CFD MODELLING OF WIND VENTILATION INSIDE BUILDINGS

Ronaldo Horácio Cumplido Neto[1], ronaldo.neto@ufv.br
Álvaro Messias Bigonha Tibiriçá[1], alvaro.tibirica@ufv.br
Marcelo Pereira Coelho[2], marcelo.coelho@ufv.br
Júlio Cesar Costa Campos[1], julio.campos@ufv.br
[1]Departamento de Engenharia de Produção e Mecânica
[2]Departamento de Engenharia Agrícola
Universidade Federal de Viçosa
Av. P. H. Rolfs, s/n – Campus Universitário – Viçosa (MG) – Brazil

Abstract. *The use of Computational Fluid Dynamics (CFD) for determination of airflow inside buildings has been subject of several researches. CFD analyses for airflow are based on the numerical solution of Linear Momentum and Mass Conservation differential partial equations. These equations are solved numerically for a set of coupled elementary finite volumes (Finite Volume Method) obtained from domains discretisation. This work uses two domains to model wind ventilation inside buildings: an external domain that models atmospheric flow and an internal domain that models turbulent airflow inside building. When more than one domain is used, the interface between domains and external domain size are important parameters for results' quality. The objective here is to investigate how these parameters influence simulations' results. Results demonstrated that width ratio, height ratio and length ratio between the two domains influence simulations results. Interface mesh refinement improved flow smoothness. However, the error between CFD model and analytical airflow mass model increased.*

Keywords*: CFD, Ventilation, Wind*

1. INTRODUCTION

Natural ventilation uses the natural forces of wind pressure to drive air through buildings. Pressure differences inside and outside are responsible for airflow inside buildings (Khan et al., 2008). Indoor air is replaced by fresh air without use of mechanical work (Jiang et al., 2003). There is no power consumption, which contributes to energy efficiency in buildings and makes natural ventilation use attractive (Yin et al., 2010, Jiang et al., 2003).

Natural ventilation evaluation methods in buildings are divided in: field measurement, controlled experiments and numerical simulations. The first evaluates ventilation using local measurements. The second is based on controlled experiments, generally, using wind tunnels. Both methods are limited by instruments accuracy. The third works with solution of mathematical models. These models describe physical phenomena involved in natural ventilation. Among the three approaches, numerical simulation for natural ventilation prediction in buildings has better cost-benefit (Wang and Wong, 2009).

CFD (Computer Fluid Dynamics) techniques for predicting airflow caused by wind in buildings have been subject of several authors' works (Kim and Park, 2010, Wang and Wong, 2009; Visagavel and Srinivasan, 2009; Asfour and Gadi, 2008, Jiang et al. 2003; Ayad, 1999). A CFD model is based on concept of dividing the solution domain into sub-domains (zone) – coupled elementary volumes (Finite Volume Method). For each zone, mass, linear momentum and energy conservation partial differential equations are solved numerically. Comparisons between experiments using wind tunnels and CFD simulations have shown a good agreement (Asfour and Gadi, 2007).

Average airflow rate calculation inside buildings can be estimated through analytical method. It is a validation parameter for CFD simulation. Equation 1 relates airflow rate (Q_n) to wind velocity (V).

$$Q_n = \rho A_{eff} V \frac{(C_{pn} - C_{pi})}{\sqrt{|C_{pn} - C_{pi}|}} \quad (1)$$

ρ is air density, A_{eff} is opening effective area, C_{pn} is pressure coefficient at opening n and C_{pi} is pressure coefficient inside space.

This paper reproduced wind-driven CFD simulations described in Asfour and Gadi (2007) for normal wind direction. External domain dimensions and interface between domains refinement were investigated. Objective is to verify influence of external domain dimensions and interface mesh refinement between domains on simulations' results. Section 2 describes the model and methodology used for running CFD simulations. Section 3 presents simulations' most relevant results. Section 4 concludes the paper.

2. MODEL DESCRIPTIONS AND METHODOLOGY

2.1. Model configuration

Three buildings models were modeled. These models were proposed by Asfour and Gadi (2007): "cases have nearly the same volume (125 m^3 and 128 m^3), but different aspect ratios (1:1, 1:2 and 2:1)". Figure 1 shows models geometry and Table 1 summarizes their dimensions. Openings area is 4 m^2 (2 m x 2 m). They are centered on facade and normal to the wind direction.

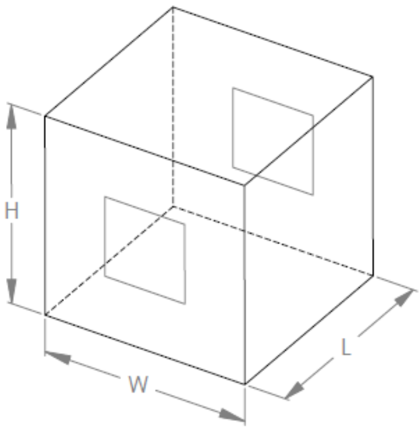


Figure 1 – Buildings model geometry with W (width), L (length) and H (height) dimensions.

Table 1 – Width, length and height of three geometries modeled.

	Width (W)	Length (L)	Height (H)
Geometry 1	5,0 m	5,0 m	5,0 m
Geometry 2	4,0 m	8,0 m	4,0 m
Geometry 3	8,0 m	4,0 m	4,0 m

To model airflow around buildings (atmospheric flow), an external box was modeled. The buildings were positioned inside the box (see

Figure 2). Simulations used dimensions proposed by Asfour and Gadi (2007): 30 m x 30 m x 20 m for the three buildings geometries. Visagavel and Srinivasan (2009) proposed the use of external boxes dependent on the building geometry dimensions for two dimensional CFD simulations. After preliminary trial runs, they found dimensions with sufficient sizes to model external airflow. The same methodology was adopted in this paper. Initially, Asfour and Gadi (2007) external box dimensions were adopted. The dimensions were increased until the airflow around buildings was fully developed – velocities at external box boundaries had almost returned to the values they would have if airflow had not been disturbed by building. Figure 3 shows external airflow using Asfour and Gadi (2007) dimensions and using dimensions for fully developed external airflow. Table 2 summarizes the relationship between external box and building sizes for fully development external airflow.

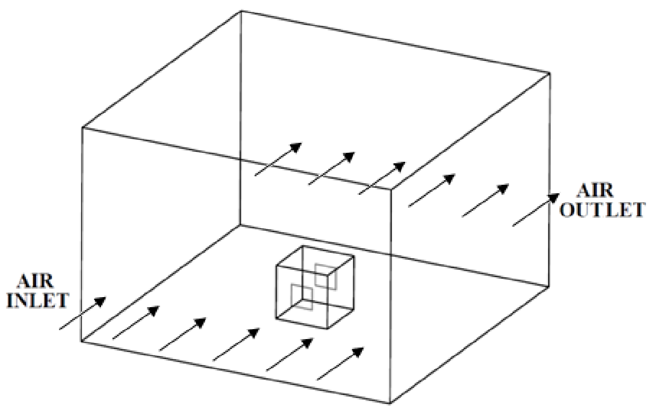


Figure 2 – External box and building model

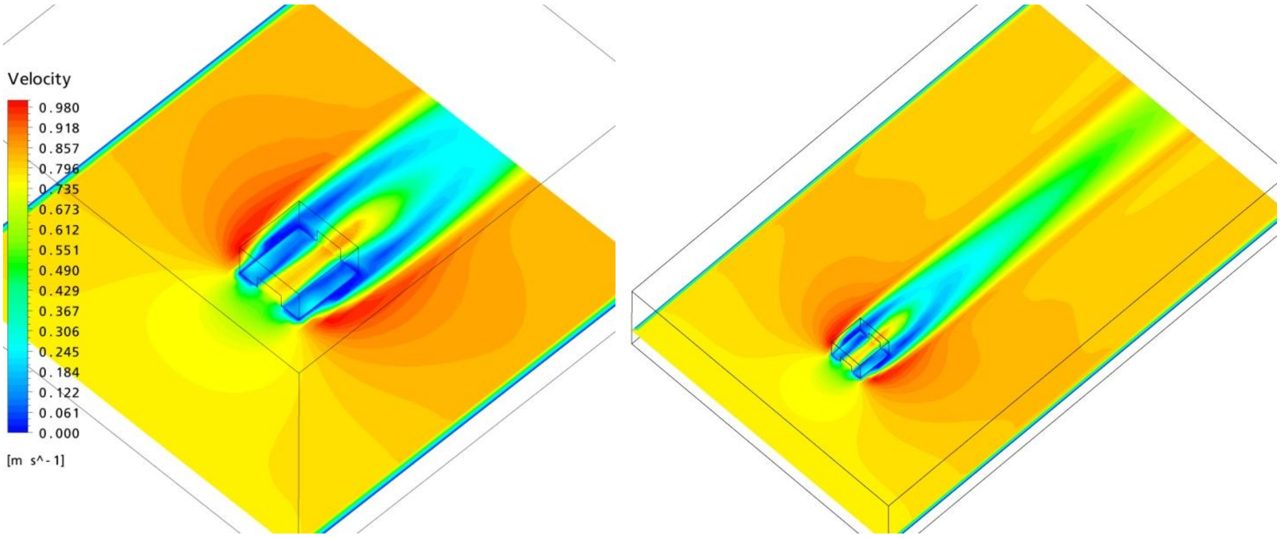


Figure 3 – Airflow around building (geometry 1): (a) Asfour and Gadi (2007) external box dimensions; (b) fully developed airflow external box dimensions.

Table 2 – Relationship between external box and buildings sizes (W, L and H) for fully development airflow.

	Width (W)	Length (L)	Height (H)
External Domain 1	4W	14L	2H
External Domain 2	5W	11L	2H
External Domain 3	4W	19L	2H

2.2. Numerical method

To solve CFD models, Finite Volume Method (FVM) was applied. FVM divides the solution into small cells where the governing conservation laws are solved. The models simulated in this paper solve Mass and Momentum Conservation Laws and use k-ε turbulence model. Airflow was modeled as incompressible and isothermal. Convergence criteria of 10^{-4} were chosen for all equations. Ansys CFX 11, one the most widely used commercial software, was used for CFD simulation.

2.3. Boundary conditions

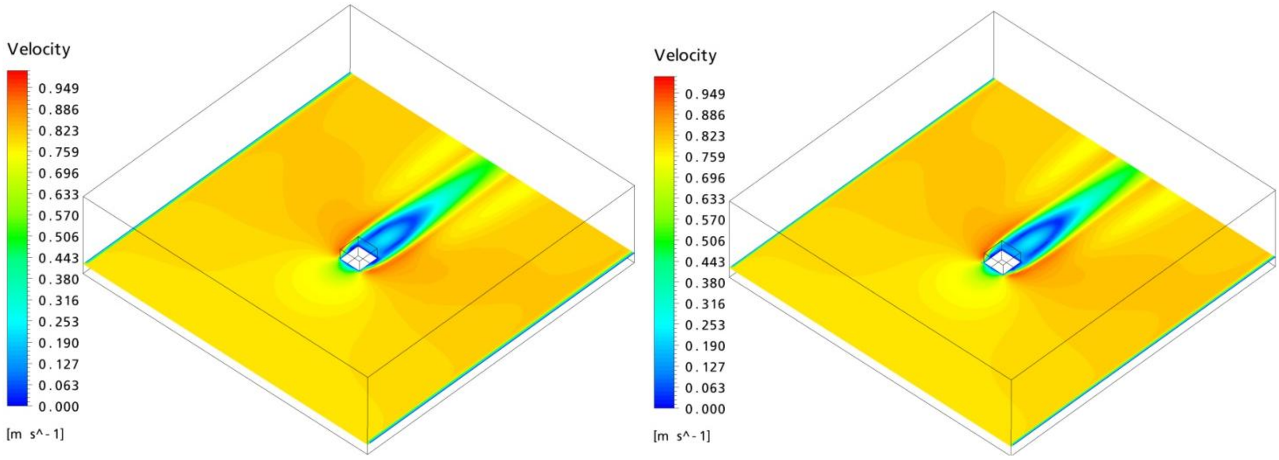
A parabolic profile was used to model wind velocity variation due to height increase (Eftekhari *et al.*, 2003; Asfour and Gadi, 2007; Montazeri and Azizian, 2008; Wang and Wong, 2009). Equation 2 describes wind profile.

$$\mathrm{V} = \mathrm{V_r a H^b} \tag{2}$$

V is wind velocity, V_r is reference wind velocity at 10 m (meteorological data), *H* is the height where wind velocity is estimated, *a* is a parameter related to wind velocity and terrain nature and *b* is an exponent related to wind velocity and height to the ground. This paper uses the same data described in Asfour and Gadi (2007): *a* = 0.68, *b* = 0.17 (both for open country terrain) and reference velocity (V_r) = 1.0 m/s. Atmospheric pressure (1 atm – 101.325 kPa) and temperature of 25 ^{o}C were used as reference values.

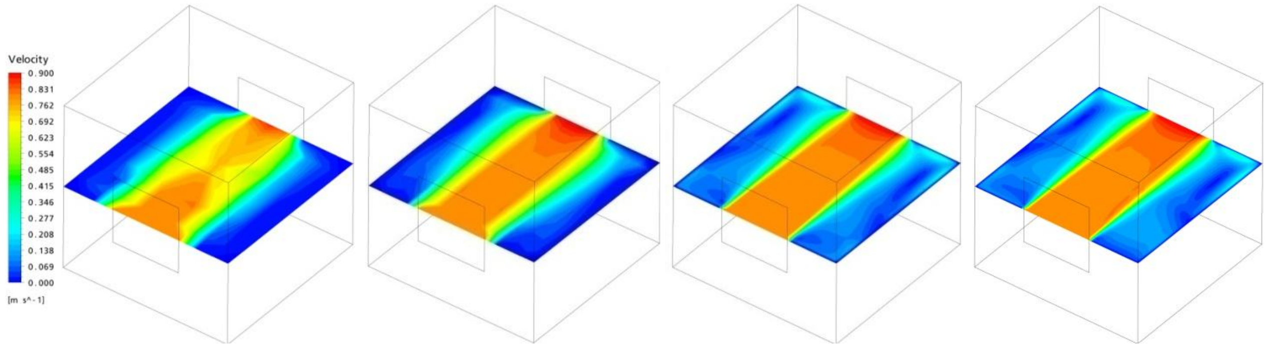
2.4. Computational grid

Two computational grids were employed: one for atmospheric flow (external domain) and another for airflow inside building (internal domain). Both were hexadominant structured meshes. Mesh refinement tests were applied for both domains. No significant differences were found between 0.8 m and 0.6 m spacing for external domain (see Figure 4). The last one was the spacing used by Asfour and Gadi (2007). Internal domain refinement test converged for 0.10 m spacing mesh (see Figure 5). Simulations for 0.8 m and 0.6 m mesh spacing were not time consuming. Therefore 0.6 m spacing was chosen for external domain. Mesh refinement between internal and external domains were done. Figure 6 shows the interface between domains refinement. Table 3 summarizes internal and external domains' mesh characteristics.



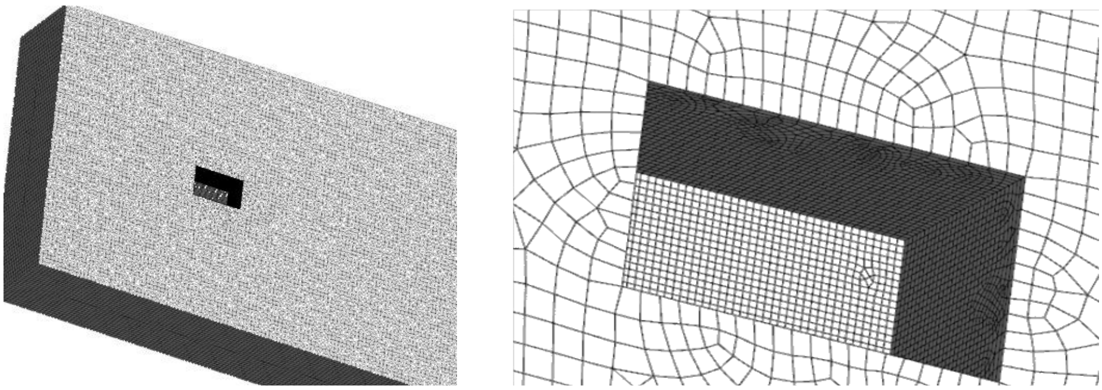
(a) (b)

Figure 4 – Velocity contours for external domain mesh spacing (a) 0.80 m and (b) 0.60 m.



(a) (b) (c) (d)

Figure 5 - Velocity contours for internal domain mesh spacing (a) 0,60 m; (b) 0,40 m; (c) 0,10 m; (d) 0,08m.



(a) (b)

Figure 6 – External domain (a) overview and (b) interface refiniment.

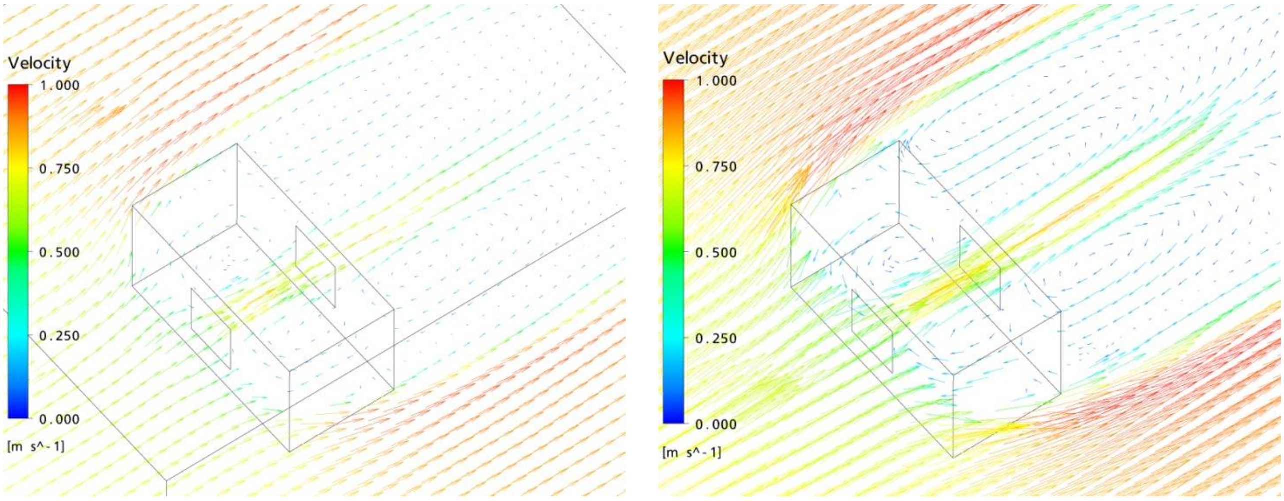
Table 3 – Internal and external domain meshes characteristics.

	Mesh Type	Elements Size	Number of Elements
Internal Mesh 1	Hexadominant	0,10 m	125 000
Internal Mesh 2	Hexadominant	0,10 m	128 000
Internal Mesh 3	Hexadominant	0,10 m	128 960
External Mesh 1	Hexadominant	0,60 m	153 340
External Mesh 2	Hexadominant	0,60 m	153 318
External Mesh 3	Hexadominant	0,60 m	398 371

3. RESULTS

3.1. External domain dimensions

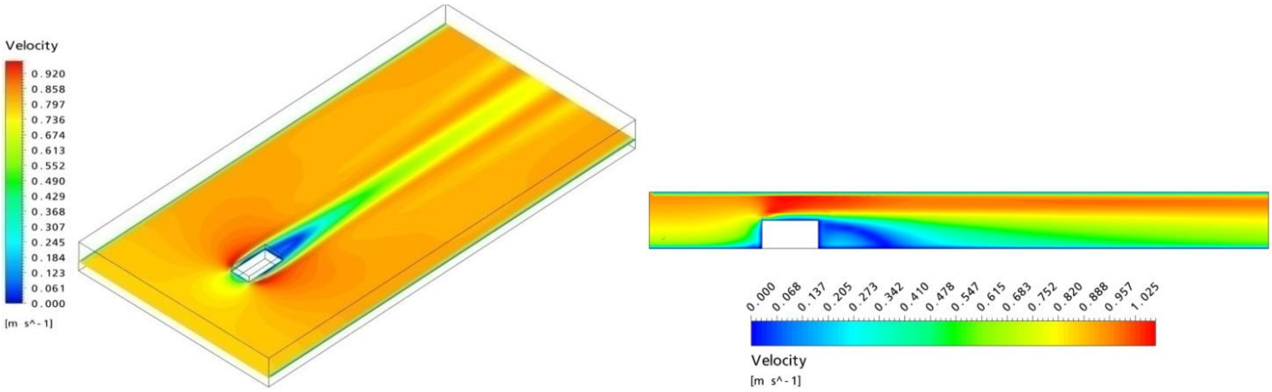
Figure 7 shows airflow patterns for two different sizes of external domain. The first (Figure 7a) was a 30 m x 30 m x 20 m box. These dimensions were used by Asfour and Gadi (2007) for external domains of the three geometries simulated. The second (Figure 7b) was a 32 m x 76 m x 8 m box proportional to building dimensions (8 m x 4 m x 4 m). Theses dimensions ensure fully developed external airflow.



(a) (b)

Figure 7 – Geometry 3 external airflow pattern for (a) 30 m x 30 m x 20 m external domain and for (b) building-proportional external domain (32 m x 76 m x 8 m).

Figure 8 shows external airflow velocity contours for geometry 2. The size of external domain was 20 m x 88 m x 8 m. Airflow around building was fully developed. Table 4 summarizes mass flow through building using fully airflow developed external domain and 30 m x 30 m x 20 m external domains used by Asfour and Gadi (2007) for the three geometries simulated. Percentage difference was also calculated for each geometry.



(a) (b)

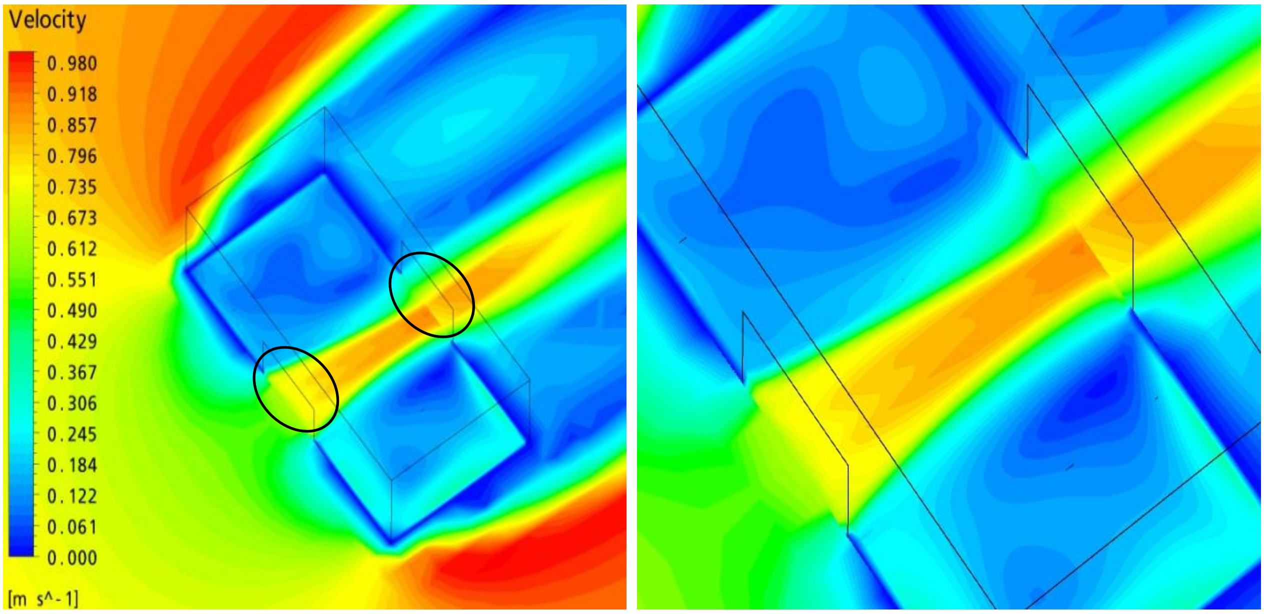
Figure 8 – Geometry 2 velocity contours for fully developed external airflow: (a) horizontal plane and (b) longitudinal plane.

Table 4 – Mass flow computation for fully developed airflow external domain and 30 m x 30 m x 20 m external domain for the three simulated geometries.

	Fully developed airflow external domain	30 m x 30 m x 20 m external domain	Difference (%)
Geometry 1	3, 067	3, 156	2,820 %
Geometry 2	2, 807	2, 847	1, 405 %
Geometry 3	2, 854	2, 647	-7, 820%

3.2. Internal and external domains interface

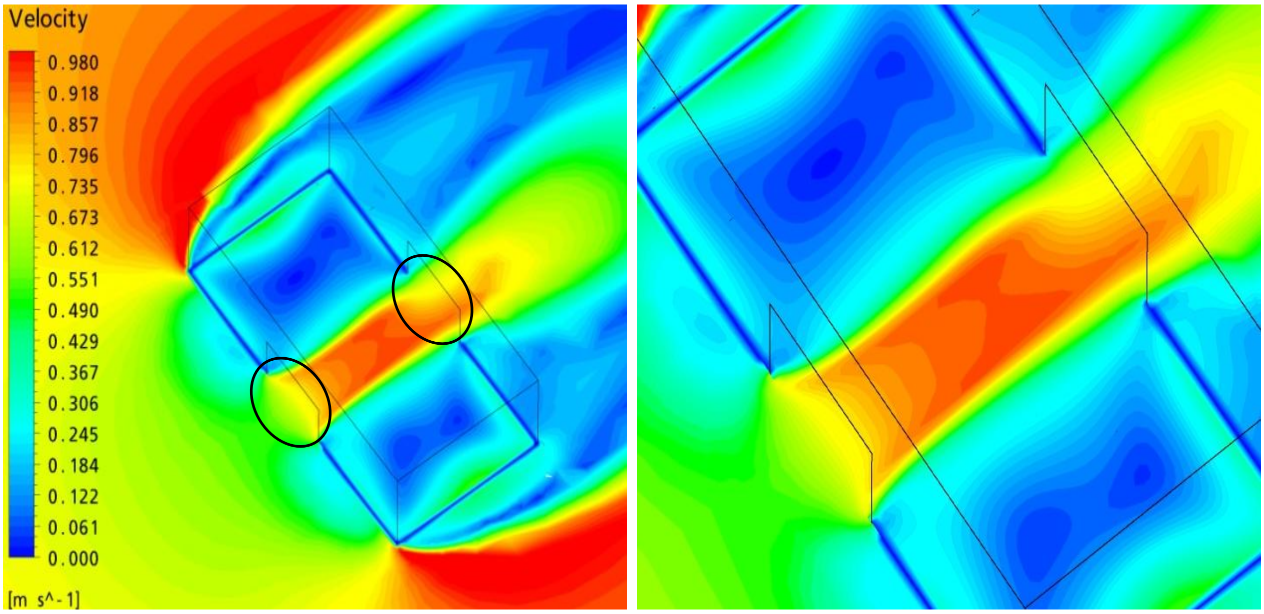
Figure 9 shows geometry 3 velocity contour. No special interface was built for this CFD model. Discontinuity was observed between external and internal flow. However, flow general characteristics were preserved.



(a) (b)

Figure 9 – Geometry 3 velocity contour for simulation without special interface between external and internal domains: (a) discontinuities were observed at openings; (b) openings magnification.

Figure 10 shows geometry 3 velocity contour. A special interface was built for this CFD model to preserve continuity between the external and the internal domain. However, the interface mesh generated for this case caused flow asymmetry.

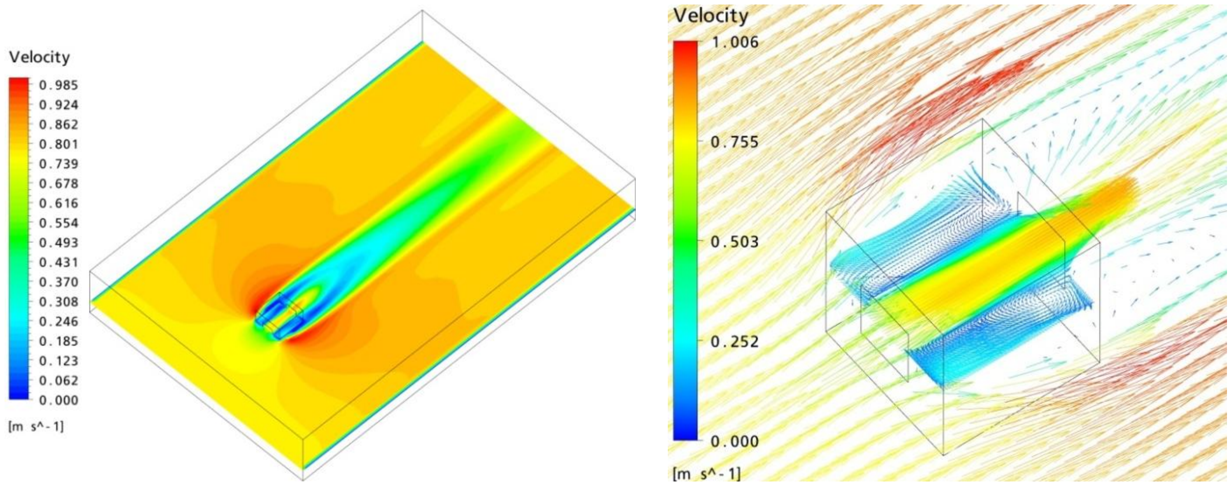


(a) (b)

Figure 10 - Geometry 3 velocity contour for simulation with special interface between external and internal domains: (a) asymmetry was observed at exit opening; (b) openings magnification.

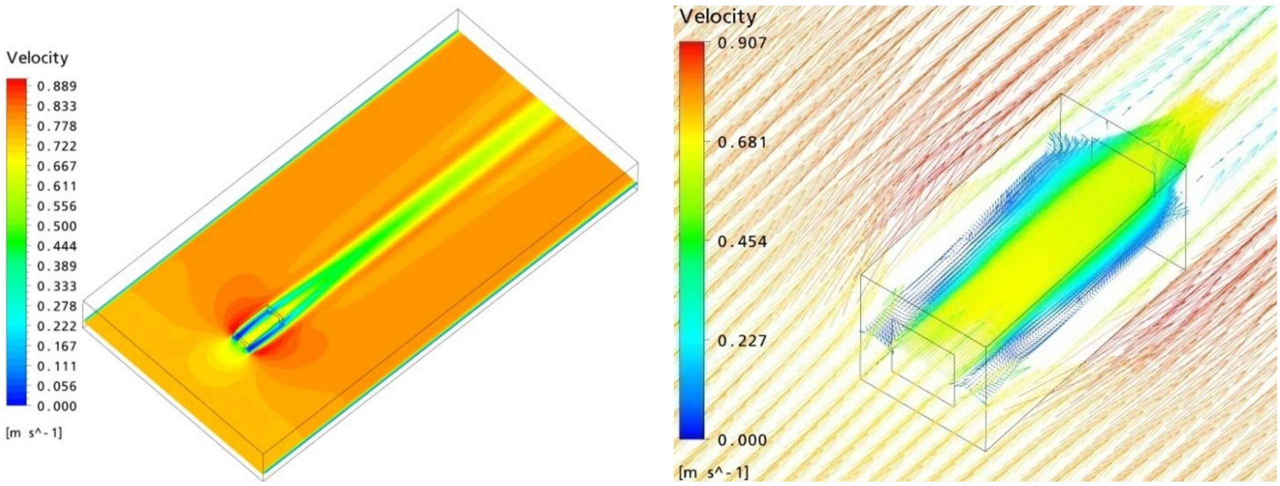
3.3. CFD simulations results

Figure 11, Figure 12 and Figure 13 show CFD simulation results for geometries 1, 2 and 3, respectively. All results were obtained using external domain size proportional to building dimensions. External airflow was considered fully developed for the three geometries.



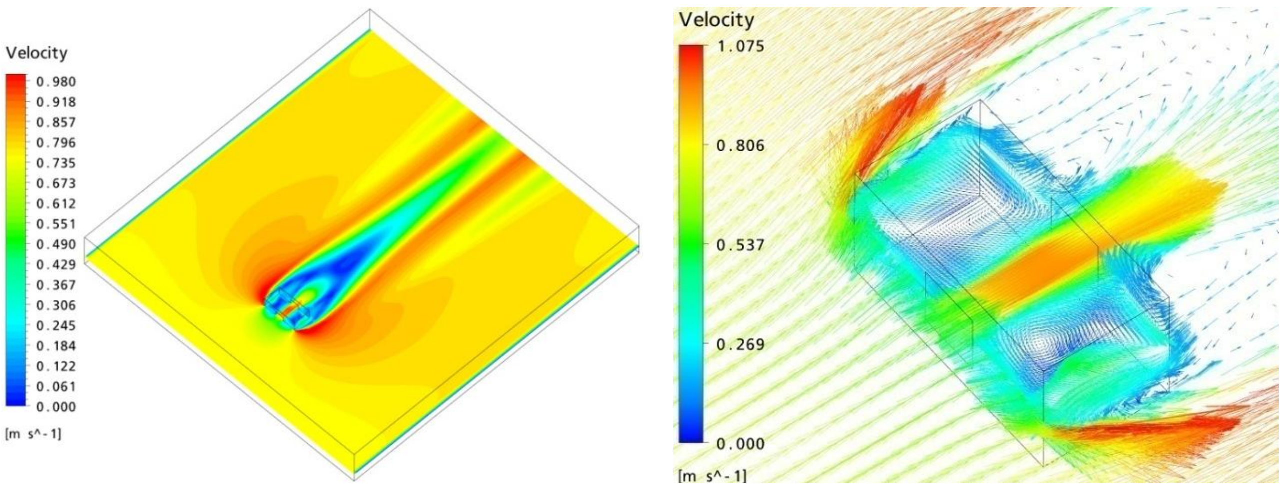
(a) (b)

Figure 11 – Geometry 1 simulation results: (a) velocity contours; (b) velocity vectors.



(a) (b)

Figure 12 - Geometry 2 simulation results: (a) velocity contours; (b) velocity vectors.



(a) (b)

Figure 13 - Geometry 3 simulation results: (a) velocity contours; (b) velocity vectors.

3.4. CFD simulations and analytical model average mass flow comparison

Table 5 summarizes average mass airflow inside buildings calculated using analytical model and CFD simulations for geometries 1, 2 and 3. CFD simulations used fully developed external airflow. Inlet and outlet mass flow were the same. This result obeys Mass Conservation Law for control volumes. Errors were calculated using analytical model as reference.

Table 5 – Average mass airflow calculated using analytical model and CFD model.

	Analytical Mass Flow (kg/s)	CFD Mass Flow – Inlet (kg/s)	CFD Mass Flow – Outlet (kg/s)	Inlet Error (%)	Outlet Error (%)
Geometry 1	2, 844	3, 067	3, 067	7,841%	7,841%
Geometry 2	2, 808	2, 807	2, 807	-0,036%	-0,036%
Geometry 3	3, 152	2, 854	2, 854	-9,454%	-9,454%

4. CONCLUSION

This paper reproduced the models geometry used by Asfour and Gadi (2007) for normal wind-driven ventilation CFD simulation. Three geometries with two openings centralized in opposite facades were simulated. The wind was normal to the opening facade. External domain dimensions were increased from size described in Asfour and Gadi (2007) to size which airflow around the buildings were fully developed. External domain dimension increase influenced airflow behavior around and into the buildings modeled. Average mass flow difference from -7.82% to 2.82% was observed between the two sizes used for external domains. Qualitative differences of velocity profile were also observed between fully developed air flow and not fully developed air flow. An interface mesh was used between external and internal domains. The interface had improved the smoothness of flow between external and internal domains. However, the error between CFD model with interface between domains and analytical model increased.

5. ACKNOWLEDGEMENTS

Authors thank to FAPEMIG, FUNARBE, CAPES and CNPq for financial support to the research group.

6. REFERENCES

Asfour, O. S.; Gadi, M. B. (2007). “A Comparison between CFD and Network models for predicting wind-driven ventilation in buildings”. Building and Environment, v. 42, p. 4097-4085.

Asfour, O. S.; Gadi, M. B. (2008) “Using CFD to investigate ventilation characteristics of vaults as wind-inducing devices in buildings”. Applied Energy, v. 85, p. 1126-1140.

Ayad, S. S. (1999). “Computational study of natural ventilation”. Wind Engineering and Industrial Aerodynamics”, v. 82, p. 49-68.

Eftekhari, M.M.; Marjanovic, L. D.; Pinnock, D.J. (2003). “Air flow distribution in and around a single-sided naturally ventilated room”. Building and Environment, v. 38, p. 389-397.

Jiang, Vi; Alexander, D.; Jenkins, H.; Arthur R.; Chen Q. (2003). “Natural ventilation in buildings: measurement in a wind tunnel and numerical simulation with large-eddy simulation”. Wind Engineering and Industrial Aerodynamics, v. 91, p. 331-353.

Khan, N.; Su, Y.; Riffat, S. B. (2008) “A review on wind driven ventilation techniques”. Energy and Buildings, v. 40, p. 1586-1604.

Kim, T. J.; Park, J. S. (2010) “Natural ventilation with traditional korean opening in contemporary house”. Building and Environment, v. 45, p. 51-57.

Montazeri, H.; Azizian, R. (2008). “Experimental study on natural ventilation performance of one-sided wind catcher”. Building and Environment, v. 43, p. 2193-2202.

Visagavel, K.; Srinivasan, P. S. S. (2009). “Analysis of single side ventilated and cross ventilated rooms by varying the width of the window opening using CFD”. Solar Energy, v. 83, p. 2-5.

Wang, L.; Wong, N. H. (2009). “Coupled simulations for naturally ventilated rooms between Building Simulation (BS) and Computational Fluid Dynamics (CFD) for better prediction of indoor thermal environment”. Building and Environment, v. 44, p. 95-112.

Yin, W.; Zhang, G.; Yang, W.; Wang, X. (2010) “Natural ventilation potential model considering solution multiplicity, window opening percentage, air velocity and humidity in china”. Building and Environment, v. 45, p. 338-344.

7. RESPONSIBILITY NOTICE

The following text, properly adapted to the number of authors, must be included in the last section of the paper:
The author(s) is (are) the only responsible for the printed material included in this paper.