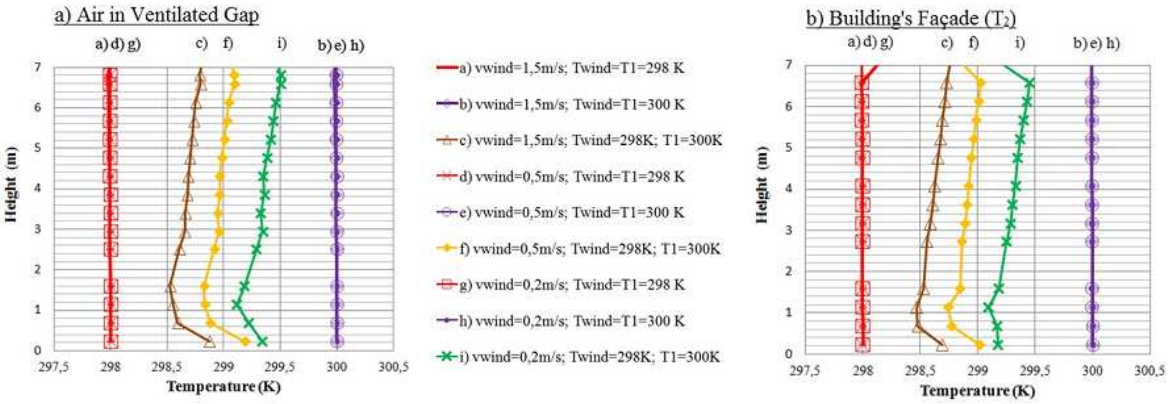


Figure 9. Temperature performance

For a given wind velocity (fixed dynamic pressure) the air temperature along the façade increased due to the heat transmitted from the cladding. The temperature gradient along the façade for the case where the buoyancy effect was significant (case c) was of the same order of magnitude as for the case in which buoyancy effect as hardly seen (case i), although air temperature (T_{air}) was higher. This behaviour confirmed that the buoyancy effect was minor compared with the effect of the dynamic pressure.

From the buildings energy behaviour point of view, the main objective of the ventilated façade is to reduce the load gain of the building to decrease the cooling power required to keep a comfortable indoor environment in summer. The thermal performance of the ventilated façade relies on buoyancy: first the slabs of the exterior coating are heated up.

Then, the heat load is transferred through the cladding material to its interior face. Finally, it produces an ascending mass flow air in the ventilated cavity by natural convection. The thermal behaviour of the façade is different depending whether it is summer or winter as it was explained by Giancola in an experimental case [17]. In summer, the ventilated façade reduces the heat transfer to the indoor environment. Equation 4 shows that the total load gain q (W/m^2) in the building depends on the U-coefficient (W/m^2/K) and the temperature gradient of the inner and the outer face of the wall. Equation 5 shows how U-coefficient is calculated. It is considered the wide of each material layer which built the external wall e_i (m.), its heat-conductivity coefficient k_i (W/m/K) and the indoor and outdoor convective-coefficient h_i & h_e respectively (W/m^2/K).

$$q = U_{wall} \cdot (\Delta T) = U_{wall} \cdot (T_2 - T_{indoor}) \frac{W}{m^2} \tag{4}$$

$$U = \frac{1}{\left(\frac{1}{h_i} + \sum \left(\frac{e_i}{k_i}\right) + \frac{1}{h_e}\right)} \tag{5}$$

The present report quantifies the energy reduction due to the natural convection buoyancy-driven effect in the ventilated cavity. The U-value remained constant for all calculations and the unique change was the temperature drop. 35% of the ventilated façade energy savings were due to the buoyancy-driven effect. This result was calculated with the temperature drop in the building's façade (Figure 9, "h" line, T_2) respect the temperature of the inner face of the ventilated slab (Figure 9, "i" line, T_1) for the low wind velocity (v_{wind}=0.2m/s) case. In these particular conditions, the buoyancy driven effect supposed a reduction of 10 W/m^2 (out of a total of approximately 28 W/m^2 reduction in cooling power due to the presence of the air in the ventilated façade of the building).

6. Conclusions

A ventilated façade is a powerful system used to improve the thermal behaviour of the buildings. Natural ventilation (NV) has an important rol in the ventilated façade performance and can be explained by two phenomena: wind-driven ventilation and buoyancy-driven ventilation. While wind is the main mechanism of wind-driven ventilation, buoyancy-driven ventilation occurs as a result of the directional buoyancy force that results from temperature differences between interior and exterior walls. In this contribution, the natural convection thermal effect of an OJVF and its capacity for cooling in summer conditions has been quantified.

On the one hand, as wind velocity was increased over 0.5 m/s no air velocity changes in the ventilated gap due to the buoyancy-driven effect (BDE) could be observed. The velocity increase due to BDE was negligible respect the effect of the wind-driven forces. In the particular case of 1.5m/s wind velocity and 8K temperature gradient between wind and the indoor cladding face, the air velocity increase due to BDE was negligible. Consequently, the behaviour of the air velocity in the ventilated façade mostly depends on outdoor weather conditions. On the other, as wind velocity was decreased under 0.5 m/s, the air velocity was being increased due to the BDE having a higher specific weight in the total air velocity change. Furthermore, in the operating conditions, at low wind velocities (v_{wind}=0.2m/s) as the temperature gradient determining the BDE was increased, the velocity was also proportionally increased. Therefore the BDE became more important in the behaviour of the ventilated façade in terms of air velocity.

In addition, the quantification of the temperature drop in the external face of the building was the most important effect to determine the thermal and energy performance of the building with a ventilated façade passive system. In the modelled conditions (2K temperature drop between the air and the internal cladding face temperature and v_{wind}=0.2m/s), the temperature of the external face of the façade was decreased due to the ventilated façade effect at low wind velocities. The 35% of the temperature decrease was due to natural convection buoyancy-driven effect; and 65% due to wind-driven effect.

In this study the worst case for air movement in the ventilated gap was analysed: the ventilated façade was closed at his bottom. Further research should be done with a ventilated façade opened at its bottom part.

The main conclusions summarized as a points are the following ones:

- At high wind velocities (>1.5m/s) the buoyancy-driven effect is negligible respect the wind-driven effect in summer.
- At high wind velocities the buoyancy-driven effect doesn’t influences the ventilated façade cooling effect in summer.
- At low wind velocities (<0.5m/s) the buoyancy-driven effect has a higher specific weight in the total air velocity change in the façade.
- The more the air velocity in the ventilated façade is accelerated, the more the cooling effect is reached.
- At low wind velocities (<0.5m/s); the higher temperature gradient is, the more cooling by the buoyancy-driven effect is achieved.

Table 4. Nomenclature

Magnitude	**Unit**	**Description**
v_{wind}	m/s	Wind velocity
T_{wind}	K	Wind temperature
v_{air}	m/s	Air velocity in the ventilated gap
v_g	m/s	Velocity grid
T_{air}	K	Air temperature in the ventilated gap

Magnitude	**Unit**	**Description**
T_2	K	External façade layer temperature
T_1	K	Internal face of the cladding layer temperature
T_{indoor}	K	Indoor temperature
e_i	mm	Width of the building
ρ	Kg/m^3	Fluid density
$\vec{v}$	m/s	Fluid velocity
S_m	m^2	Mass source in the control volume
t	s.	Time
p	Pa	Static pressure
$\bar{\tau}$	N/m^2	Stress tensor
g	m/s^2	Gravitational force
μ	Pa·s	Eddy viscosity
F	N	Outher force
q	W/m^2	Surface heat transfer
k	W/m/K	Conductivity coefficient
h	$W/m^2/K$	Convective coefficient
U	$W/m^2/K$	Overall heat transfer coefficient
I		Unit tensor

References

[1] Kolokotroni, M., Robinson-Gayle S., Tanno S., Cripps A. Environmental impact analysis for typical office façades. Building Research and Information. January-February 2002.

[2] Balocco, C. A simple model to study ventilated façades energy performance. Energy and Buildings 2002, Vol 34, Pp 469-47.

[3] Patania, F. ; Gagliano, A. ; Nocera, F. ; Ferlito, A. ; Galesi, A. Thermofluid-dynamic analysis of ventilated façades. Energy and Buildings 2010, Vol 42. Pp 1148 – 1155.

[4] Sanjuan, C. ; Suárez, M.J.; González, M. ; Pistono, J.; Blanco, E. Energy performance of an open-joint ventilated façade compared with a conventional sealed cavity façade. Solar and Energy 2011, Vol 85. Pp 1851 – 1863.

[5] Hussain, S.;. Oosthuizen, P.H. Numerical investigations of buoyancy-driven natural ventilation in a simple three-storey atrium building and thermal comfort evaluation. J. of Applied Thermal Engineering 2013. Vol. 57. Pp 133-146.

[6] Seytier, C., Naraghi, M. Combined Convective-Radiative Thermal Analysis of an Inclined Rooftop Solar Chimney. J. Sol. Energy Eng 2013. 135(1).

[7] Pasztory, Z.; Peralta , P. Peszlen , I. Multi-layer heat insulation system for frame construction buildings. J. Energy and Buildings 2011. Vol 43. N 2–3, Pp 713–717.

[8] Clark, J.; Peeters, L.; Novoselac, A. 2013. Experimental study of convective heat transfer from windows with Venetian blinds. Building and Environment, Vol. 59, N 1, Pp 690-700.

[9] Suárez,M.J. Sanjuan,C. Gutiérrez,A.J.; Pistono,J. Blanco,E. 2012 Energy evaluation of an horizontal open joint ventilated façade, Applied Thermal Engineering 2012, Volume 37.Pages 302-313.

[10] Wilmer, P.; De Carli, M. Evaluation of various CFD modelling strategies in predicting airflow and temperature in a naturally ventilated double skin façade. 2012. Applied Thermal Engineering, 2012. Vol.37, Pp.267-275.

[11] Gratia E, De Herde A. Optimal operation of a south double-skin façade. Energy and Buildings 2004. Vol 36. Pp 41 – 60.

[12] Linden, P.F. The Fluid Mechanics of Natural Ventilation. Annual Review of Fluid Mechanics 2009. Pp 201-208.

[13] Mingotti, N., Chenvidyakarn T., Woods A.W. The fluid mechanics of the natural ventilation of a narrow-cavity double-skin façade. Building and Environment 2011. Vol 46. Pp 807 – 823.

[14] Gratia E., De Herde, A. Is day natural ventilation still possible in office buildings with a double-skin façade? Building and Environment 2004, Vol 39. Pp 399 – 409.

[15] Chen Q. Ventilation performance prediction for buildings: a method overview and recent applications. Building and Environment 2009, Vol 44. Pp 848 – 858.

[16] Lomas, K.J., Cook M.J., Short C.A. Commissioning hybrid advanced naturally ventilated buildings: a US case stud”. Building Research and Information 2009. Vol 37. Pp 397 – 412.

[17] Giancola, E., Sanjuan, C. ; Blanco, E. ; Heras, M.R. Experimental assessment and modeling of the performance of an open joint ventilated façade during actual operating conditions in Mediterranean climate. Energy and Buildings 2012, Vol 54. Pp 363 – 375.

[18] Sanjuan, C.; Suárez, M.J.; Blanco, E.; Heras, M.R. Development and experimental validation of a simulation model for open joint ventilated façades. Energy and Buildings 2011. Vol 43. Pp 3446 – 3456.

[19] González M. Blanco E. Pistono J. Adjusting and energy simulation model by means of CFD techniques to analyze open-joint ventilated façade performance, in: WREC-X, Glasgow. Solar and Energy 2008. Vol 85. Pp 1851 – 1863.

[20] Mora-Pérez, M.; López-Patiño, G.; Bengochea-Escribano, M.A.; López-Jiménez, A. “Quantification of the efficiency of the ceramic ventilated façade by means of computational fluid mechanics techniques”. Boletín de la Sociedad Española de Cerámica y Vidrio 2011. Vol 50. Pp 99-108.

[21] Justus CG, Mikhail A. 1976. Height variation of wind speed and wind distributions statistics. Geophysical. Research Letters 1976. 3(5):261-4.

[22] Marioscini, C.; Strachan, B.; Semprini, G.; Morini, G.L. Empirical validation and modeling of a naturally venilated rainscreen façade. Energy and Buildings 2011. Vol 43. Pp 853 – 863.

[23] Wang L., Wong N. H. “The impacts of ventilation strategies and façade on indoor thermal environment for naturally ventilated residential buildings in Singapore”. Building and Environment 2007. Vol 42. Pp 4006–4015.

[24] Omar S. A., Mohamed B. G. “A comparison between CFD and Network models for predicting wind-driven ventilation in buildings”. Building and Environment 2007. Vol 42. Pp 4079–4085.

[25] Kang J-H, Lee S-J. Improvement of natural ventilation in a large factory building using a louver ventilator. Building and Environment 2008. 43(12): 2132–41.

[26] CD-Adapco. Star CCM+ User’s Manual. 2011.

Annex V

Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation

Abstract: Nowadays building designers have to face up to new strategies to achieve the best sustainable building designs. Well planned natural ventilation strategies in building design may contribute to a significant reduction on building's energy consumption. Natural ventilation strategies are conditioned to the particular location of each building. To improve natural ventilation performance of a building, the analysis of the influence of the location and the surrounding buildings on wind flow paths around the design building is a must. New computational tools such as Computational Fluid Dynamics (CFD) are particularly suited for modelling outdoor wind conditions and the influence on indoor air conditions prior to building construction. Hence, reliable methodologies are necessary to support building design decisions related to naturally ventilated buildings prior to construction.

This paper presents a case study for the selection of the best future building location attending to natural ventilation behaviour inside the building, conditioned by different evolving environment. A validated CFD model is used to represent outdoor and indoor spaces. The methodology explains how to qualitatively and quantitatively analyze wind paths around and through a building to quantify the natural ventilation performance. The best location, from two real possible solutions, is then selected.

Keywords: Natural Ventilation (NV); Computational Fluid Dynamics (CFD); Building Design Stage, urban planning

1. Introduction

For the last 10 years energy efficiency has become more popular, mainly because of the climate change and environmental degradation. There is an increasing necessity to reduce the amount of energy required to provide thermal comfort conditions in buildings. More sustainable buildings are then designed thanks to the development and advances of new design techniques [1].

Ventilation is a must to ensure a high indoor air quality and comfort. Methods for ventilating a building may be divided into mechanical/forced and natural strategies. Mechanical ventilation requires an external input power supplier to work while natural ventilation (NV) relies on wind and thermal buoyancy as driving forces [2]. The potential benefits of NV include lower energy costs compared with mechanical ventilation, as well as an improved indoor environmental quality and occupant satisfaction [3]. Therefore to achieve more sustainable buildings, designers are promoting the utilisation of NV to improve and decrease the building energy consumption. While NV benefits may be achievable, its implementation and design presents certain challenges since it requires careful analysis of many variables.

Natural ventilation, also called passive ventilation, uses natural outside air movement and pressure differences to both passively cool and ventilate building as early stated. The NV approach design requires an appropriate understanding of principles of local wind patterns, climate conditions, airflow around the building, building orientation, pressurization and façade design [4]. Design of naturally ventilated buildings is also influenced by the airflow through the building. Driving pressures derived from wind and thermal buoyancy are low compared to those produced by fans in mechanical ventilation systems. Therefore the buildings shaping, orientation relative to prevailing wind direction(s) and façade design should be considered in the initial design stages of construction to exploit local wind natural forces to drive the air through its interior.

Natural ventilated buildings do not only suggest that natural airflow influences building design, but also that it might be a concept-making factor of the entire project. Building designers should take into account all these variables, even when its quantification before building construction would be not an easy issue. In fact, increasingly new technology methods allow nowadays designers to evaluate different architectural solutions to improve natural ventilation in its design stage. What happens then is that building designers often do not consider NV solutions due to its lack of expertise on using proper technologies to evaluate and implement it. Thus, designer's awareness and understanding of the location potential is mandatory for proper design of a naturally ventilated building which reaches its maximum energy advantages. The relative position of the building with respect to surrounding buildings and prevailing winds may affect the whole performance of the building. Therefore when designers have the option for choosing the building's position and orientation, the most energy efficient solution should be selected. Often, it is not possible to choose the building position or there is no space to rotate the building. Even in these cases, designers should choose the best façade opening distribution to optimize the natural ventilation resources of the location. Anyway, evaluation should be done for each particular case that might contribute to make buildings more site specific.

The evaluation of NV performance pretends to provide information concerning indoor/outdoor airflow parameters in a building. There are many recent applications for predicting the ventilation performance of buildings. Ventilation can be typically predicted or evaluated using a wide range of methods: analytical models, empirical models, small and full-scale experimental models, multi-zone models, zonal models and CFD models. An overview of the methods is summarized by Chen Q. [5], Yuguo L. and Heiselberg P. [6]. Among all of them, computational fluid dynamics (CFD) simulations have been the most popular in recent years [7-10]. CFD has been widely used for simulation of wind and air movement in existing buildings. Literature is profuse in documents based on CFD direct application and experimental validation in order to design wind behaviour of NV in buildings [11-19]. Nevertheless, CFD has not been almost used in building early design stage, in which CFD can help building designer to achieve better and more detailed understanding of issues involving ventilation. Additionally, in this stage, there is no access to full-scale testing facilities although several studies have been done in CFD and reduced-scale models [21-21]. CFD enables building designers to choose the best constructive solutions by simulation techniques and not by trial-and-error experimental techniques, which need much time and economic resources to be performed. Nevertheless, CFD simulations have to be validated to assure that results are reliable.

To simulate correctly the specific conditions of each location, a complex analysis of the building and of the background is necessary. Developing reliable CFD models requires a high level of expertise, which in many cases may not be available [22]. A wide CFD knowledge is essential to well define an indoor computational model and handle complex boundary conditions [23-24]. This is a handicap to be solved in the future. Nevertheless, in recent years, several standards and guidelines have been created to help designers to produce credible and verifiable CFD results [25-27]. In addition, CFD developers are increasingly creating more user-friendly software to allow non-specialized companies to work with CFD. In fact, companies are paying increasing attention to create more and more specialized CFD simulation divisions. Even when high computer power is necessary, parallel simulations help to reduce computational times. Although this methodology involves certain computational capabilities, every day designers are more prepared to integrate engineering analysis with architectural design.

Natural ventilation offers the opportunity of reducing mechanical requirements of Heating, Ventilating and Air Conditioning (HVAC) systems by using the natural driving forces of external wind and the buoyancy effect from internal heat dissipation. Wind patterns around buildings affects ventilation, infiltration rates and associated heat losses through it i.e. energy consumption of the building. With regards to the location of the building, the analysis of surrounding wind effect on the prevailing wind direction will be crucial when designing the building in such way that wind pressures, that will drive air flow through its apertures, will be created to passively ventilate the building. Wind analysis should be locally done, since wind pressure varies considerably due to its interaction with urban context, like buildings, and natural environment. These interactions create complex air flows and turbulence [20].

The present study shows a coupled CFD analysis of the effects of different building locations on the indoor airflow distribution of a naturally ventilated building in an urban environment. Selection of the future location to improve wind usage for NV purposes is carried out from the building design stage. Air velocity, indoor air change rates and air velocity streamlines are analysed to better understand natural ventilation effect in the building. Impact of surrounding environment, including the main buildings, on the air flow behaviour through the building are modeled by CFD. Other aspects, like temperatures and sun heating, are assumed to be constant, while some others are already defined: building shape, façade openings and indoor layout.

The objectives of the study are outlined in section 2. The geometry of the building and surroundings are described in section 3. Section 4 describes the computational model and the followed validation procedure. Section 6 presents the simulation results and its discussion. The paper conclusions are summarized in section 7. Finally, section 8 and 9 conclude the paper with the recommendations for further analysis and acknowledgements respectively.

2. Objectives

Designers have to face up the ability of dealing with complex designs that take into account many requirements to improve building energy performance i.e. local environmental conditions and building surroundings. Designers should choose, when possible, between more or less energetically location alternatives using an objective criterion. One of the most architectural implications on an advanced naturally ventilated building is the consideration of the influence of surrounding environment regarding wind patterns [28].

The present paper follows a design methodology to face up the site selection procedure of a building. Relationship between building location, surrounding environment and natural ventilation is examined. Two available sites for the building are analysed and compared under a NV point of view. The objective is to demonstrate that comparative results of different architectural alternatives should be done to get objective criteria to select building location at the design stage. The methodology summarizes a qualitatively and quantitatively analysis to choose the best building site using a computational fluid dynamics technique through a case study. The computational model is created with a systematic procedure and validated with full-scale measurements to assure credible results.

The primary goal of this study is to offer a better understanding of the building location presuppositions associated with the utilization of natural wind driving forces to improve natural ventilation and therefore building energy efficiency.

3. Description of building and surroundings

The present case study analyzes the design and construction of a modular sustainable and energy-efficient building. The aim of the entire project is to promote awareness of designers, students and society towards energy efficiency design of buildings and promote specialization of the professionals involved in this field. The building has been globally designed so that every construction element has a functional use and a bioclimatic purpose.

The project is structured in three phases. Firstly a theoretical design that fulfills the requirements of the main building standards is done. The paper is focused on the analysis of the best location for using passive strategies such NV to reduce building energy consumption at the design stage. Secondly, the building is constructed attending to initial design requirements. Finally, the validation of the hypothesis and theoretical models developed at the design stage are done.

The analysed building is a rectangular-shaped one floor building (13.77 m. x 5 m.) 4 m. height. The indoor layout is organised and shaped to provide low resistance air paths with two main rooms connected through a small corridor where the bathroom is placed. The main room has an outside door (2.5 m. x 3.5 m.) and the secondary room has a window (1.5 m x 3.7 m), both in the South face. Both rooms have two narrow windows placed in the upper North face (0.5 x 2 m. and 0.5 x 1 m. respectively). There is a vertical window (2 x 0.5 m.) in the building East face. The bathroom, which is placed between the main and the secondary room, has a door to the secondary room and a window in the upper North face (0.5 x 2m.). A sliding door close the corridor between the main room and the bedroom

A natural ventilation concept is therefore integrated in the building body. The building is provided with several passive elements to protect against solar gains such as a pergola at the entrance, louvers elements in the South & East windows and a modular ventilated façade made of modular porcelain stoneware tile systems. The structure of the building is made of steel profiles and a concrete platform 0.5 m. height. There is a machinery room in the building West face, where appliances for heating, air conditioning, ventilation and others are placed.

The building will be placed in Valencia in two possible sites within the campus of the Universitat Politècnica de València (East of Spain). The campus is located in the East of Valencia and very close to the Mediterranean Sea. The city and its surroundings are located on very flat terrain; height differences are limited to less than 3 m. Nevertheless the surroundings buildings heights are from 14 to 34 m. height, which might influence wind patterns and NV effect on the building.

4. Computational model

4.1 Solver settings

A commercial code for CFD is implemented in the present research: Star CCM+. A 3D computational model is created to obtain a mathematical analytical solution of the problem. The Reynolds-averaged Navier–Stokes equations (or RANS equations) are solved in a geometrical domain taking into account the turbulent phenomena. The review literature shows that RANS techniques are more convenient for NV simulation and designing purposes [29]. The standard k- ε model is used for representing turbulence as used in [30]. Segregated flow model steady-state regime is used for the simulation. The gravity model is selected to include the buoyancy source terms in the momentum equations. The model is defined in a unique region with a single mesh so that indoor and outdoor volumes are together computed. Therefore the effect of the nearest buildings on the indoor air distribution is obtained minimizing the computational error that would result from a “decomposed domain” in which indoor and outdoor environments are computed in different mathematical models.

4.2. Computational geometry

The computational domain cuts off the surrounding environment at which wind effect is of main interest. The computational geometry is reproduced in detail using the design drawings and full-scale measurements. The near buildings are only modelled by their main external shape. Other small elements placed near the building are taken into account by imposing a roughness height on their surfaces. The study case building is modelled by its internal and external shape because indoor NV behaviour is of particular concern. However detail of the windows shading devices and indoor furniture distribution are not simulated.

Figure 1 shows the simulated building (front and back view). The main room where the door is placed is 67.7 m^3, the bathroom is 16,48 m^3 and the secondary room is 88.17 m^3. The total indoor volume considered is 112.71 m^3.

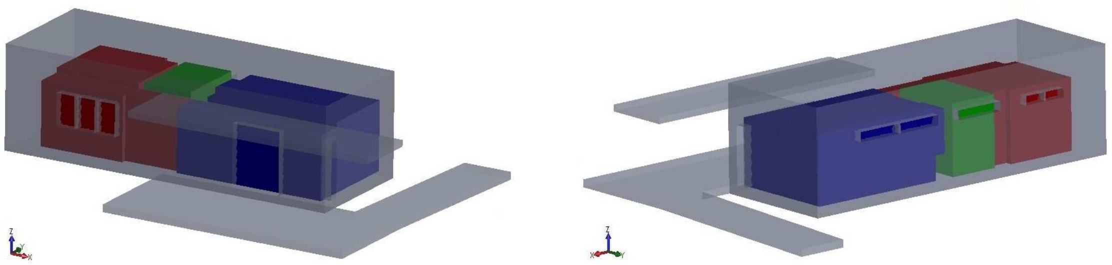


Figure 1. Design building indoor layout (front (left) and back (right) view)

The entire computational domain is designed attending to specific recommendations [31]. The inlet boundary surface is placed $5 \cdot H_{max}$ away from the first building placed before the case study building in the prevailing wind direction, where H_{max} is the height of the tallest building (34 m.). The outflow boundary is placed $15 \cdot H_{max}$ behind the last building modelled in the prevailing wind direction to allow flow development. In order to prevent an artificial acceleration of the flow over the tallest building, the top of the computational domain is $5 \cdot H_{max}$ away from the tallest building. The lateral walls are placed $1.5 \cdot W$, where W is the width of the urban area represented (111.3 m.).

4.3. Boundary conditions

The specification of consistent boundary conditions must be done to achieve a CFD model that truly represents the real environment. In this case, a wind tunnel (WT) is simulated with a velocity inlet profile set in the WT East face. Pressure outlet condition is defined in the WT West face and symmetry planes are set in the WT top, South and

North faces. Finally all the other faces in the model are set as walls. The wind tunnel boundary selection is detailed in the following sections.

4.3.1. Velocity inlet

At the inlet of the computational domain the velocity profile is defined based on the reference velocity measured at the reference height. The available information from the nearby meteorological station in Valencia [32] is used to determine the wind speed at the reference height. The measured wind speeds ranged between 0.0 and 15.8 m/s. Wind speeds are distributed normally with an average value of 3.05 m/s and a standard deviation of 1.47 m/s at 10 m. height for summer months. The wind data is adapted for height using Justus and Mikhail expression [33] (1).

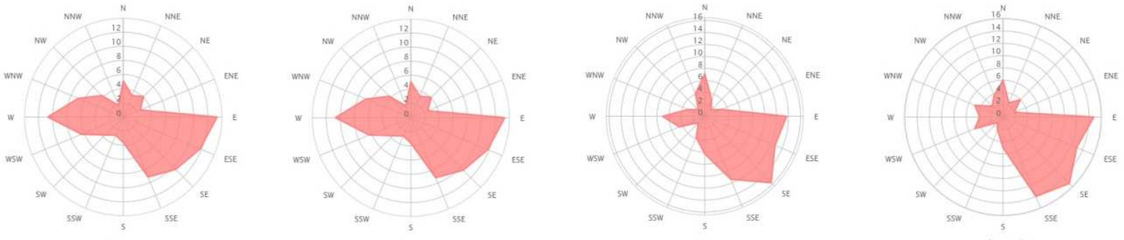
$$\frac{\bar{u}(z)}{\bar{u}_a(z_a)} = \left(z / z_a \right)^n \qquad (1)$$

In which u(z) is the speed at z height, $u_a(z_a)$ represents the average speed measured at z_a height and “n” can be calculated as defined by expression (2). The velocity is measured in m/s and the height in m.

$$n = \frac{0.37 - 0.0881 \cdot \ln(\bar{u}_a)}{1 - 0.0881 \cdot \ln(z_a/10)} \qquad (2)$$

The definition of the wind direction at a given location is not a simple task as wind can blow and fluctuate from any direction. However, statistical information about wind direction can be used to determine the prevailing wind direction. The compass rose follows this idea, so that it is commonly used when analysing the wind resource at a given location. The compass rose represents wind speed and direction typically distributed at a particular location. Knowing the prevailing wind and the effect of nearby buildings at a particular location is essential for NV design. On the one hand, the impact of wind on the building shape creates areas of positive and negative pressure that should be taken into account to improve the NV flow. On the other hand, the presence of other buildings can modify the prevailing wind direction and it can generate turbulence trails that affect the NV flow.

The definition of the velocity inlet direction is done taking into account the wind direction on the warmest seasons in Valencia, in which NV can play an important role on energy consumption reduction of the building. With CFD help, the most suitable location of the building is analysed to assess the viability of the prevailing wind driven flows and the effect of surrounding buildings on NV performance to reduce the building energy consumption in summer. The prevailing wind on the warm seasons in Valencia (summer, autumn and spring) is approximately East-South-East (ESE) direction. Figure 2 shows the wind direction and frequency on the warmest months in Valencia. Therefore ESE direction is defined as the prevailing wind direction in the CFD ***model and set in the WT East face.***



May June July August

Figure 2. Wind Rose at Valencia Meteorology Station on the warmest months in %

The fluctuating approach of wind is not taken into account for the present study as it is very costful and actual resources limit the amount of cases that can be considered. Nevertheless it is important to know this mechanism to predict the ventilation efficiency before design [20]. Further research will be done by the authors to analyze this issue.

Finally, the prevailing wind direction is set in the CFD model using the Justus and Mikhail expression (1) in the WT East face. A complete wind analysis will analyze not only the prevailing wind direction but also the complete wind interaction in the building. Therefore statistical wind information is used to cover the most probable cases. Nevertheless the approach can be repeated to each wind direction and modulus. Different inlet boundary conditions will provide different ventilation results. For the sake of conciseness, only the most probable inlet wind condition has been here depicted.

4.3.2. Pressure outlet and symmetry planes

The face of the computational region placed behind the buildings is defined as the outlet boundary condition (wind tunnel West face). All wind entering through the wind tunnel East face leaves the wind tunnel through the wind tunnel West face. Based on the literature review [34] a relative static pressure of zero is set at the outlet of the computational domain. The top and the lateral walls of the computational domain are prescribed as symmetry condition to enforce parallel flow. To prevent a too strong artificial acceleration of the flow, the blockage ratio obeys the recommendation given in [35], below 3%.

4.3.3. Walls

The computational faces not defined as inlet/outlet and symmetry plane boundary conditions are defined as walls. On the one hand building walls are defined as smooth surfaces and treated as non-slip walls. On the other hand, the bottom surface is defined with a roughness length z_0 =0.5 m. to represent the numerous obstacles as for instance bushes and parklands. The roughness height is based on the updated Davenport roughness classification [36].

To reduce the number of grid points in the wall-normal direction and therefore the computational costs, a wall function is applied as an alternative approach to compute the wall shear stress [31]. A wall treatment is used in the CFD model to place the first computational node outside the viscous sublayer and to make suitable assumptions about how the near-wall velocity profile behaves to obtain the wall shear stress. Since the near-wall sublayer is not resolved, estimating the velocity gradient from a linear variation will not give an accurate approximation of the shear stress at the wall. Certain turbulence models need to solve such layers, so modellers have to make sure that enough data points (prism layers) are present in that region. There are clearly many applications where it is necessary to solve the viscous-affected region. However, if the near-wall mesh resolution is not consistent with the modelling assumptions, significant errors can result. In this case, to considerate it, the y+ value is used, which is a non-dimensional distance (based on local cell fluid velocity) from the wall to the first mesh node. The high-y+ wall treatment implies the wall-function-type approach in which it is assumed that the near-wall cell lies within the logarithmic region of the boundary layer (y+>30) [37]. Correct values of y+ allows a proper assessment of the current CFD model mesh.

4.4. Grid generation and grid independence of the solution

The Finite Volume Method (FVM) is used with a 3D mesh. Computational results depend crucially on the grid that is used to discretise the computational domain in the FVM. Therefore a grid size study is done, to assess grid independence of the solution. The recommendations of Franke et al. are followed to generate the grid [31]. Therefore this grid is created with at least 10 cells per building side and 10 cells per cube root of building. The mesh base size is settled as 2 and 4 m. for the first and the second building position, respectively, with a slow growing rate. Building positions are described in section 5. The computational domain had a maximum blockage ratio of 2,6%, which is below the recommended maximum of 3% [26, 31].

The grid quality should be high for not introducing large errors and for capturing the important physical phenomena in the main volume of interest. A mesh size of 25 mm is set in the near wall regions. The prism layer thickness measures 2 mm. The exterior volume does not require a mesh as fine as in the indoor buildings volume. Figure 3 shows the mesh near the one-floor building represented in a horizontal and vertical plane.

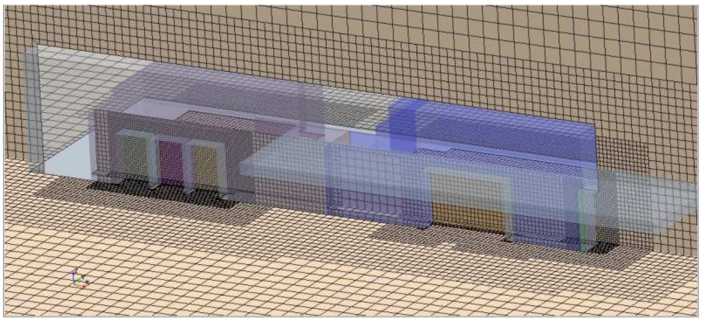


Figure 3. Volume Mesh near the studied region

As introduced before, every numerical solution depends on the grid that is used. Therefore, a grid independence study is done to assess the validity of the numerical results for different grids. Three different size meshes of every model were created: M1 and M2. Ferzinger et al. requirements for consecutive grids [38] are followed. The models are created with different cell accuracy. The first model (M1) has 390025 cells (M1.1), 864801 cells (M1.2) and 1330180 cells (M1.3). The second model (M2) has 766131 cells (M2.1), 1205064 cells (M2.2) and 1762943 cells (M2.3).

The results of the different meshes are compared to analyse the grid independence of the solution. A qualitative analysis is done with velocity streamlines in each model. No appreciable differences are observed for each model on the indoor velocity streamline distribution. A quantitative analysis of this comparison is presented in Figure 4. Two indoor line-probe are placed in the centre of each room to compare the velocity result in each mesh. Graphs results show that for models 1 and 2, the main room centre line-probe results are very similar with a maximum relative difference of 5.2% and 4.8% respectively. The relative difference in the second room centre line-probe of the model 2 is higher than the relative difference in the model 1 at 2.5m. height (see Figure 4). Nevertheless M2.3 mesh computational time is such that this relative difference can be assumed. Therefore, models with fewer cells are chosen to continue further with the research in order to achieve a balance between accuracy and computational time.

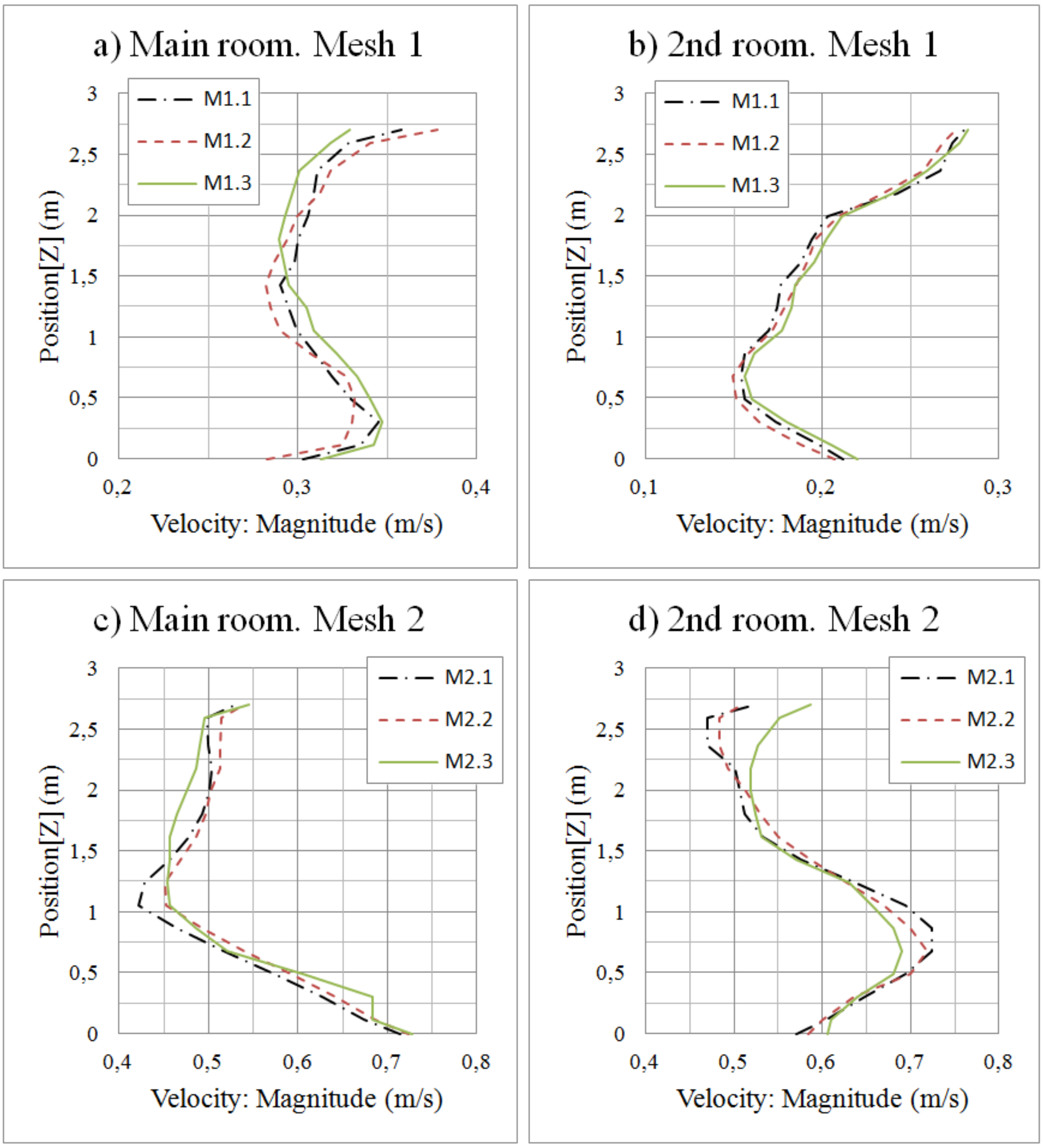


Figure 4. Velocity Magnitude comparison with different mesh sizes. a) Main room with mesh 1; b) Second room with mesh 1, c) Main room with mesh 2, d) Second room with mesh 2

4.5. Validation with full-scale wind velocity measurements

To verify a CFD solution not only the grid-sensitivity analysis is a must but also full-scale wind velocity measurements to prove the reliability of the CFD model. In this case, indoor air velocity measurements cannot be done as the building has not still been constructed. In fact the aim of the research is to choose between the best location for the building attending to NV building performance before constructing it. Nevertheless on-site wind velocity measurements can be done in the place where the building is going to be located approximately to check that the order of magnitude of the CFD results are reasonable, at least in terms of external conditions for the actual location.

The wind measurements are done using three “hotwire” air-speed transmitters (HD403TS). The hotwire air-speed transmitters measure air velocity between 0 and 5 m/s, with an accuracy of ±0.03 m/s. The air-speed transmitters are located in a vertical line at three heights (1 m., 2 m. and 3m.). The location of the sensors is selected taking into account the wind flow patterns so the future building does not influence them. The vertical line is placed three meters in front of the South-West building corner in location 1 and five meters in front of the centre of the future East opening in location 2 as indicated in figure 5. Note that figure 5 shows the case study building virtually placed in the two studied sites. The position of the vertical control lines is outlined (mm). In these locations the wind flows measurement with and without the one-floor building are supposed to be similar.

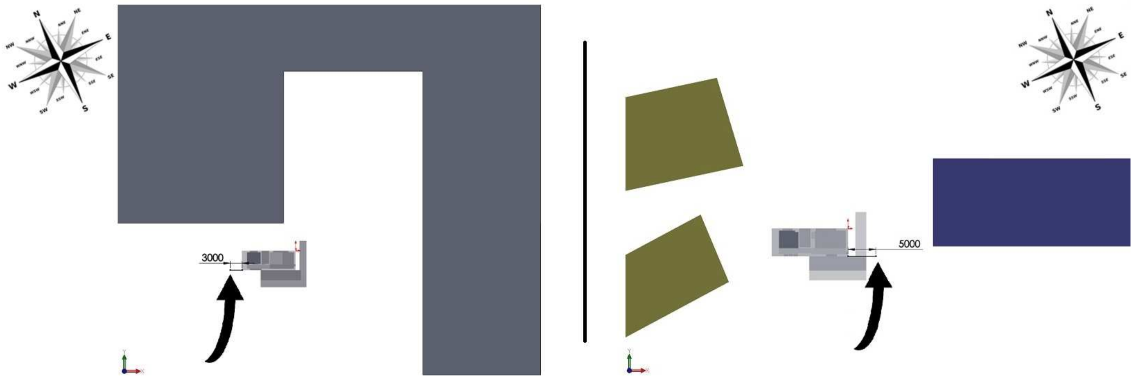


Figure 5. Position of the control lines in Location 1 (Left) and Location 2 (Right) in (mm).

To gather data that supported CFD model validation, the experiment is performed on a typical ESE windy day. Measurement devices are measuring wind velocity during all day, while external wind conditions are monitored by the Meteorology Station [32]. Wind data provided by field experiments represent averages over 10 min or 30 min time intervals. Longer intervals are not feasible, because meteorological conditions already change within 30 min [35]. Then the 30 min. average “hotwire” air-speed transmitters measurements [32] that were more steady (2.93 m/s ± 0.41 m/s.) are chosen to cross data with the CFD results. The validation criterion is determined by 0.20 m/s absolute difference between measured and simulated wind speeds. The CFD model is modified by adjusting grid refinements of the interesting area and boundary conditions, since it is enough to meet the specified validation criteria. Table 1 shows the comparison between simulated and measured wind velocities in the three positions checked for the two building locations. The CFD wind results at z=2 m. (building location 1) and z=3 m. (building location 2) are slightly out of the validation criteria range. Unfortunately several CFD modifications are done but it was not possible to adjust the three measurements at the same time. Nevertheless, it is concluded that the wind velocity order of magnitude between the CFD model and the “hotwire” air-speed transmitters are within the confidence interval of the measurements in all cases.

Table 1. CFD model and “hotwire” air-speed results and measurements

Building location	Height [z] (m)	HD403TS (m/s)	CFD model (m/s)
1	1	0.93 ± 0.12	0.985654
1	2	1.07 ± 0.19	1.190761
1	3	1.24 ± 0.15	1.236461
2	1	1.62 ± 0.14	1.664610
2	2	1.78 ± 0.08	1.850315
2	3	2.08 ± 0.26	2.238317

5. Optimal building location attending to NV achievements

As indicated, the particular case study consists of the analysis of two possible locations under a natural ventilation point of view to improve the future energy performance of one building from the design phase. The one floor small building studied is detailed in section 3, it is located at the Polytechnic University of Valencia. The study shows a coupled 3D CFD analysis of the effects of different outdoor urban environments defined by the surrounding buildings. The research is specially focused on the impact of the mass-distribution of buildings on wind flow patterns and indoor air distribution for the prevailing wind direction.

–

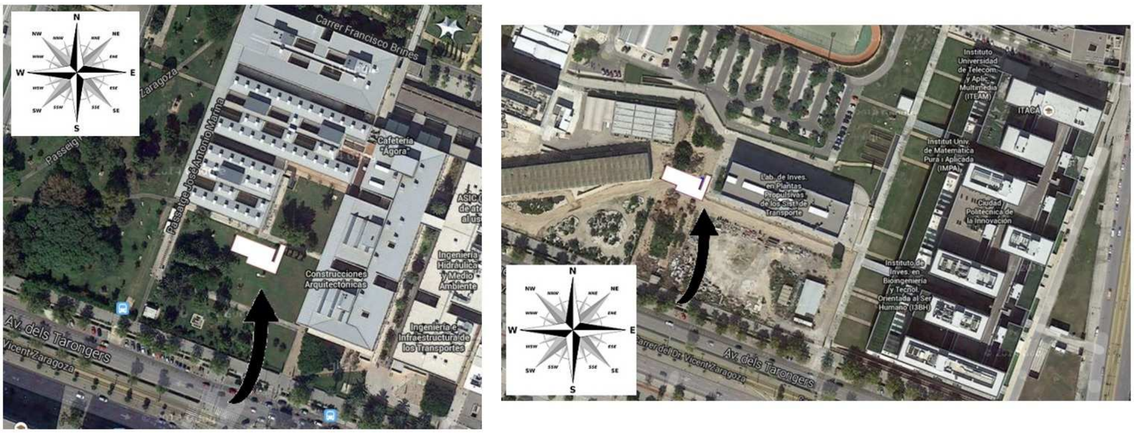


Figure 6. Aerial plan view of the building site options. Left) Location 1. Right) Location 2

The two proposed sites are depicted in Figure 6. On the one hand, it is seen that the immediate surroundings of the location 1 consists of a U-shaped medium building 14 m. height that may block the South-East, North-East and North-West wind direction to the house. The 14 m. height building is placed 32 m. from the East face of the case study building in the East-South-East direction and 7 m. from the North face in the North-North-East direction. There are some big trees in the immediate West, South-West and

South direction that may influence the wind patterns in the building location although there are no big buildings that can block the wind in these directions.

On the other hand, the immediate surroundings of location 2 have a 17 m. height and 65 m. long building placed 15.3 m. from the East face of the case study building in the East-South-East direction. This building may block the North-East wind direction. There are two 5.5 and 7 m. height buildings placed 5 m. from the West face of the case study building in the West-North-West direction. These buildings may block the North-West wind direction. The South-East, South and West-South-West direction have some provisional blocks smaller than 1 m., since it is an area under construction procedure. It is projected a plane green park in this area. This means that these directions will not have any big block for these wind directions. Additionally there is a 34 m. height building placed 124 m. from the case study building in the East-South-East direction that may also influence the prevailing wind patterns [14]. Figure 7 shows the East-South-East view of the two locations from ground level. Figure 8 shows the computational model used.

Figure 7. Ground level NW-SE view of the building location. Left) Location 1. Right) Location 2

The architectural configurations of the buildings are represented as simple blocks as illustrated in Figure 10. Figure 10 shows a South-West view of the two locations. Features smaller than 1 m. are not represented. The case study building of interest is represented with more details to achieve more reliable CFD results as pointed in section 3. All openings are completely opened for the simulation, even though the main entrance is completely closed.

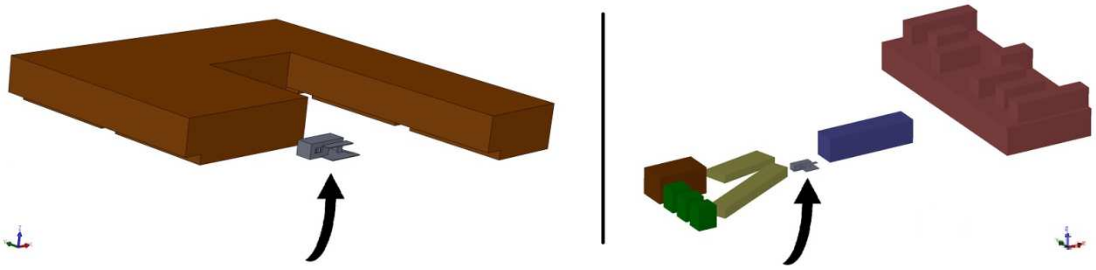


Figure 8.Computational model. Left) Location 1. Right) Location 2

As stated before, the prevailing wind in the warmest seasons in Valencia (summer, autumn and spring) is East-South-East. Therefore a quantitative and qualitative analysis of the prevailing wind patterns affection is proposed to determine the most suitable location to improve NV performance in the future building using CFD.

6. Results and discussion

The purpose of this research is to add to the literature a case study to select the best building location among several options to improve the NV behaviour and hence to reduce the energy consumption before its construction, in a very preliminary design stage. Other factors that also influence the NV behaviour i.e. the building shape, size, orientation and opening positions, are not changed as there are not among the points of interest in this study. Further analysis may be done taking into account these parameters in the future.

Two locations with two different surroundings are evaluated using a reliable CFD model defined and validated in section 4. The CFD model includes the case study one-floor building placed in each environment described in section 5. The validated CFD models are used to assess the influence of the surrounding buildings on the NV behaviour of the building in the prevailing wind direction (ESE). In both sites there are tall buildings than may block the prevailing wind and influence local wind conditions at low level. Nevertheless the potential blocking buildings are placed at different lengths from the case study building. On the one hand the location 1 has a 14 m. height building placed 32 m. in front of the case study building in the ESE direction. On the other hand the location 2 has a 34 m. height building placed 124 m. in front of the studied building in the ESE direction. Both of them can modify the prevailing wind direction influence regarding what NV behaviour concerns. The discussion focuses on the quantitative and qualitative analysis of the wind flows through the house openings of the building for each location.

The CFD simulation allows the visualization of velocity vectors and streamlines to analyze the air flow through the building. The streamlines are traced out paths by massless particles that follow the airflow. Air velocity streamlines allow designers to know from which opening the air flow tends to enter or to go out of the building in an indoor environment and which is the way followed through the building. Dead zones can easily be detected and design measures can be done to avoid them. Additionally, it is visualized how the wind flow is developed near the buildings. At the design stage, innovative measures can be taken to make maximum use of wind flows to reduce energy consumption and to improve occupants comfort.

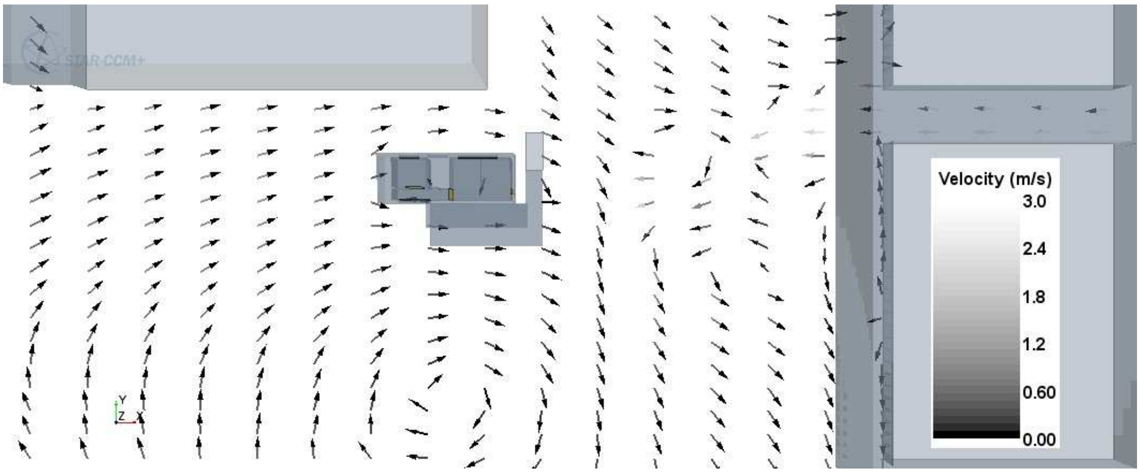


Figure 9. Location 1. Velocity vector visualization at 1.7 m. height horizontal plane

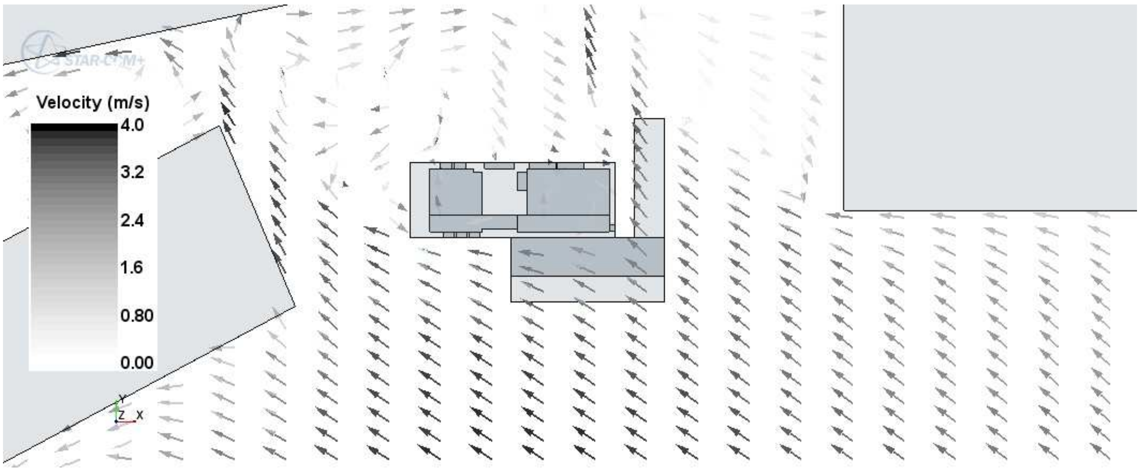


Figure 10. Location 2. Velocity vector visualization at 1.7 m. height horizontal plane

Figure 9 visualizes the velocity vector distribution at a 1.7 m. horizontal plane in the region of interest near the case study building. Figure 9 shows how the near surrounding buildings affect the prevailing wind direction completely changing the wind angle of incidence in the case study building. It is observed that the case study building is placed in a wind recirculation area behind the main surrounding building. For the prevailing wind direction, the wind flow passes near the machine room before entering the building through the narrow North openings. The machine room is placed in the West face of the building. Unfortunately this is not the best wind path desired because the air flow may be warm up before entering the building. In addition the west side of the buildings are the warmest due to local climate conditions. Therefore, these additional heating loads would need to be compensated by a cooling system to maintain a comfortable indoor environment, which may increase the total energy consumption of the building. This effect is not energy efficient desirable as the study is valid for the warm months as mentioned in section 4.3.1. Moreover indoor comfort could be affected due to the excessive warmed air flows.

Figure 10 visualizes the wind velocity flow near the case study building in the location 2. In this case, the wind velocity direction follows the prevailing wind direction as the surrounding buildings do not modify the prevailing wind flow. Opposite than in location 1, where it is observed that the building is placed in a region of high instability due to turbulence produced by the recirculation area behind the surrounding building, the wind flow is completely developed in location 2. This facilitates the natural ventilation of the building in location 2, due to prevailing dominant wind in the direction of the windows, without added turbulences.

Figure 11 and 12 show the streamlines represented near the region of interest, where the case study building is located in the middle of the figures in both simulations. Note that the CFD model does not take into account the pressure differences due to buoyancy forces. For instance, velocity streamlines were not taken into account at high heights. The possible airflow variation due to buoyancy effect is relatively small as there are no temperature change for the small height of the building (less than 5 m.) and therefore it can be neglected. Figure 12 shows that the wind velocity streamlines direction are not modified by the 34 m. height building placed 124 m. in front on the case study building in the East-South-East direction. Consequently the computational model can be simplified for future studies.

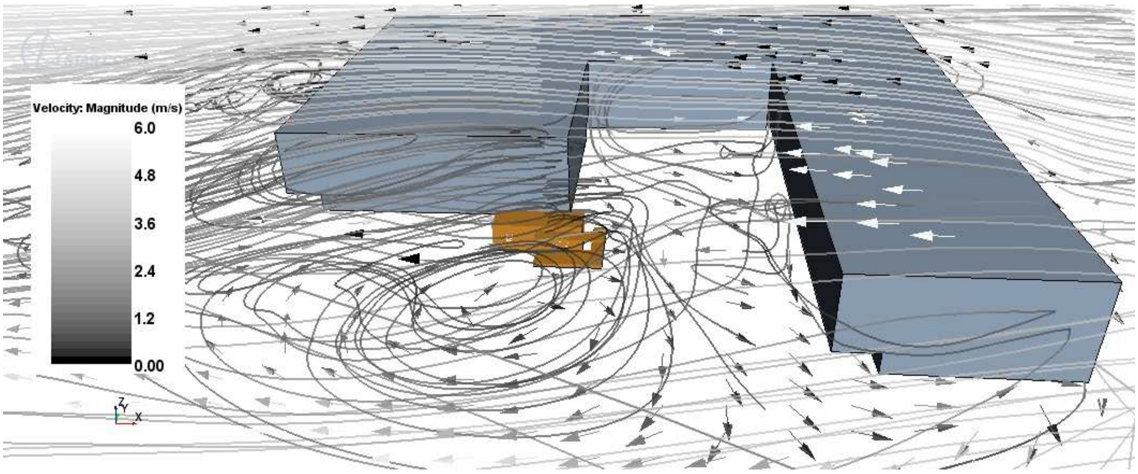


Figure 11. Location 1. Velocity streamlines visualization

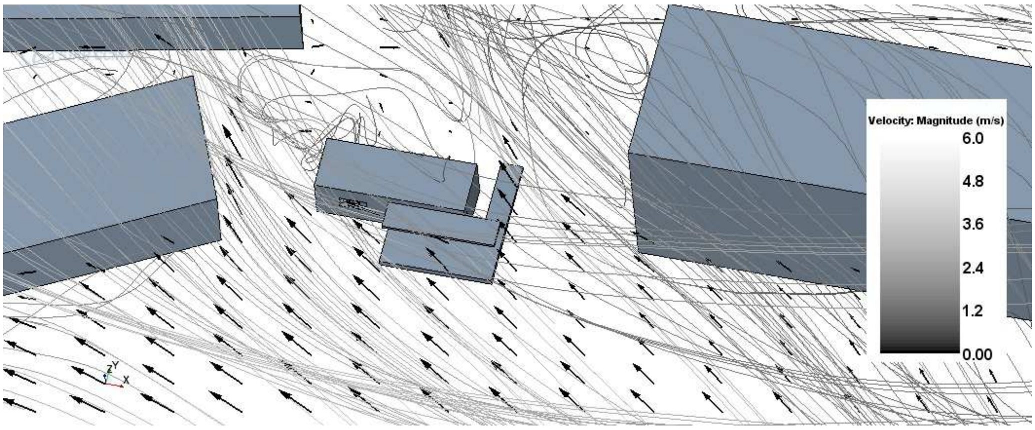


Figure 12. Location 2. Velocity streamlines visualization

Figure 13 and 14 depict the velocity streamlines at the building in location 1 and 2 respectively from the ESE direction. On the one hand the CFD visualization shows a complex flow pattern inside the building placed in location 1. It is observed that the wind flow enters the building through the narrow North openings, as explained before, and the East opening. The air flow leaves the building through the South opening of the building. The flow in the building does not completely swept away the West room.

Moreover it is difficult to determine a specific air pattern in this room. In fact, several dead zones appear in the room (North-West and South-West corners).

On the other hand, there is an air swept effect from the South to the North face of the building that covers the whole building in the case study in location 2. All the wind clearly enters through the South openings and the West opening and leaves the building through the North openings. The South-facing openings create a low pressure which helps to drive the clean air inside the building. From the NV point of view, the completely sweep effect produced in the building in location 2 ensures better indoor air replacement.

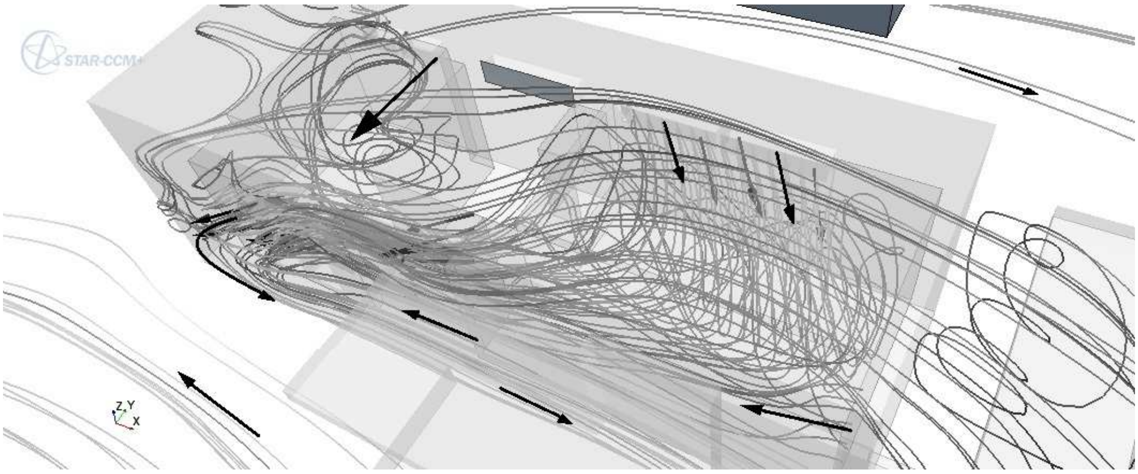


Figure 13. Location 1. Velocity streamlines visualization in the near zone of the building

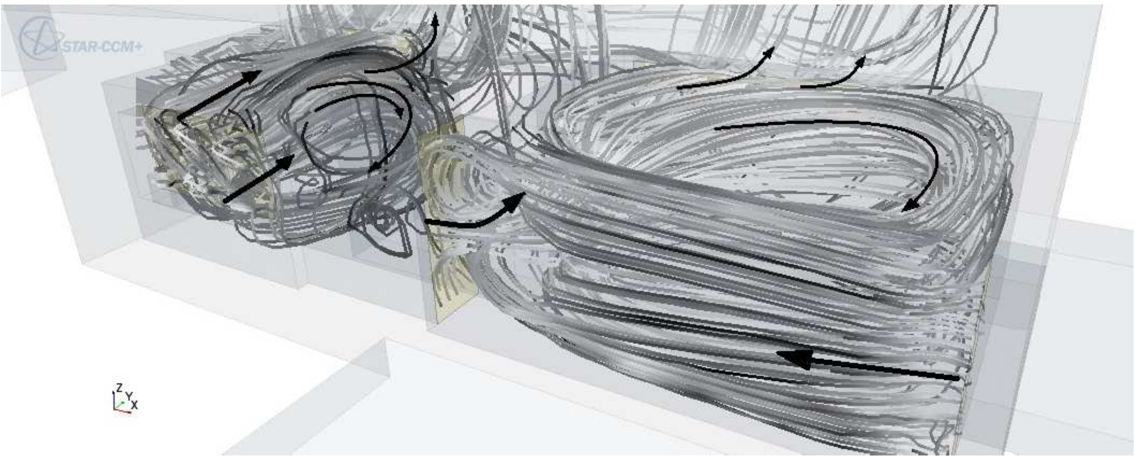


Figure 14. Location 2. Velocity streamlines visualization in the near zone of the building

Velocity streamlines allow designers to have an idea about the air flow development in the building for the prevailing wind direction. Therefore designers can improve their designs based on the location that better enhance natural ventilation. Moreover additional quantitative analysis must be done to compare results and determine which of the two locations could improve NV.

One way to analyze the indoor airflows is by a measure of the air replacement time (t in seconds), which is the time that indoor air needs to be completely renovated. This time is calculated dividing the building volume (V in m^3) and the fluid density (ρ in kg/m^3) by the total air flow rate (f in kg/s), as indicated in expression 3.

$$t = \frac{V \cdot \rho}{f} \qquad (3)$$

Natural ventilation is much more useful as less time needs the indoor air to be renovated. To quantify the effectiveness of wind driven flows for ventilation, inlet and outlet air flow rates through the building are determined and compared for each location. To avoid computational errors the time result is divided to calculate the portion of time needed to renovate the indoor air. In this case, the building placed in location 2 needs 13.1% less time to renovate the indoor air than the building in location

1. This is explained because in the location 2 the prevailing wind enters directly through the South and East openings, which are bigger than the narrow North openings for which the wind flow enters the building in location 1.

Therefore it is concluded that although the air velocities through the building in location 2 are similar than in location 1, the air sweep effect is more effective for the building in location 2. Moreover higher levels of comfort may be achieved due to fewer air temperatures in the building in location 2 in summer and mid-season conditions. It is concluded, then, that the location 2 allows a better performance of the building under a NV point of view for the prevailing wind in summer.

7. Conclusions and future works

When the location of a building can be chosen, designers should study the influence of the urban environment in terms of energy efficiency at the design stage. One of the local conditions that influences the energy behaviour of the building is natural ventilation (NV). A well naturally ventilated designed building can improve the energy consumption and contribute to acquire more comfortable and healthier indoor conditions. Designers should be capable of making the best decisions based on measurable and comparable parameters and scenarios under an energy efficiency point of view before building construction.

The paper provides a detailed and systematic impact analysis of the prevailing wind and the urban environment on the NV behaviour of a case study building at the design stage by computational fluid dynamics (CFD) techniques. The case study is a one-floor building that is going to be built in Valencia (Spain). All of the calculations use the same building dimensions and shape, while maintaining the opening positions and sizes constant. The paper is focused on the NV approach. The aim of the paper is to show an applied methodology in a case study to help building designers to easily improve the building’s energy design and behaviour using advanced computational tools at the design stage.

The study determines the basic wind flow distribution around the surrounding buildings and the air flow distribution in the case study building for the purpose of NV in a quiescent and steady state environment. In this case NV is assessed by determining the evolution of the complex outdoor wind flow patterns in the indoor air flow distribution. The CFD model is defined accurately, it is validated to assess the reliability of the results and it is used to compare the NV behaviour of a building placed in two different locations at the design stage. NV is quantitatively measured by the time that the air volume needs to be renovated. NV is qualitatively analysed by the velocity vectors and streamlines and its visualization in the indoor environment.

The paper shows that the wind patterns are different depending on the surrounding environments and that there can be quite different from expected. Local studies for each site are then required. It is concluded that one of the locations could improve up to 13.1 % the NV behaviour of the building only considering the prevailing wind direction. Moreover the air path coming into the room passed through the machine room in one location, which can produce an undesirable increase of the air flow temperature entering the building. This point is not energy efficient desirable as more energy would then be required to maintain comfortable indoor conditions in summer.

Therefore, building designers can make better design decisions adding high added value to the sustainable building designs. The following conclusions can be drawn from the study:

- The analysis of the urban environment when designing sustainable buildings can improve the building energy efficiency behaviour in terms of natural ventilation.
- Natural ventilation performance needs to be quantified to optimize building and to reduce energy consumption in buildings at the design stage.

- Computational fluid dynamics is a feasible technique to determine the wind flow path development around a building prior to its construction. This technique can be used as a virtual laboratory to simulate any wind conditions and to establish any sort of behaviour in future buildings, even in initial steps of the design process as the present one.

The present study provides the initial observation of wind flows around buildings prior to construction and the analysis of the natural ventilation implications using CFD techniques. The ultimate aim of the work is to add case study aiding with the design process of naturally ventilated buildings.

Although the study has been performed in a metropolitan background, interesting analysis can be done for all kind of scenarios comparing, for example, metropolitan, country and see front scenarios. For that, the CFD model has to be changed by introducing the new building surroundings and boundary conditions for the same building.

As mentioned in section 7, the study is performed for a steady state outdoor environment. However, in a real case wind conditions change and these results are not so straightforward. Therefore, fluctuating approach of wind is mainly required to be covered in the future as different boundary conditions. It is also recommended a complete study of the windows operation i.e. which windows should be closed or opened to achieve the best natural ventilation performance depending on the wind direction and outdoor conditions. The study of the indoor comfort conditions under a NV point of view should be also performed. In any case, CFD techniques are a promising strategy to address all these considerations, when sufficient computational resources are available.

Compared with other similar subjects, the investigated design decisions of naturally ventilated buildings at the design stage have less detailed results. Therefore, there is a requirement for further researches on real case studies that design innovative solutions for naturally ventilated buildings. Then, when buildings are constructed, test should be done to check the reliability of previous designs. Consequently new test procedures can be developed to improve building designs under the point of view of energy efficiency from its design stage.

Acknowledgements

The present study was carried out within the frame of "**E**DIFICACIÓN **E**CO **E**FICIENTE, E3” (*E3 echo efficient building design*) research programme, which was partly financed by CDTI: “Proyectos Tecnológicos de Empresas” (*companies technical projects*) and a consortium of companies formed by BECSA, Rockwool Peninsular, CERACASA, ApliCAD and ATERSA.

References

[1] Fonseca i Casas, P., Fonseca i Casas, A., Garrido-Soriano, N., Casanovas, J. (2014). Formal simulation model to optimize building sustainability. Advances in Engineering Software 69, 62-74.

[2] Linden, P.F. (2009). The Fluid Mechanics of Natural Ventilation. Annual Review of Fluid Mechanics. 201-208.

[3] Zhiwen L., Jianing Z., Jun G., Lixia H. (2007). Estimating natural-ventilation potential considering both thermal comfort and IAQ issues. Building and Environment; 42(6), 2289–98.

[4] Suresh B.S., Srikanth M., Robert F. B. (2011) Passive building energy savings: A review of building envelope components. Renewable and Sustainable Energy Reviews 15, 3617–3631

[5] Chen Q. (2009), Ventilation performance prediction for buildings: A method overview and recent applications. Building and Environment 44, 848–858.

[6] Yuguo L., Heiselberg P. (2003). Analysis methods for natural and hybrid ventilation: a critical literature review and recent developments. International Journal of Ventilation; 1(4), 3–20.

[7] Martin L., James A., Heiselberg P., Yuguo L., (2006) Stathopoulos T. Achieving natural and hybrid ventilation in practice. International Journal of Ventilation; 5(1), 115–30.

[8] Khan N., Yuehong S., Riffat Saffa B. (2008) A review on wind driven ventilation techniques. Energy and Buildings 40(8), 1586–604.

[9] Ray SD, Gong N-W, Glicksman LR, Paradiso JA. (2014) Experimental characterization of full-scale naturally ventilated atrium and validation of CFD simulations. Energy and Buildings; 69, 285–91.

[10] Mora-Pérez M., López-Patiño G., López-Jiménez P.A. (2014) Quantification of ventilated façade effect due to convection in buildings. Buoyancy and wind driven effect. Researches and Applications in Mechanical Engineering 3.

[11] Moosavi L., Mahyuddin N., AbGhafar N., Ismail M.A. (2014) Thermal performance of atria: An overview of natural ventilation effective designs. Renewable and Sustainable Energy Reviews 34, 654–670

[12] Foucquier A., Robert S., Suard F., Stéphan L., Jay A. (2013) State of the art in building modelling and energy performances prediction: A review. Renewable and Sustainable Energy Reviews 23, 271–288

[13] Adamu Z.A., Price A.D.F., Cook M.J. (2012) Performance evaluation of natural ventilation strategies for hospital wards - A case study of Great Ormond Street Hospital. Building and Environment 56, 211–222.

[14] James L., Atila N. (2012) Cross ventilation with small openings: Measurements in a multi-zone test building. Building and Environment 57, 377–386.

[15] van Hooff T, Blocken B. (2010) On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium. Computational Fluids; 39, 1146–55.

[16] Visagavel K, Srinivasan PSS. (2009) Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening using CFD. Solar Energy; 81(1), 2 –5.

[17] Wei Y. (2010) Potential model for single-sided naturally ventilated buildings in China. Solar Energy 84, 1595–1600.

[18] Ji Y., (2007) Cook M.J., Hanby V. CFD modelling of natural displacement ventilation in an enclosure connected to an atrium. Building and Environment 42, 1158–1172.

[19] Wang L., Wong N.H. (2007) The impacts of ventilation strategies and façade on indoor thermal environment for naturally ventilated residential buildings in Singapore. Building and Environment 42, 4006–4015.

[20] Ji L, Tan H, Kato S, Bu Z, Takahashi T. (2011) Wind tunnel investigation on influence of fluctuating wind direction on cross natural ventilation. Building and Environment; 46, 2490 –9.

[21] Van Hooff T, Blocken B, Aanen L, Bronsema B. (2011) A venturi-shaped roof for wind-induced natural ventilation of buildings: wind tunnel and CFD evaluation of different design configurations. Building and Environment; 46, 1797–807.

[22] O’Grady W, Keane M. (2006) Specification of an IFC based software application to support CFD simulation, In: The 11th international conference on computing in civil and building engineering, Montreal, Canada.

[23] Hajdukievwicz M., Geron M. Keane M.M. (2013) Formal calibration methodology for CFD models of naturally ventilated indoor environments. Building and Environment 59, 290–302.

[24] Chen Q, Zhai Z. (2003). The use of computational fluid dynamics tools for indoor environmental design. In: Advanced building simulation, Taylor & Francis; p. 119–40 [chapter 5].

[25] Hajdukievwicz M., Geron M. Keane M.M. (2013) Calibrated CFD simulation to evaluate thermal comfort in a highly-glazed naturally ventilated room. Building and Environment 70, 73–89.

[26] Tominaga Y, Mochida A, Yoshie R, Kataoka H, Nozu T, Yoshikawa M, et al. (2008) AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings. Journal of Wind Engineering and Industrial Aerodynamics; 96(10–11), 1749–61.

[27] Norris S.E., (2010) Appropiate boundary conditions for computational wind engineerg in models revisited. The Fifth International Symposium on Computational Wind Engineering (CWE2010) Chapel Hill, North Carolina, USA. May 23-27

[28] Lomas J.K. (2007). Architectural design of an advanced naturally ventilated building form. Energy and Buildings 39, 166–181

[29] Ray SD, Gong N-W, Glicksman LR, Paradiso JA. (2014). Experimental characterization of full-scale naturally ventilated atrium andv alidation of CFD simulations. Energy and Buildings 69, 285–91.

[30] Shao. J., Liu J., Zhao J. (2012). Evaluation of various non-linear k-epsilon models for predicting wind flow around an isolated high-rise building within the surface boundary layer. Building and Environment 57, 145–155

[31] Franke J, Hellsten A, Schlünzen H, Carissimo B. (2007). Best practice guideline for the CFD simulation of flows in the urban environment.

[32] www.windfinder.com.
http://es.windfinder.com/windstats/windstatistic_valencia.htm

[33] Justus CG, Mikhail Amir. (1976). Height variation of wind speed and wind distributions statistics. Geophysical. Research Letters; 3(5), 261–4.

[34] Evola G., Popov V. (2006). Computational analysis of wind driven natural ventilation in buildings, Energy and Buildings 38, 491–501.

[35] Franke, J., Hirsch, C., Jensen, A.G., Krus, H.W., Schatzmann, M., Westbury, P.S., Miles, S.D., Wisse, J.A., Wright, N.G., (2004) Recommendations on the use of CFD in wind engineering. In: van Beeck, J.P.A.J. (Ed.), Proceedings of the International Conference Urban Wind Engineering and Building Aerodynamics, von Karman Institute.

[36] Wieringa J. (1992). Updating the Davenport roughness classification. Journal of Wind Engineering and Industrial Aerodynamics 41(1), 357–68.

[37] CD-Adapco Star CCM+ User’s Manual.

[38] Ferziger, J.H. and Peric, M. (2002). Computational Methods for Fluid Dynamics, Springer Verlag, Berlin Heidelberg New York, 3rd edition.

Annex VI

A CFD Study for evaluating the effects of natural ventilation on indoor comfort conditions

Abstract: There is an increasing interest in improving energy efficiency in buildings due to the increased awareness about environmental impact and energy cost. Natural ventilation is an environmentally friendly technique which has become more attractive way for reducing energy use while it also provides acceptable comfort conditions. The research shows a case study building in which the natural ventilation effect due to wind-driven forces on indoor comfort conditions is evaluated. Moreover, the architectural solutions selected during the building design phase to improve the natural ventilation behaviour are successfully validated in a full-scale building. The indoor comfort conditions are evaluated through contrasted performance indicators: draught risk (DR), predicted percentage of dissatisfied people (PPD) and predicted mean vote (PMV) indexes. The results show that air movement due to natural ventilation allows increasing indoor air temperature maintaining the initial comfort conditions. Therefore, the mechanical air conditioning use can be postponed until the indoor air temperature is high and would, consequently, reduce the total building energy consumption. Thereby, a proper natural ventilation focus during the initial design stage could improve the building energy efficiency without compromising the indoor comfort conditions.

Keywords: natural ventilation; energy efficiency; comfort conditions; computational fluid dynamics

1. Introduction

The increasing global concern about the environment has increased the demand for energy efficient buildings during the last few decades. In this sense, the use of passive mechanisms is being promoted to take advantage of natural energy resources. Complementally, a certification procedure has been developed in order to regulate a given prominence to sustainable designs [1]. Innovative design methods and solutions are becoming more popular to achieve environmentally friendly buildings [2,3]. In this framework, natural ventilation (NV) has become an increasingly sustainable method for reducing the buildings energy operational cost. Natural ventilation is a passive mechanism that takes advantage of wind energy resources to achieve acceptable comfort conditions in buildings. Moreover, lower operative energy consumption as well as improved indoor environmental quality are included in the potential benefits of NV [4]. NV is based on pressure differences to exchange indoor air with outdoor air without any mechanical system. The system relies on pressure differences caused either by wind or by buoyancy forces.

The implementation of natural ventilation systems presents certain challenges, especially in the systems in which the wind driven effect ought to be maximized. In this sense, the main challenge is to minimize the draught risk and ensure comfort conditions. Designers have to ensure that the initial design solutions made during the design phase will work once the buildings have been built. NV has been traditionally investigated for more than 50 years using experimental techniques [5]. However, as experimental measurements cannot be done before building construction; software and numerical methods are necessary to simulate outdoor and indoor environments in order to predict NV behaviour [6,7] and ensure acceptable comfort conditions [8]. Consequently, traditional design techniques should be combined with innovative methodologies based on numerical methods.

Mathematical methods based on computational fluid dynamics (CFD) have become one of the most used techniques to determine NV flows recently [9]. CFD allows testing several building design solutions avoiding the full scale construction considering different environmental conditions, which results in an efficient design method. Despite CFD allows simulating environmental conditions, results must be validated with experimental measurements. Experimental results are a must to ensure the reliability of a CFD model; otherwise, wrong conclusions could be obtained regarding the building energy performance. Moreover, the importance of CFD validation before the building construction is that once the building is built, no other architecture solutions can be assessed. Accordingly, results obtained from simulations must be reliable enough to ensure a proper initial building design.

Simulation models have mainly been used in existent buildings to assess energy efficiency solutions [2,10-12]. Moreover, there are less preliminary NV behaviour studies in residential buildings. However, the NV assessment is a well-established practice for important buildings such as hospitals and high-rise buildings [13-18]. The present research is focused on the indoor comfort conditions evaluation in a case study building that had been designed following a NV design strategy [13,19]. The strategy is based on considering the effect of the local wind, the neighbouring buildings and the building orientation and openings design. The naturally ventilated building is designed taking into account the conditions for acceptable indoor environments of the most relevant standards [20-22]. Nevertheless, it may happen that the predicted NV strategies do not fulfil the initial design criteria [10]. Therefore, the validation of the effect of the initial architectural alternatives selection should be a must.

In the present case, the architectural alternatives have been selected to take maximum profit of the wind driven forces. The one-floor building configuration, which is detailed in the next section, makes the wind-driven forces the main ventilation drivers instead of the buoyancy effect. Complementarily to the natural ventilation thermal effect analysis, as presented in [23], the present contribution is only focused on the analysis of the wind-driven forces effect on comfort conditions and its potential energy savings.

2. Materials and Methods

The present research is conducted in a full-scale building in which the indoor comfort conditions are assessed through CFD techniques. The full-scale building is mainly used for the validation of the CFD simulations.

2.1.Full-scale building description

The case study building is a one-floor building (13.77 m × 5 m), 4 m high. Figure 1 shows the case study building.

Figure 1. Case study building outside view (south view).

Figure 2 shows the internal layout of the case study building and each façade orientation. The figure also represents the fluid region used in the indoor computational model. The indoor has two rooms connected by a short corridor. The main room (A) is connected to the secondary room (B) through the corridor (C). The main room has an outside door south-south-west (SSW) oriented. It is also provided with a 2 × 0.5 m vertical lateral windows east-south-east (ESE) oriented. Figure 3 shows the main room indoor layout. The secondary room has three 1.5 m × 1.2 m windows in the SSW façade. Each room has narrow windows 0.5 m high placed in the upper part of the north-north-east (NNE) façade. The opening shape and position in the building is the result of the NV design strategy followed by Mora-Pérez et al. [19].

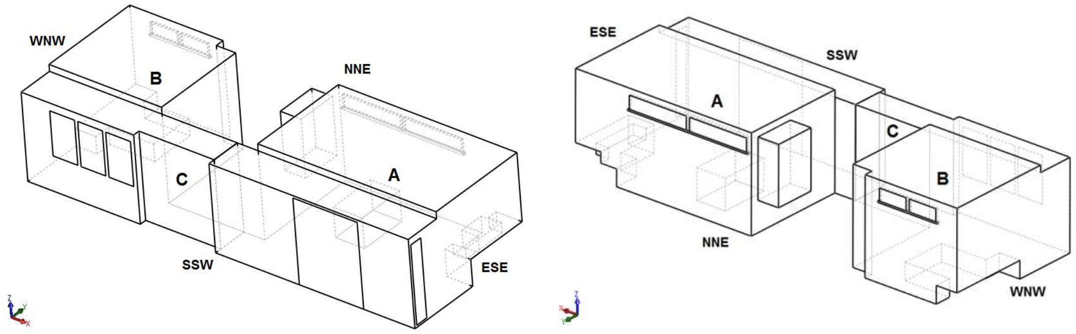


Figure 2. Indoor computational model fluid region.

Figure 3. Case study building indoor view. Main room. (SSW-NNE view).

The building is located close to the Mediterranean Sea in Valencia (Spain). The location and orientation of the building is the result of the NV design strategy followed by Mora-Pérez et al. [13]. The immediate surroundings consist of a 17 m high and 65 m long building located 15.3 m from the ESE building face and two 5.5 and 7 m high buildings placed 5 m from the WNW building face. Figure 4 shows the case study building location and orientation, the near buildings and the surrounding area.

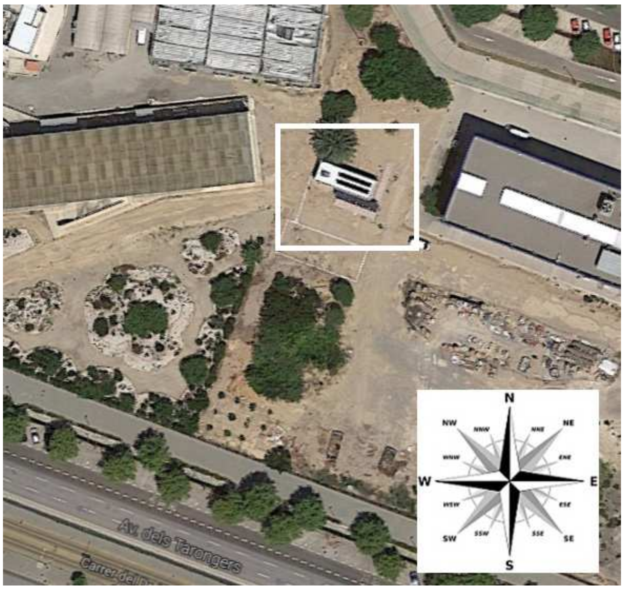


Figure 4. Case study building location and orientation.

2.2.Computational model definition

A commercial CFD software (Star CCM+) is used to predict the indoor air flow behaviour and the comfort indicators in the building. The software simulates the outdoor and the indoor environment in a coupled domain. The outdoor environment is composed only by the relevant surrounding buildings described in the previous section. The indoor environment is represented in more detail because it is the area of interest. In this area, the CFD software calculates and visualizes comfort condition indicators such as draught risk (*DR*), predicted mean vote (*PMV*) and predicted percentage of dissatisfied people (*PPD*) in a 3D volume.

2.2.1. Computational geometry and mesh

In order to reach a balance between detail level and computational time, the geometry that represents the surrounding environment is simplified [24]. The effect of some elements such as trees placed leeward and small elements have already been considered in the terrain roughness height and the wind profile boundary conditions definition that are detailed in the next section. The indoor layout is represented in more detail because the air behaviour in the building is of particular concern. The main static furniture such as the sofa, the bed and the kitchen appliances are modelled in the indoor environment. Figure 5 shows the case study building and the three surrounding buildings previously described.

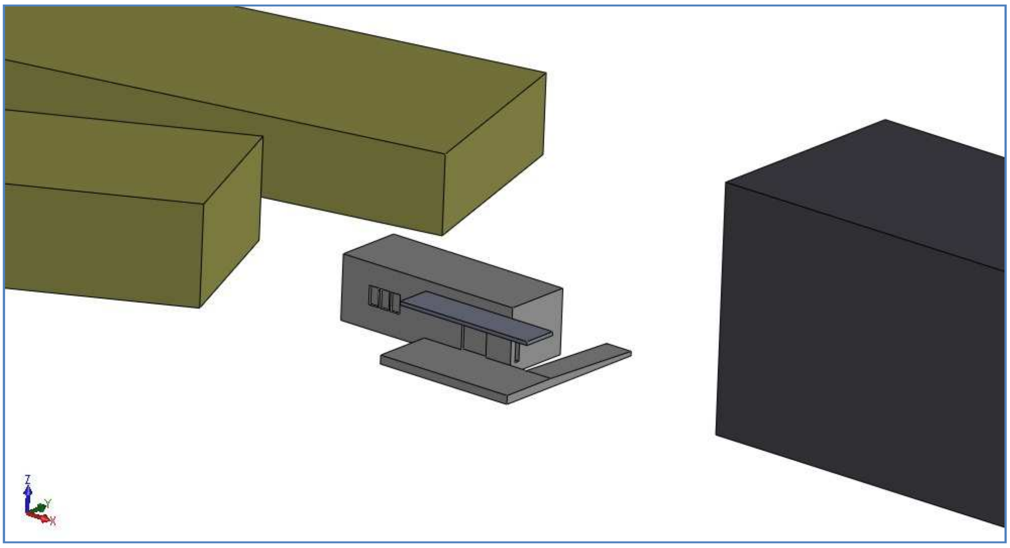


Figure 5. Outdoor CFD model.

The outdoor volume is enclosed within 6 walls in which the boundary conditions are defined [13]. The velocity inlet boundary surface is 5 H_{max} away from the case study building in the prevailing wind direction, where H_{max} is the height of the tallest near building (17 m). The outflow is located following the same rule 15 H_{max} in the opposite side and 1.5 D each lateral boundary, where D is the width of the urban area represented. The top boundary is 5 H_{max} away from the tallest building. The domain is divided into small cells using the trimmer volume mesh, which is particularly suitable for modelling aerodynamic flows due to its ability to refine cells in wake regions [24]. The mesh near the indoor area of interest (≤25 mm) is tighter than the mesh away from the area of interest (≥4 m). In order to verify that the results are independent of cells number, several simulations with different size meshes are performed, i.e., reducing the cell size following the CFD recommendations [25]. There is a limit in which the mesh size reduction does not improve the computational error, whereas the computational time is highly increased. An error of 4.8% appears when comparing CFD results for velocities with different mesh accuracies, while computational time is 54% higher [13]. Therefore, error of 4.8% is assumed as adequate to reach an equilibrium balance within computational time and results accuracy. In this case, the selected model has 766.131 cells [13]. Figure 6 shows the indoor mesh in a horizontal plane.

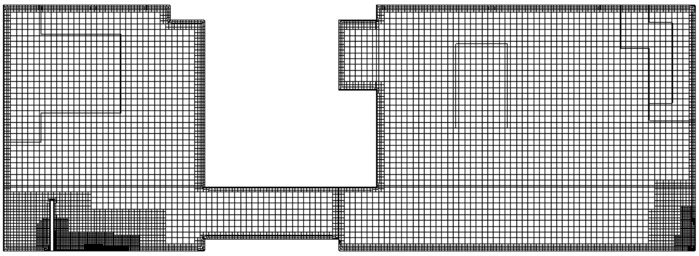


Figure 6. Volume mesh in a horizontal plane. Height = 1.4 m.

2.2.2. *Boundary conditions*

The outdoor environment is enclosed by 6 boundary conditions set in the limits of the outdoor domain. The wind is simulated by using a velocity inlet profile (east face boundary). Pressure outlet condition simulates the wind sink that is opposite to the velocity inlet boundary (west face boundary). Both lateral and top faces are considered as symmetry planes (south, north and top faces) to enforce parallel flow. The updated Davenport classification is used to define the roughness height of the ground (h_0 = 0.5 m) [26].

The velocity inlet profile is defined taking into account the wind measurements done from the Universitat Politècnica de Valencia and Valencia meteorological stations [27]. It is quite time-consuming to cover all indoor air scenarios coming from all wind direction so a statistical analysis is done in order to choose the most probable wind direction during summer period: ESE [28]. Furthermore, wind velocities are approximately normally distributed; the wind mean velocity is 3.05 m/s and the standard deviation is 1.47 m/s. Then, the simulations are run with three different wind velocities modulus: the average value 3.05 m/s and the average value plus minus the standard deviation, 1.58 m/s and 4.52 m/s. The air is considered ideal gas.

The power law equation is used to calculate the vertical wind speed profile (1) that is shown in Figure 7.

$$\frac{U(z)}{U(z_r)} = \left(\frac{z}{z_r}\right)^{\alpha} \tag{1}$$

where $U(z)$ is the wind speed at height z (measured in m), $U(z_r)$ is the reference wind speed at z_r = 10 m height and α is the power law exponent [29]. The power law exponent definition is complex because it varies with such parameters as day time, wind speed, temperature, surface roughness and some other mechanical and thermal mixing parameters [29]. In any case these parameters could be mainly classified depending on the surface roughness or on the atmospheric stability [30]. Some authors have proposed empirical methods for calculating it [29]. In this case the selected exponent calculation was proposed by Justus and Mikhail and is function of velocity and height so it depends on the site [31]. The power law exponent is defined by eq (2).

$$\alpha = \frac{0.37 - 0.088 \times \ln(U_r)}{1 - 0.088 \times \ln(z_r/10)} \tag{2}$$

where U is given in m/s and z_r in m.

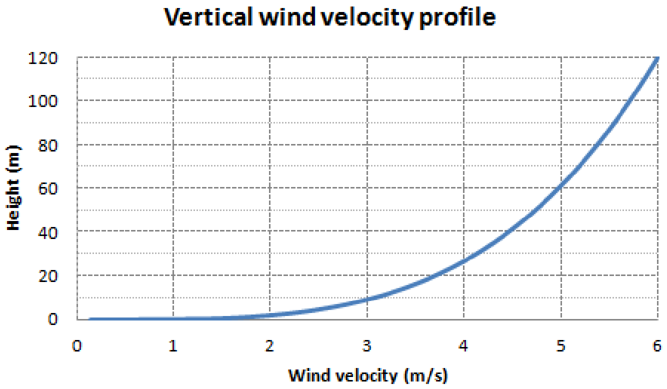


Figure 7. Wind velocity profile.

2.2.3. Solver settings

The mathematical definition of the problem is done through Reynold-averaged Navier-Stokes equations as recommended for NV purposes [32]. The equations are solved in a 3D domain. The modelling technique discretizes the 3D volume in small cells in which mass and momentum conservation equations are solved. K-epsilon and the segregated flow model are used to represent the turbulence [33]. The mass conservation equation is solved by the software (3).

$$\frac{\partial \rho}{\partial t} + \nabla \rho \vec{v} = S_m \tag{3}$$

where ρ stands for the fluid density, t is the time, $\vec{v}$ is the velocity and S_m represents the mass contained in the control volume. Navier-Stokes momentum equation is considered as (4).

$$\frac{\partial(\rho \vec{v})}{\partial t} + \nabla \rho(\vec{v}\vec{v}) = -\nabla p + \nabla \bar{\bar{\tau}} + \rho \vec{g} + \vec{F} \tag{4}$$

where p stands for the static pressure, $\bar{\tau}$ the stress tensor defined by eq (5) and $\vec{g}$ and $\vec{F}$ represent the gravitational and outer forces respectively. μ is the eddy viscosity and I is the unit tensor.

$$\bar{\tau} = \mu \times \left[\left(\nabla \vec{v} + \nabla \vec{v}^{T} \right) - \frac{2}{3} \times \nabla \vec{v} I \right] \tag{5}$$

The discretization of the volume is done following the finite volume method [24].

2.3. Indoor comfort indexes analysis strategy

The balance between NV performance and comfort conditions in the occupied spaces is analysed. CFD techniques are utilized to evaluate the indoor comfort conditions initially. *DR* is assessed as the most common cause of local discomfort. The draught caused by air velocity produces an undesired local cooling feeling in the human body [20]. The results of the indoor air velocity are reported in the current paper.

Indoor temperature can be maintained slightly higher than desired to make more energy efficient buildings in summer conditions. The increase in temperature may lead to an air speed increase to achieve similar comfortable feeling, although high air velocity may cause draught discomfort, especially when the air flow may occur on the nude parts of the body. According to Orosa-Garcia research [34], the highest indoor air velocity should be 0.9 m/s in summer conditions. Moreover, *DR* calculation takes into account human heat loss because of air flow. The amount of heat loss depends on the average air velocity, turbulence and temperature. The percentage of people predicted to be dissatisfied due to draught is calculated according to eq (6).

$$DR(\%) = \left(34 - T_a\right) \times \left(v - 0.05\right)^{0.62} \times \left(0.37 \times v \times T_u + 3.14\right) \tag{6}$$

where v stands for the air velocity (m/s), T_a is the air temperature (ºC) and T_u is the turbulence intensity (%). The boundary conditions for this formula are: $20 < T_a$ (ºC) <

26; $0.05 < v$ (m/s) < 0.5 and $0 < T_u$ (%) < 70. Turbulence intensity represents the ratio between average air speed and speed fluctuation (7).

$$T_u = \frac{v_{RMS}}{v} \times 100 \qquad (7)$$

in which average air speed and speed fluctuation are computed as following:

$$v_{RMS} = \sqrt{\frac{1}{3} \times \left(v_x'^2 + v_y'^2 + v_z'^2\right)} = \sqrt{\frac{2}{3} \times k} \qquad (8)$$

$$v = \sqrt{v_x^2 + v_y^2 + v_z^2} \qquad (9)$$

where k is the turbulent kinetic energy (m^2/s^2).

Secondly, *PMV* and *PPD* indoor comfort indexes are evaluated [35]. *PMV* predicts the mean value of thermal sensation votes of a large group of persons on a 7-point scale (+3 too hot, 0 neutral and −3 too cold), eq (10). *PPD* predicts the percentage of a large people group to feel dissatisfied according to thermal conditions, eq (11).

$$PMV = \left(0.303 \times \exp\left(-0.036 \times M\right) + 0.028\right) \times L \qquad (10)$$

$$PPD = 100 - 95 \times e^{\left(-0.03353 \times PMV^4 - 0.2179 \times PMV^2\right)} \qquad (11)$$

where M is the metabolic energy production; L is the thermal load on the body expressed as eq (12).

$$\begin{aligned} L = {} & M - \frac{3.05}{1000} \times \left(5733 - 6.99 \times \left(M - W\right) - Pa\right) - 0.42 \times \left(M - W - 58.15\right) \\ & - \frac{1.7}{1000} \times \left(5867 - Pa\right) - 0.0014 \times M \times \left(34 - Ta\right) \\ & - 3.96 \times 10^{-8} \times Fcl \times \left(\left(Tcl + 273\right)^4 - \left(Tr + 273\right)^4\right) - Fcl \times hc \times \left(Tcl - Ta\right) \end{aligned} \qquad (12)$$

where *W* is active work, *Tcl* is the clothing surface temperature and *Fcl* is the clothing area factor. The clothing area factor should be according to the thermal clothing insulation (*Icl*) and vice-versa. *Ta* is the ambient temperature in each region of the control volume and *Tr* is the average radiant temperature. *Pa* is the partial pressure of water vapour (considering 50% of relative humidity). *hc* is the convective heat transfer coefficient (internal flow, turbulent flow). The clothing surface temperature is calculated by eq (13)

$$Tcl = 35.7 - 0.028 \times (M - W) - 0.155 \times Icl \times \begin{pmatrix} 3.96 \times 10^{-8} \times Fcl \times \left((Tcl + 273)^4 - (Tr + 273)^4\right) \\ + Fcl \times h_c \times (Tcl - Ta) \end{pmatrix} \quad (13).$$

Indoor comfort conditions are divided in three categories according to indoor air temperature and *PPD*, *PMV* and *DR* indexes among others [21]. Table 1 shows the indoor comfort conditions evaluation criteria.

Table 1. Thermal environment categories in summer conditions [21].

Category	Indoor air temperature, (ºC)	Predicted percentage of dissatisfied, *PPD*	Predicted mean vote, *PMV*	Percentage of dissatisfied due to draught, *DR*	Maximum mean air velocity, (m/s)
A	24.5 ± 1.0	<6	$-0.2 < PMV < +0.2$	<15	0.18
B	24.5 ± 1.5	<10	$-0.5 < PMV < +0.5$	<20	0.22
C	24.5 ± 2.5	<15	$-0.7 < PMV < +0.7$	<25	0.25

The strategy consists in checking whether similar comfort indexes that are reached with lower air temperatures could be achieved by the only effect of natural ventilation. Therefore, it should be assessed whether equivalent comfort conditions to lower air temperatures are reached with higher air temperatures and higher air velocities due to NV. Consequently the indoor target temperature would be increased maintaining the comfort conditions and improving the energy performance of the building. Three different wind temperatures and three different wind velocities are used to compare *DR*, *PMV* and *PPD* comfort indexes. The indoor air temperature is set at 26, 25.5 and 24.5 ºC in each simulation according to Table 1 category limits [21] and field measurements during the warm season. All the registered temperatures in the measurement interval were included within this interval. Moreover, 26 ºC is used as non-common indoor air temperature because it is higher than the maximum air temperature allowed in the national regulation [36]. The air temperature is fixed in the formula of each comfort condition index. However, the wind velocity boundary is set at three different values for each air temperature in order to cover a wide velocity range and to determine its impact on indoor comfort conditions: 3.05 m/s, 1.58 m/s and 4.52 m/s.

Comfort conditions are analysed in the spaces in which occupants are usually located. The occupied zone is confined by vertical and horizontal planes [37], the vertical planes are placed 0.5 m from the internal walls and 1.0 m from the external windows and doors. The horizontal planes are placed 0.05 m (lower boundary) and 1.8 m (upper boundary) above the floor. Special draught and temperature agreements are done in transit zones, in which it could be difficult to meet the thermal comfort requirements. Figure 8 shows the occupied zone considered in the simulations.

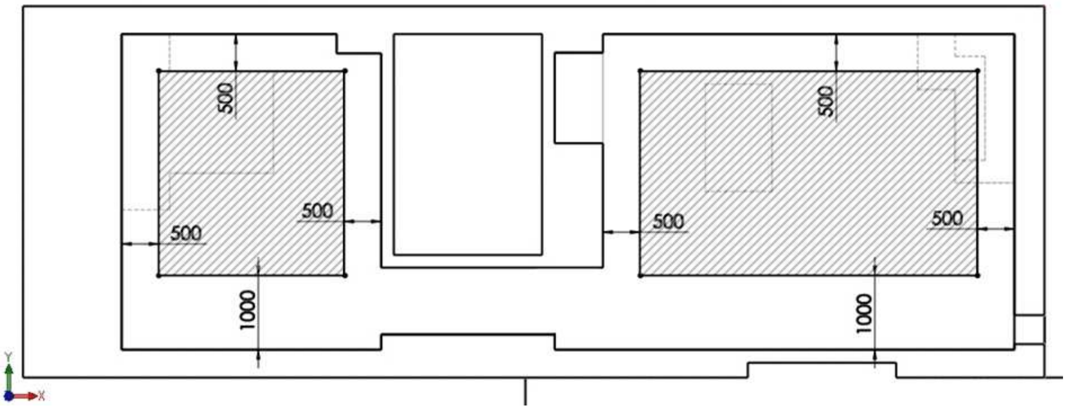


Figure 8. Occupied Zone in the building (units in cm).

2.4. Full-scale measurements

Experimental measurements must be done to validate the CFD simulation reliability. The simulation is validated by comparing the CFD numerical results with experimental measurements. This validation strategy has been successfully used by different authors [7,15,31,38-40]. The indoor velocity sensors are selected to measure the air velocities accurately. The simulated wind speed has a nominal value of 3.05 m/s on a typical windy day during the warm season. Consequently, hot wire air-speed sensors ranged between 0 and 5 m/s are selected. The equipment has an accuracy of ±0.03 + 0.2% m/s (+ 0.2% is the percentage respect the full scale measurement). Figure 9 illustrates the wind measurements considered in a typical ESE direction windy day with an average wind speed near 3 m/s [27]. The period of time in which the indoor measurements could be compared with the CFD calculations are the periods considered quasi-stead-state. That is when the wind fluctuations are not higher than ±20% of the average value during 10 to 30 min intervals. Longer intervals are not feasible because the outdoor conditions are considered steady for less than 30 min [41]. In this case, the period between 11:00 and 12:20 h is considered as a quasi-stead-state period of time.

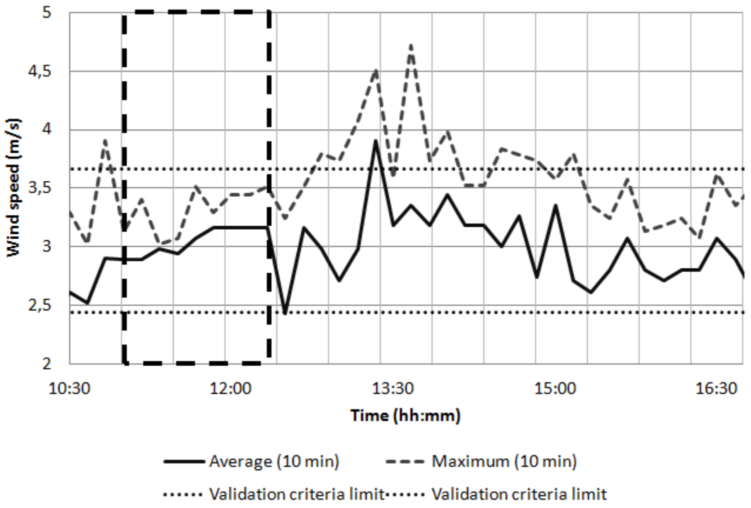


Figure 9. Typical daily wind profile in the warm season [27].

The indoor measurements are done along six vertical axes in order to determine the air vertical profile in the building. Moreover, the air velocity measurements should be focused on the place in which the human body is more sensible to air flow changes. Thus, the sensor height is chosen according to the position of the nude parts of the human body: 0.5, 1.2 and 1.7 m. These heights are carefully chosen to represent the main nude human body parts location while seated and on foot (face, hands and lower legs). Figure 10 shows the positions where the vertical axes are placed in the building layout.

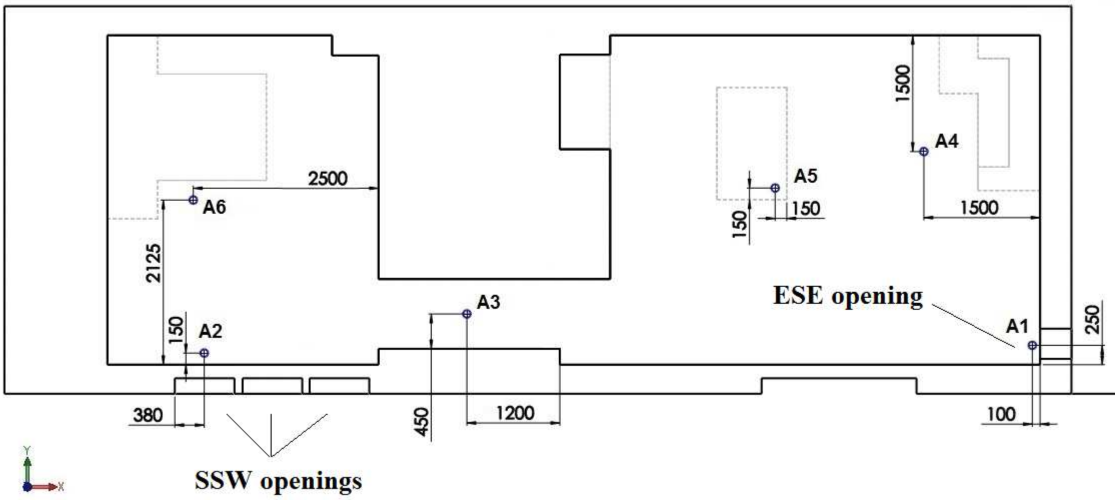


Figure 10. Vertical axes sensors location (units in mm).

Position A1, A2 and A3 are used to know the air velocity through the vertical ESE window, the left SSW window and the corridor, respectively. A4 and A5 are placed near commonly used areas in the main room; and finally A6 is located in the secondary room.

Among the multiple opening configurations that could be set in the building, only the most favourable is used to analyse the comfort indexes in the building: Vertical ESE and left SSW windows are opened (90º), NNE upper windows are opened (20º), central and right SSW windows are completely closed. Other opening configurations should be modelled to analyse different ventilation rates and comfort indexes in the future.

3.Results and Discussion

Indoor full-scale measurements are used to determine the real NV behaviour and validate the CFD simulation. Then, the simulation is used to analyse human comfort under different natural ventilation conditions.

3.1.CFD indoor model validation

The validation is carried out comparing full-scale measurements and CFD calculations. The measurement validation criteria is set at 0.04 m/s absolute difference between measured average air speed and simulated air speeds. This velocity value accounts for 9.70% of the measured average air speed, which is smaller than 10% as indicated in [38]. Figure 11 shows the CFD calculations together with the full-scale measurements. The CFD results are represented along the complete building height (vertical lines) and the full-scale measurements are represented at each point by a horizontal line (it shows the measurement range, minimum and maximum values) and the nominal value. The indoor measurements are taken when the average wind speed meets the conditions depicted in section 2.4. Moreover, the indoor measurements are done in some points according to section 2.4. The graphed results show a good agreement between the calculated and the measured air velocities. All average measurements accomplished the validation criterion except from A1 measurements, only for the lowest point.

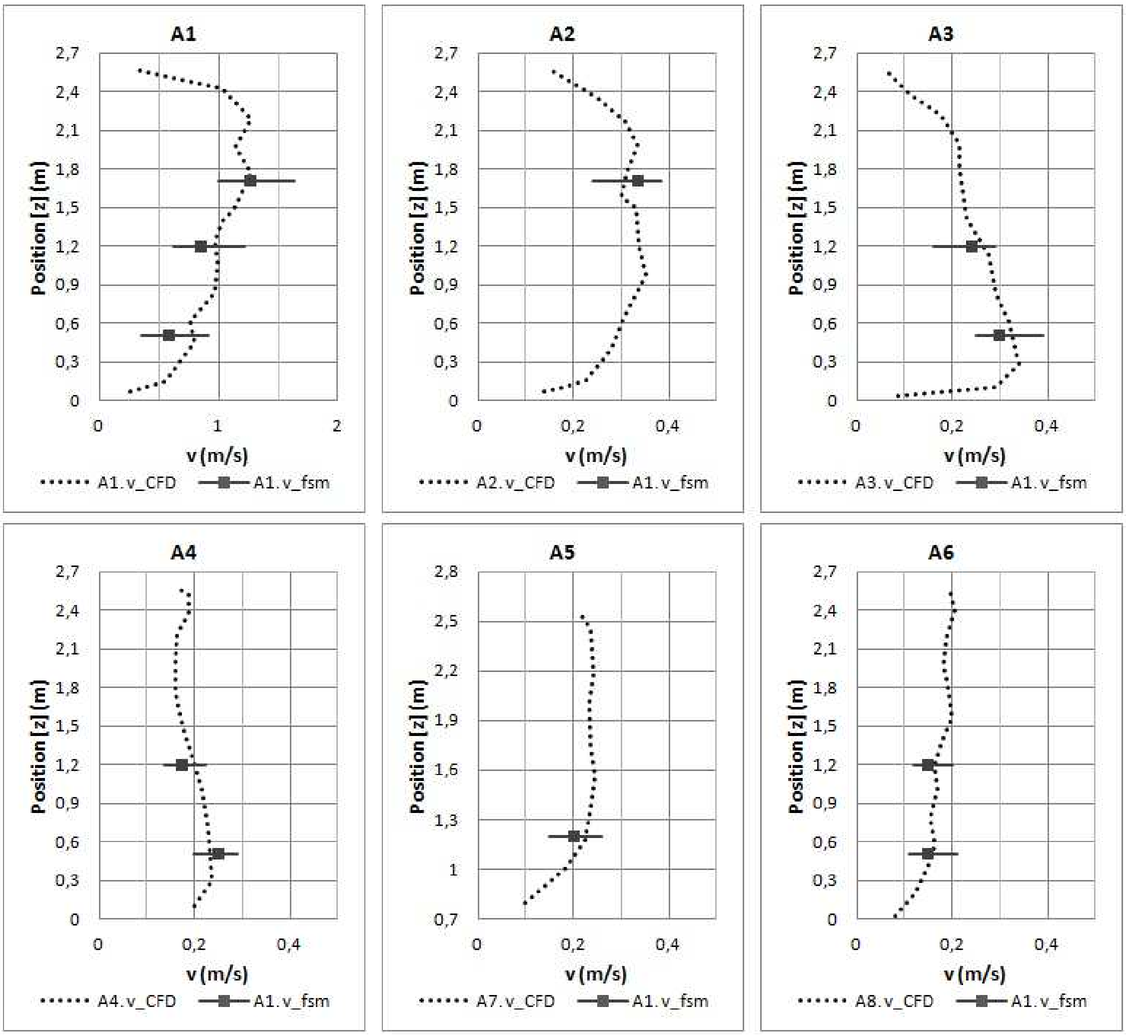


Figure 11. CFD and full-scale measurements comparison. Axes 1 to 6.

Figure 11 shows the air velocity profile entering into the building in each vertical axe. A1 air calculations at height lower than 0.9 m are slightly higher than expected. The reason could be the position that is near the vertical window placed in the prevailing wind direction. This is a vertical narrow window from which the complete renewal air enters into the building. Therefore, higher velocities in this non-occupied zone are expected. Consequently, the validation limits are too narrow for the air speed measured at A1 position. Nevertheless, the maximum deviation of measurements rises up to 11.58% at this position, which is an acceptable percentage of deviation taking into account the inlet air velocity variability. Analysing the maximum mean air velocities shown in Table 1, A1 measurements are not acceptable as air speed ranges from 0.3 to 1.71 m/s and the maximum allowed mean air velocity is 0.25 m/s. Nevertheless, higher velocities than 0.4 m/s take place mainly in the non-occupied zone as shown in Figure 12, which shows 0.15 m/s as average air speed in the occupied zone. Figure 12 shows the air velocity modulus distribution in the occupied and non-occupied indoor volume. The CFD cells that accomplish each velocity condition are grouped in Figure 12.

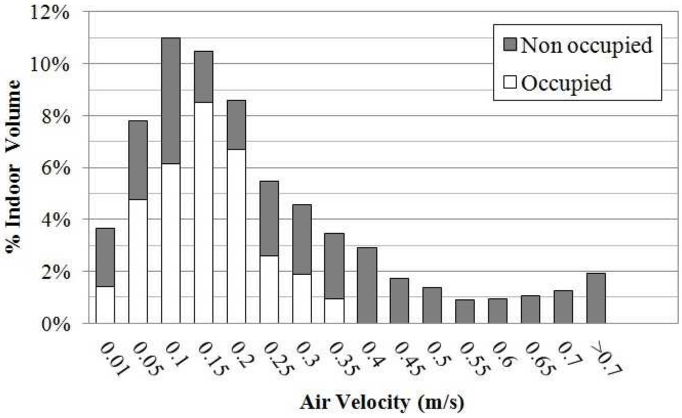


Figure 12. Air velocity distribution in the indoor volume.

A2 position is placed near the opened window in the secondary room from which indoor air comes out of the building. Although measurements are done in a non-occupied area near the window, air velocity results are near the maximum mean air velocity for indoor environment category C.

The corridor is a delicate area regarding comfort conditions approach. It is placed between the main and the secondary room and it is aligned with the air coming from the main air inlet (ESE vertical window) in the prevailing air direction (ESE). Consequently, air velocities in the corridor are expected to be slightly higher than the maximum allowable value for category C [20]. Figure 11 shows A3 air average velocities ranged between 0.18 and 0.33 m/s. The corridor is a passageway and the air velocity deviation is reasonable in comparison with the inlet air velocities, so it is considered as acceptable.

Additional measurements are done to assess the behaviour of the computational model with the real scenario in the occupied zone (A4, A5 and A6). Figure 11 shows a suitable air prediction. Besides, measurements are lower than the maximum mean air velocity value (category C) [20].

To quantify the difference between the values predicted by the CFD model and the values actually measured (A1 to A6), the root-mean-square error (RMSE) is used. RMSE is defined by the eq (14).

$$RMSE = \sqrt{\sum_{i=1}^{N} \frac{\left(\phi_i - \phi_{i,obs}\right)^2}{N}} \tag{14}$$

where $Ø_i$ is the predicted value for *i* point, $Ø_{i,obs}$ is the mean measured value for *i* point and *N* is the number of analysed points. The deviation in relative terms is acceptable, with RMSE = 7.49. The discordances between the measurements and the models are lower than 7.5.

In conclusion, the compared data have a physical sense and the air velocities magnitude order between the CFD model and the full-scale measurements are within the validation criterion established [33].

3.2. Indoor comfort conditions analysis

Natural ventilation creates a particular indoor environment. On the one hand, NV is an efficient method to improve the indoor comfort conditions. On the other, NV could lead to high air velocities that may cause discomfort. Moreover it could have significant effects on the indoor temperatures distribution producing temperature fluctuations that may cause undesired comfort conditions. In this particular case, comfort conditions are evaluated through indicators such as *DR*, *PMV* and *PPD* under steady conditions. The indoor conditions dynamic changes and its fluctuation implications on comfort conditions should be assessed in the future.

Regarding the *PMV*, it depends on occupant and environmental parameters. It considers the occupants' physical activity (metabolic rate of people who work in an office with light activity, M = 93 W/m^2) and the thermal resistance of their clothing (during the summer period the thermal clothing insulation considering no active work, *Icl* is set at 0.5 clo and the corresponding clothing area factor *Fcl* is set at 1.15). The convective heat transfer coefficient is set at hc = 4.6863 W/m^2·K. Air temperature values should be set in the indexes formulas. The limiting temperatures of each indoor environment category described in Table 1 are used to cover a wide range of possible environmental conditions (24.5 ºC, 25.5 ºC and 26 ºC in each simulation). The external wind velocity boundary condition is set at three different values for each air temperature in order to cover a wide velocity range and to determine its impact on indoor comfort conditions: 3.05 m/s, 1.58 m/s and 4.52 m/s.

3.2.1. *DR* analysis

First of all, draught risk is lower for people feeling warmer and higher for people feeling cooler for the whole body [21]. Figure 13 shows the *DR* distribution in the occupied zone (in percentage). *DR* is kept below Category A indoor environment limit (15%, for the tested conditions). *DR* remains concentrated around *DR* = 3% with a temperature increase from 24.5 ºC to 26 ºC for lower air velocities (1.58 m/s). When air velocity is increased, *DR* rate is spread in the indoor environment. *DR*, ranged between 5% and 13%, is distributed in lower volume percentages (between 12% and 16%). Then *DR* comfort index has no critical values to achieve a comfortable indoor environment (*DR* < 15%).

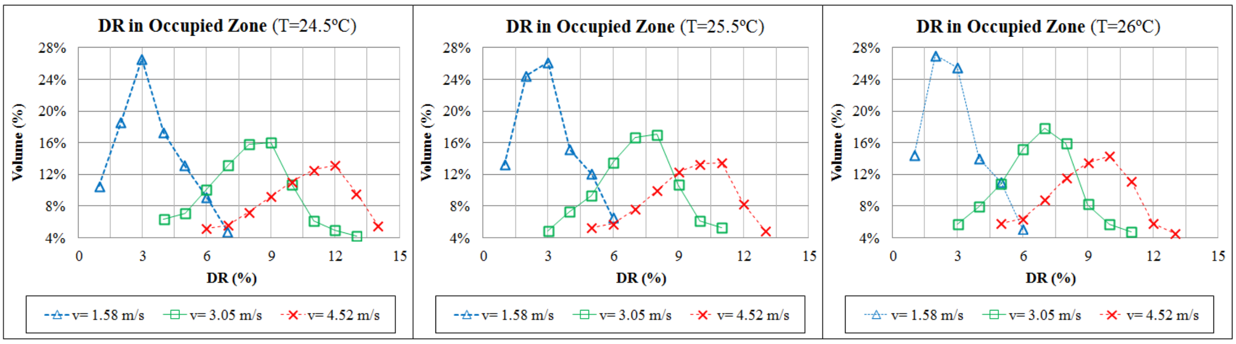


Figure 13. Draught Risk distribution in occupied zone.

The highest tested indoor air temperature does not influence *DR* as shown in figure 13; the three graphs have approximately the same air velocity distribution shape for each temperature. *DR* is almost concentrated at *DR* = 3% for *v* = 1.58 m/s; *DR* follows a normal distribution with an average value at *DR* = 8% for *v* = 3 m/s and *DR* = 11% for *v* = 4.5 m/s.

Nevertheless, the higher *DR* is calculated in the non-occupied zone. Figure 14 shows how the highest *DR* is located in front of the vertical ESE window, near the main room WNW wall and along the passageway. Although higher *DR* is achieved in these places, *DR* is lower than 20% in the whole building. Then, the complete building is classified as Category B attending to *DR* calculation.

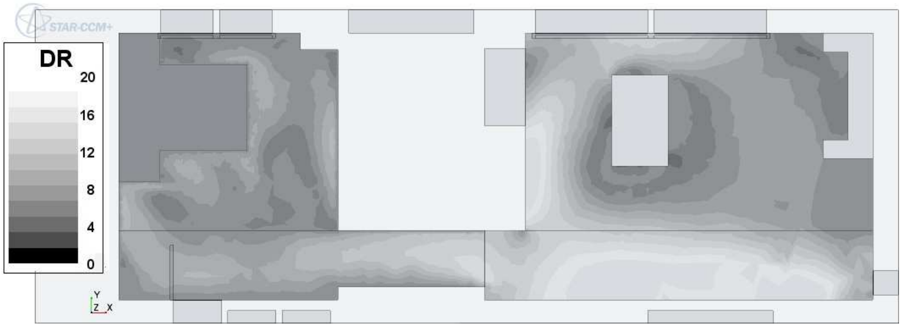


Figure 14. Draught Risk distribution at h = 0.6 m, T = 24.5 ºC.

In conclusion, NV achieves an acceptable *DR* distribution in the occupied zone for the prevailing wind during the warm season.

3.2.2. *PMV* analysis

Although *DR* is lower than 15% in the occupied zone, the building could not be classified as Category A [20] due to the other comfort indexes. Then, figure 15 presents the *PMV* distribution in the occupied zone (occupied volume in %) for each indoor air temperature and wind velocity tested. *PMV* index is kept between 0.2 and 1 (almost neutral, slightly warm) for all tested conditions. Moreover, the air velocity *PMV* effect is almost negligible if wind velocity is lower than 3 m/s. In summer conditions, for people feeling warm in their body, an air movement increase will decrease the warm discomfort and will therefore be beneficial. *PMV* index shows this phenomenon in the building. Although *PMV* is kept in a narrow range for all tested conditions, Figure 15 shows how *PMV* index is slightly improved for higher wind velocities (4.5 m/s). However, *PMV* is not as concentrated as for lower wind velocities (1.58 m/s and 3 m/s), in which 80% of the volume has the same *PMV* index (*PMV* = 0.5 for *T* = 24.5 ºC; *PMV* = 0.75 for *T* = 25.5 ºC and *PMV* = 0.88 for *T* = 26 ºC).

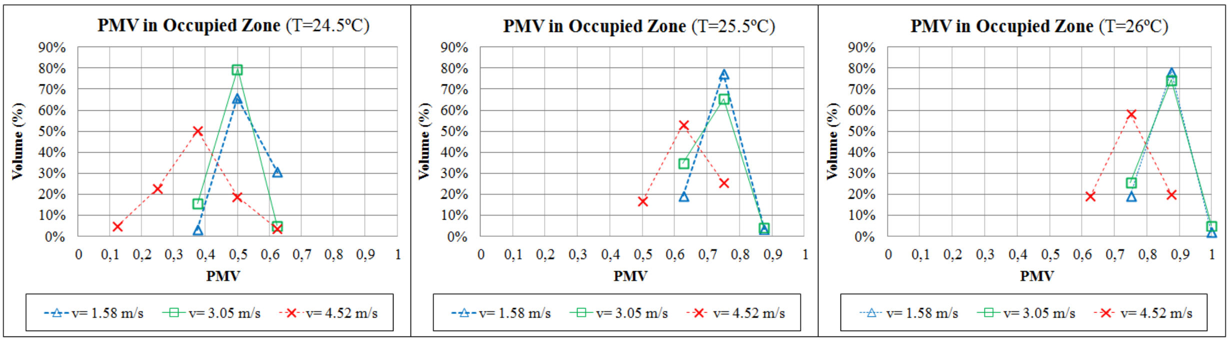


Figure 15. *PMV* index distribution in occupied zone.

According to *PMV* comfort index, the indoor environment cannot be classified as Category A as *PMV* is higher than 0.2 in the occupied zone. The indoor environment is classified as Category B for indoor air temperature lower than 25.5 ºC and higher temperatures means Category C [20].

3.2.3. PPD analysis

Figure 16 presents *PPD* distributions in the occupied volume (in %) for each indoor air temperature and each wind velocity. Similar to *PMV*, the indoor environment could neither be classified as Category A according *PPD* comfort index, since it is higher than 6% for almost the whole occupied zone. Nevertheless, it is classified as Category B for *T* = 24.5 ºC and Category C for higher temperatures. Figure 16 shows that *PPD* ranged between 6% and 17% in most of the occupied zone. Lower wind velocities have *PPD* index more concentrated than high wind velocities. Predicted dissatisfied people are more spread along the occupied volume for air velocities higher than 3 m/s. Figure 16 shows that people feel more comfortable with higher than with lower air velocities in summer conditions, as it is expected.

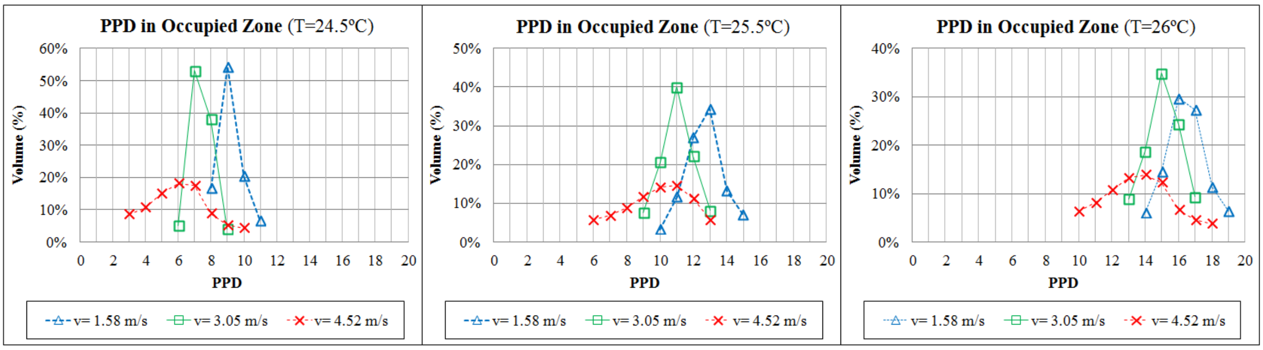


Figure 16. PPD index distribution in occupied zone.

PPD in the occupied zone for *T* = 24.5 ºC and lower air velocities (1.58 m/s) is similar than *PPD* in the occupied zone for *T* = 25.5 ºC and higher air velocities. *PPD* ranges between 8% and 12% is concentrated in 99% of the occupied zone for case *T* = 24.5 ºC and *v* = 1.58 m/s; *PPD* ranges between the same interval are concentrated in 61% of the occupied zone for case *T* = 25.5 ºC and *v* = 4.5 m/s. Similarly, *PPD* ranges between 11%

and 15% are concentrated in 93% of the occupied zone for case T = 25.5 ºC and v = 1.58 m/s; *PPD* ranges between the same interval are concentrated in 58% of the occupied zone for case T = 26 ºC and v = 4.5 m/s. This calculation demonstrates that *PPD* comfort index belonging to a lower indoor air temperature is ≈ 60% achieved with higher indoor air temperatures and higher air velocities. Therefore, the wind-driven NV effect improves the energy savings by increasing the target indoor temperature, maintaining 60% the predicted percentage of people likely to feel uncomfortable. In other words, comfort feeling associated with a lower indoor air temperature is achieved by increasing air velocity using a non mechanical system such as NV.

4. Conclusions

The paper presents the results of a numerical and experimental research carried out to analyse natural ventilation (NV) effect on indoor comfort conditions and assess the energy efficiency potential improvement. The research is done in case study building that had been previously designed to maximize its NV behaviour since the building design stage. Computational fluid dynamic (CFD) techniques are used to simulate the NV behaviour through the building and calculate comfort conditions indexes in the indoor environment. The energy efficiency strategy aims to slightly increase the indoor temperature without compromising the initial comfort conditions. The slightly increased temperature feeling should be compensated by increasing the air velocity by means of NV.

Firstly, experimental full-scale measurements are used to validate the CFD model. The CFD calculations show a good agreement with the full-scale measurements. The comparison shows a root-mean-square error lower than 7.5%. Secondly, the simulation is used to calculate and visualize comfort conditions indexes at different conditions in the warm season. Three air temperatures and three wind speeds are carefully chosen to represent a wide range of summer environmental typical conditions, according to recorded temperatures in the measurement conditions. Comfort conditions are assessed through the indexes draught risk (*DR*), predicted mean vote (*PMV*) and predicted percentage of dissatisfied people (*PPD*). The CFD results show no risk of draught in the occupied zone. *DR* is kept under Category A value (*DR* < 15%) [20] for all tested conditions. Nevertheless, local discomfort due to draught is higher in the non-occupied area. Depending on the indoor air temperature and *PMV* and *PPD* indexes, the indoor environment could be classified as Category B or C. In any case the indoor environment is classified as “slightly warm” (*PMV* between 0.5 and 1) and *PPD* is lower than Category C limit, 15%.

The conclusion is that the NV strategy could manage to increase the indoor air temperature 1 ºC maintaining no draught risk and 60% the percentage of people likely to feel uncomfortable. Thus, energy savings are achieved due to the indoor air temperature increase without compromising the initial comfort conditions. Otherwise additional mechanical system should be necessary to maintain the comfort conditions if the indoor air temperature is increased. The energy needs of the building are then reduced. Thereby, a proper NV focus during the initial design stage could improve the building energy efficiency without compromising the indoor comfort conditions.

The ultimate aim of the research is to add a reliable NV behaviour analysis by CFD techniques and achieve more environmentally friendly buildings. Further research should include the analysis of air fluctuation and thermal behaviour (buoyancy effect, thermal inertia, radiation, etc.) implications on comfort conditions. Different opening distributions and winter conditions should be also analysed in the future.

Nomenclature:

Magnitude	Unit	Description	Magnitude	Unit	Description
W	W	Active work	v_{RMS}	m/s	Average wind or air speed
Fcl		Clothing area factor	Tcl	ºC	Clothing surface temperature
hc	W/m^2·K	Convective heat transfer coefficient	µ	m^2/s	Eddy viscosity
ρ	kg/m^3	Fluid density	g	N	Gravitational force
z	m	Height	S_m	kg	Mass
$Ø_{i,obs}$	m/s	Mean measured value for i point	Tr	ºC	Mean radiant temperature
M	W/m^2	Metabolic energy production	N		Number of analysed points
F	N	Outer force	Pa	Pa	Partial water vapour pressure
α		Power law exponent	$Ø_i$	m/s	Predicted value for i point
z_r	m	Reference height	$U(z_r)$	m/s	Reference wind speed at height z_r
h_0	m	Roughness height	p	Pa	Static pressure
$\bar{\tau}$		Stress tensor	H_{max}	m	Tallest building height
Icl	clo	Thermal clothing insulation	L	W/m^2	Thermal load on the body
t	s	Time	Tu	%	Turbulence intensity
k	m^2/s^2	Turbulent kinetic energy	I		Unit tensor
D	m	Urban represented area width	Ta	ºC	Wind or air temperature
v	m/s	Wind or air velocity	U(z)	m/s	Wind speed at height z

Acknowledgments

The research was done within the frame of "EDIFICACIÓN ECO EFICIENTE, E3” (E3 eco efficient building design) research project at Universitat Politècnica de València.

The meteorological information has been provided by the Spanish State Meteorological Agency, Ministry of Agriculture and Fisheries, Food and Environment. Información elaborada utilizando, entre otras, la suministrada por la Agencia Estatal de Meteorología. Ministerio de Agricultura, Alimentación y Medio Ambiente

References

[1] Kudryashova A, Genkov A, Mo T, Robert K-H, Ny H. (2015) Certification schemes for sustainable buildings: assessment of BREEM, LEED and LBC from a strategic sustainable development perspective. Blekindge Institute of Technology, Karlskrona. Sweden.

[2] Hanan M. T. (2015) Natural ventilation as energy efficient solution for achieving low-energy houses in Dubai. Energy and buildings 99, 284-291.

[3] U.E. Environmental Protection Agensty (EPA). Green building basic information. Retrieved Oct. 14th, 2014. Available from: http://www.epa.gov/greenbuilding/pubs/about.htm.

[4] Zhiwen L, Jianing Z, Jun G, Lixia H. (2007). Estimating natural-ventilation potential considering both thermal comfort and IAQ issues. Building and Environment 42(6), 2289-98.

[5] Hitchin E.R, Wilson C.B. (1967) A review of experimental techniques for the investigation of natural ventilation in buildings. Building Science 2, 59-82.

[6] Chen Q. (2009) Ventilation performance prediction for buildings: A method overview and recent applications. Building and Environment 44, 848–858.

[7] Coakley D, Raftery P, Keane M. (2014) A review of methods to match building energy simulation models to measured data. Renewable and sustainable energy reviews 37, 123-141.

[8] Zhu Y, Luo M, Ouyang Q, Huang L, Cao B. (2015) Dynamic characteristics and comfort assessment of airflows in indoor environments: A review. Building and Environment 91, 5-14.

[9] Etheridge D. (2015) A perspective on fifty years of natural ventilation research. Building and Environment 91, 51-60.

[10] Wang H, Lin H, C.Y. Ng V, Yu Guan L. (2015) Failure of natural ventilation strategy in a sustainable house in China. International Journal of Low-Carbon Technologies 10, 216-228.

[11] Moosavi L, Mahyuddin N, Abghafar N, Ismail M.A. (2014) Thermal performance of atria: An overview of natural ventilation effective designs. Renewable and Sustainable Energy Reviews 34, 654–670.

[12] Foucquier A, Robert S, Suard F, Stéphan L, Jay A. (2013) State of the art in building modelling and energy performances prediction: A review. Renewable and Sustainable Energy Reviews 23, 271–288.

[13] Mora-Pérez M, Guillén-Guillamón I, López-Jiménez P.A. (2015) Computational analysis of wind interactions for comparing different buildings sites in terms of natural ventilation. Advances in Engineering Software 88, 73-82.

[14] Yimin S, Wenyu Z, Chunyang Z. (2014) Preliminary Study on Natural Ventilation for Hospital Building in Hot and Humid Regions. 30th International Plea Conference. CEPT University, Ahmedabad.

[15] Zhou C; Wang Z, Chen Q, Jiang Y, Pei J. (2014) Design optimization and field demonstration of natural ventilation for high-rise residential buildings Energy & Buildings 82, 457-465.

[16] Van Hooff T, Blocken B, Aanen L, Bronsema B. (2011) A venturi-shaped roof for wind-induced natural ventilation of buildings: wind tunnel and CFD evaluation of different design configurations. Build Environ 46, 1797-807.

[17] Liu P-C, Lin H-T, Chou J-H. (2009) Evaluation of buoyancy-driven ventilation in atrium buildings using computational fluid dynamics and reduced-scale air model. Build Environ 44, 1970–9.

[18] Richard Hughes B, Kaiser Calautit J, Abdul Ghani, S. (2012) The development of commercial wind towers for natural ventilation: A review. Applied Energy 92, 606-627.

[19] Mora-Pérez M, Guillén-Guillamón I, López-Patiño G, López-Jiménez P.A. (2016) Natural ventilation building design approach in Mediterranean regions – A case study at the Valencian coastal regional scale (Spain). Sustainability 8(9), 855.

[20] ISO Standard 7730 (2005) Ergonomics of the thermal environment – Analytical determination and interpretation of thermal comfort using calculation of the PMV and PPD indices and local thermal comfort criteria.

[21] Comité técnico AEN/CTN 100. UNE-CR 1752:2008. Ventilation for Buildings—Design Criteria for the Indoor Environment; Aenor, Madrid, Spain, 2008. Available from http://www.aenor.es.

[22] ANSI/ASHRAE Standard 55-2013—Thermal environmental conditions for human occupancy.

[23] Gagliano A, Nocera F, Patania F, Moschella A, Detommaso M, Evola G. (2016) Synergic effects of termal mass and natural ventilation on the termal behaviour of traditional massive buildings. International Journal of Sustainable Energy 35, Issue 5.

[24] CD-Adapco Star CCM+ User’s Manual. Available from: www.cd-adapco.com.

[25] Franke J, Hellsten A, Schlünzen H, Carissimo B. (2007) Best practice guideline for the CFD simulation of flows in the urban environment.

[26] Wieringa J. (1992) Updating the Davenport roughness classification. Journal of Wind Engineering and Industrial Aerodynamics 41, 357–68.

[27] Agencia Estatal de Meteorología de España (AEMET). This meteorological information has been provided by the Spanish State Meteorological Agency, Ministry of Agriculture and Fisheries, Food and Environment. It is forbidden its total or partial reproduction by any other means. Available from: http://www.aemet.es (accessed on 9th July 2016).

[28] Windfinder. Available from: http://es.windfinder.com/windstats/windstatistic_valencia.htm (accessed on 15th April 2015).

[29] Manwell J.F, Mcgowan J.G, Rogers A.L. Wind energy explained: theory, design and application. 2nd ed. Wiley. ISBN 978-0-470-01500-1.

[30] Peterson E, Hennessey J. (1977) On the use of power laws for estimates of wind power potential. Journal of applied meteorology 17.

[31] Justus CG, Mikhail Amir. (1976) Height variation of wind speed and wind distributions statistics. Geophysical. Research Letters 3, 261–4.

[32] Ray SD, Gong N-W, Glicksman LR, Paradiso JA. (2014) Experimental characterization of full-scale naturally ventilated atrium and validation of CFD simulations. Energy and Buildings 69, 285–91.

[33] Shao J, Liu J, Zhao J. (2012) Evaluation of various non-linear k-epsilon models for predicting wind flow around an isolated high-rise building within the surface boundary layer. Building and Environment 57, 145–155.

[34] García JAO, (2010) A review of general and local thermal comfort models for controlling indoor ambiences, In: kumar, A. Author, Air Quality, InTech, 309-326.

[35] Fanger P.O. (1972) Thermal comfort—Analysis and applications in environmental engineering, Kingsport Press, Inc.

[36] Reglamento de Instalaciones Térmicas en los Edificios (RITE). Ministerio de Industria, Energía y Turismo, Gobierno de España. Available from: http://www.minetur.gob.es/energia/desarrollo/EficienciaEnergetica/RITE/Paginas/InstalacionesTermicas.aspx (accessed on 15th April 2015).

[37] EN 13779:2007. Ventilation for non-residential buildings—Performance requirements for ventilation and room-conditioning systems. April 2007.

[38] Hajdukievwicz M, Geron M. Keane M.M. (2013) Formal calibration methodology for CFD models of naturally ventilated indoor environments. Building and Environment 59, 290–302.

[39] Van Hooff T, Blocken B. (2010) On the effect of wind direction and urban surroundings on natural ventilation of a large semi-enclosed stadium. Comput. Fluids 39, 1146–1155.

[40] James Lo L, Novoselac A. (2012) Cross ventilation with small openings: Measurements in a multi-zone test building. Build. Environ 57, 377–386.

[41] Franke J, Hirsch C, Jensen A.G, Krus H.W, Schatzmann M, Westbury P.S, Miles S.D, Wisse J.A, Wright N.G. (2004) Recommendations on the use of CFD in wind engineering. In: van Beeck, J.P.A.J. (Ed.), Proceedings of the International Conference Urban Wind Engineering and Building Aerodynamics, von Karman Institute.