

"NATURAL CONVECTION PROPOSAL TO THE THEMATIC POLO IN RENEWABLE ENERGY AND ENVIRONMENT BUILDING AT THE SÃO PAULO STATE UNIVERSITY"

Priscila Maria Penalva Partel

Escola de Engenharia de São Carlos da Universidade de São Paulo
Av. Trabalhador Sancarlene, 400 São Carlos, SP
partel@sc.usp.br

Paulo Selegim júnior

Escola de Engenharia de São Carlos da Universidade de São Paulo
Av. Trabalhador Sancarlene, 400 São Carlos, SP
selegim@sc.usp.br

Abstract. *This paper presents a proposals for energy optimization and rationalization in the building design for the Thematic Polo in Renewable Energy and Environment at the University of São Paulo in São Carlos, to be built at Campus II. This work aims better energy efficient and sustainability during phases of construction, use and operation of the building. Since the design of the building makes a contemporary architecture interact with renewable energies technologies aimed more energy efficiency The design of the building was simulated with ANSYS software Computational Fluid Dynamics _CFX for the determination of internal flow and heat fluxes to obtain thermal comfort with minimal energy expenditure to drive air conditioning, ventilation system, etc. The natural convection induces motion in public areas of the building cooling flow through the internal heat flows between the bottom of the ramp on the ground floor and the hollow elements in mortar, in the top closure (curved beams between the cover and sealing walls). The gaps between panels cover the walls and closing the building permit cooling by natural convection, and lighting. The results obtained are greater sustainability, making the building more energy efficient.*

Keywords: *natural convection, green architecture, energy optimization.*

1. INTRODUCTION

There are public sector initiatives in construction, to encourage, standardize and disseminate sustainable technologies, mainly focused on optimization and rationalization of energy in buildings. Examples of initiatives on the national scene are: Solar Law in São Paulo (Law No. 14,459, July 3, 2007) which made compulsory the use of solar water heaters in new buildings residential, commercial and industrial. The Brazilian Association of Refrigeration, Air-Conditioning, Ventilation and Heating (ABRAVA) Brazilian Association of Engineering Building Systems (ABRASIP) and the Brazilian Association for Compliance and Efficiency Facility (ABRISTAL) released on March 18, 2009 one of the Parameter Guide as well as software_ Dimensol as references for the design of new installations solar heating for buildings of São Paulo and as bases to support the Municipality in the approval and supervision of the works of the law.

Daily demand for hot water, its temperature and the location of the property on which the equipment will be installed to determine the available solar radiation, and ambient temperature (historical averages); area available for the installation of the collectors, the conditions for architectural sizing and system integration including equipment participants Brazilian Labeling Program (PBE) of Inmetro¹.

These products contain label attesting to their quality and efficiency, indicating the average monthly production of energy (kWh/month or kWh/mês.m²). In this sense, the Minha Casa, Minha Vida 2 with the goal of building a million homes for low-income Brazilian families, provides for the use of solar thermal energy to replace electric showers. Besides these, other environmentally efficient solutions such as water recycling, collection systems and wastewater treatment and use of wood from certified, are provided. The use of solar energy, even if only for replacing electric showers in the housing program, can reduce by 520 megawatts (MW) energy demand / year and avoid the emission of 830 tons of greenhouse gases.

It is observed that in the design of buildings for the XXI century must seek greater compatibility between the resources of the local climate and the project also generating thus resulting reductions in energy consumption and CO² emissions. Accordingly, to meet the growing needs parameter during the design of architecture and urbanism, design projective stands inseparably from recent computer technologies, enabling constant evaluations from the architectural design, to the building in use. The theme Optimization and Rationalization in Energy Project buildings are placed in the context of interdisciplinary research aimed at energy towards equalizing technology, development and environment in the construction industry. This work is part of a set of actions that cover environmental education, practice and research aimed at ENERGY in the city of São Carlos, SP. According SELEGHIM (2002) 2 the Core Thermal Power and Fluids (NETeF) has the fundamental mission of enabling the association of human and material resources from different departments of the institution, in order to conduct projects in nature strongly multidisciplinary.

The contribution of the study theme of this work is analysis for optimization and rationalization of energy in the design for the Polo theme in Renewable Energy and Environment USP (Polo Ground) São Carlos proposed to Campus II of the University of São Paulo.



Figure 01 electronic model building Polo ground.

It was intended to analyze a project still in the development phase, in order to improve the energy efficiency of architectural elements present.

¹<http://www.inmetro.gov.br/consumer/tabelas.asp>.

²SELEGHIM (2002) *Establishment of a line of research in instrumentation and industrial control multiphase flows. Text systematization of career Associate Professor. Engineering Department Mecânica.EESC / USP. 2002.130p*

With this purpose, this study used the Ansys CFX. Its use is divided into three parts: Pre-processing: when are established characteristics of the fluid to be simulated (if will be modeled perfect gas, call viscous), the boundary conditions (physical limits, input and output flow) and the variables input (speed and temperature of the fluid and its boundaries, among others); Processing using numerical methods, the software finds approximations to the solutions of the equations of the model generated from the conditions imposed. The criterion for the end of the calculations is set by the user, is the accuracy of the approximations, their absolute values or numbers of iterations. In this work, we used precision 0.0001 with maximum of 100 iterations, unless highlighted other criteria; Post-Processing: All solutions are available for the model, leaving the user to choose what to analyze and how to do it. Here, preference was given to the magnitude and direction of the relative values of speed and pressure profiles and variation in the environment.

The software programs were used: Autocad, Solid Works and Ansys for a first evaluation of the air flow from the outside of the building. Other similar software and same principles could have been adopted, including yourself ANSYS with multiple extensions, applications and byproducts for the same need, these options were sized according to the analysis required and available equipment in the process. Topics were adopted according to the steps of the software simulations in finite elements: pre-processing, analysis and post-processing (Evaluation). In preprocessing was chosen geometry (longitudinal) to better analyze performance against the internal flow and lower the computational difficulty among the options of Plans, Elevations and Cortes, shown in Figure 02:



Figure 02. Plans, Elevations and Sections.

After defining the geometry figure passes by subtracting elements irrelevant to the simulation software.

In the upper position of Figure 03 presents the longitudinal section selected:

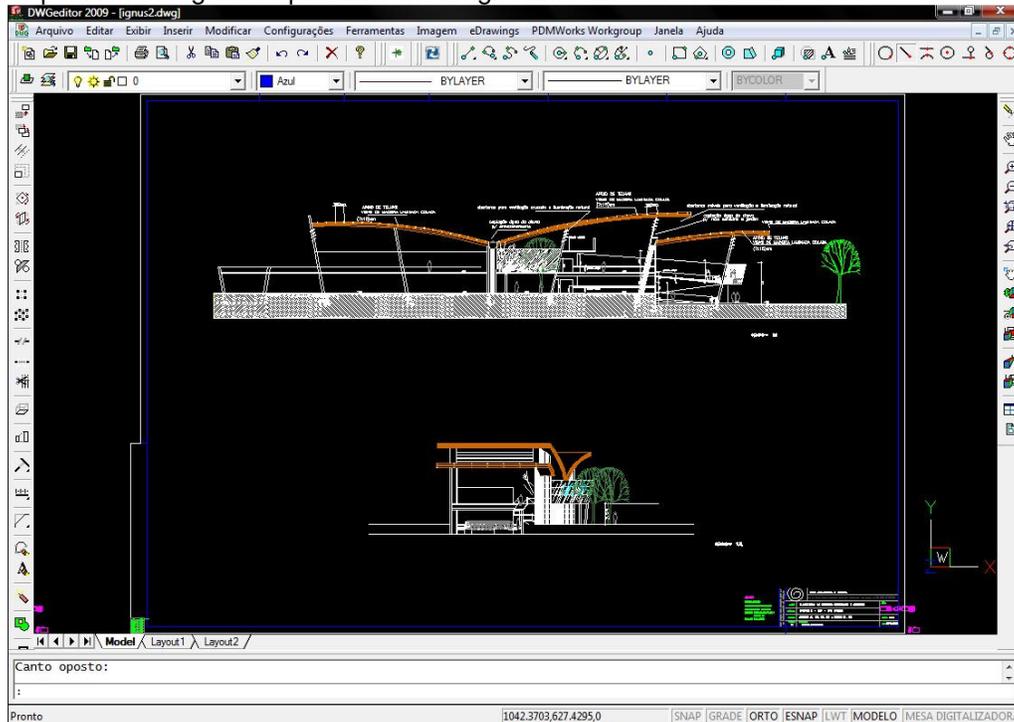


Figure 03 - Longitudinal section selected.

The geometry observed in Figure 03, was suppressed in all details, predicting the future of the computational steps (Figure 04).

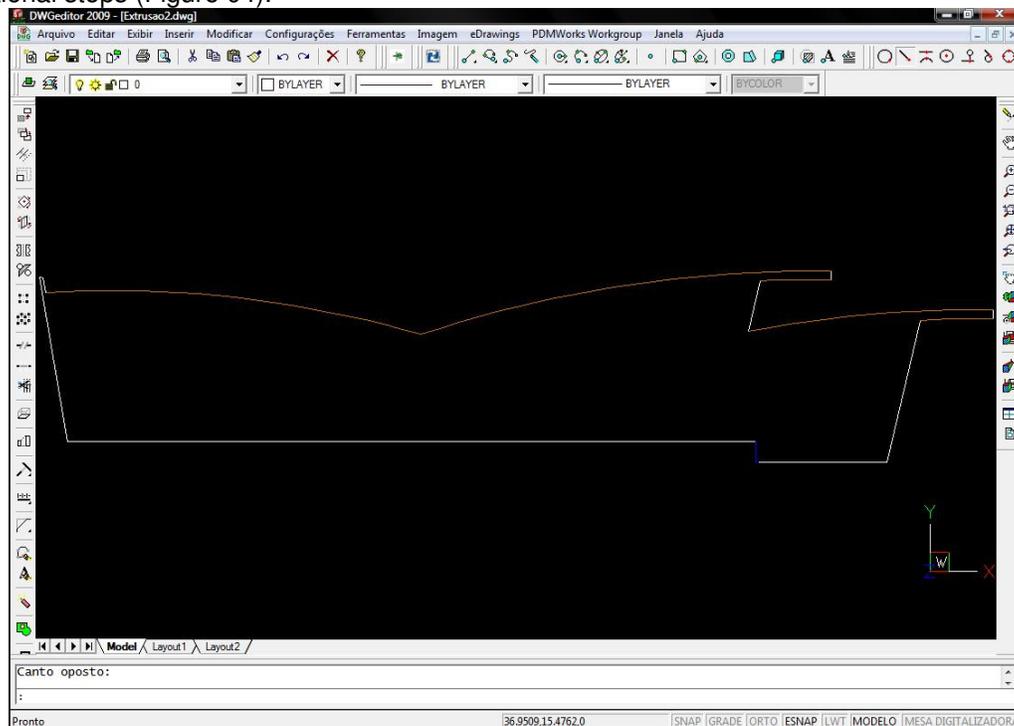


Figure 04 Simplified and clean geometry and for the specific case.

Finally, the final geometry is saved in the file format for the next software, Solid Works that uses the same extension (. Dwg) of Autocad.

This software is currently the most used by architectural firms in Brazil, which enables the interface to the CFD and can contribute to a greater use of this type of simulation.

Following on the import was done treatment of geometry in Solid Works (SW) for the removal of sharp angles (Fig. 05 smoothing of sharp angles) and determination of the volume control of the simulation (VC) shown in Figure 06 _ wrapped box of geometry.

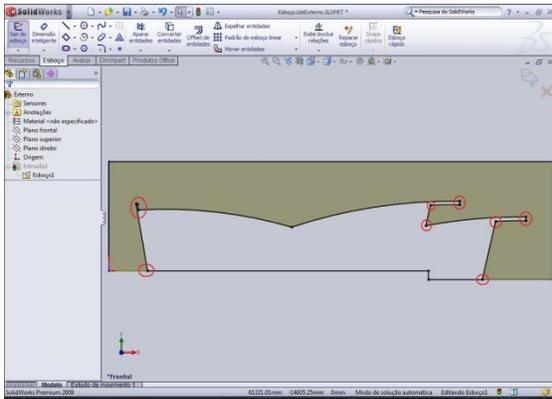


Figure. 05 Smoothing of sharp angles

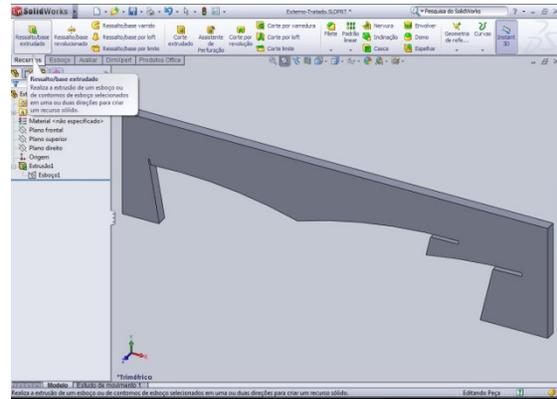


Figure. 06 volume control simulation (VC)

In the creation of the solid in SW is also an existing property in all CADs to work with 3D mechanical design, called extrusion or protrusion. This command creates a solid from 2D drawings closed (hence the existence of the VC), Fig.06. At the end of this stage of geometry, it was necessary to save the solid obtained in a useful extension to the mesh generator in extension "Parasolid", classified as universal reading.

Mesh was used to generate extension ANSYS Workbench. This extension is more interactive to be more illustrative than other forms of ANSYS (eg Multifics). When you open the software, your home screen Figure 08:

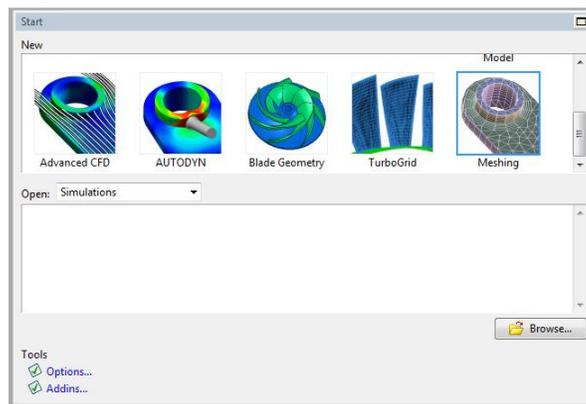


Figure 07 - Initial screen of Workbench

The second step was to choose from the possibilities of working software, the mesh generator (Meshing). Block sorting options mesh, selected: CFD, CFX-Mesh Meshing and Set Default. In the first generation, it was interesting to leave the default properties of the software, for better distribution between available volume and efficiency calculations. Confirmed the previous assessments, cared for the simplified geometry. The software automatically opened the option to refine the mesh, the mesh generation properties.

On the toolbar tab "[CFX-Mesh]" to trigger the command called "Verify Geometry" geometry was scanned for possible errors, inconsistencies computational and notices of pieces of geometry.

Subsequently presented in a block of message, information about the analysis, Figure 08:

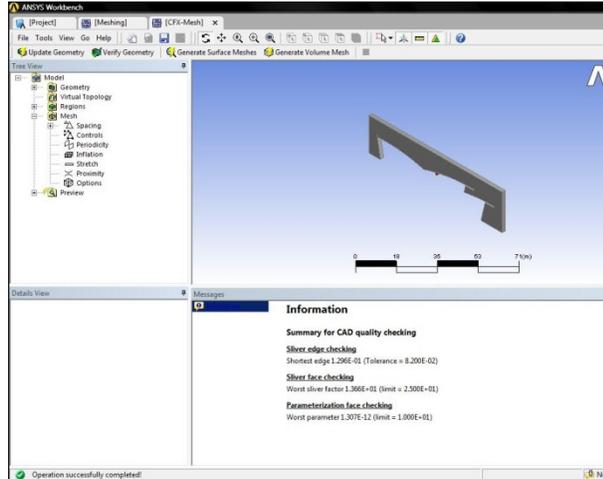


Figure 08 - Results of geometry verification

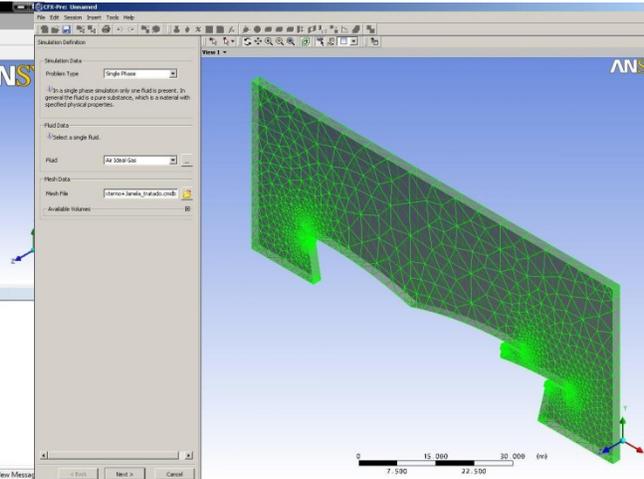


Figure 09 - Mesh and mailbox

Completed checks and geometry changes necessary until the checker did not present more errors, broke into the mesh generation. When finished meshing the "Generate Volume Mesh" provided the number of elements and nodes contained in the mesh, shown in Figure 10. In detail in the geometry, the elements (triangles) were smaller, it is called local mesh refinement. The software creates the mesh default mode, but often it is necessary to rework this fabric and make it more refined, with smaller triangles, requiring more time computational process. At the option of the fluids, the ideal gas model was enough to drain the modeling. The mesh was chosen for analysis, physical properties, patterns of steady-state reference pressure, turbulence and heat transfer satisfactory for this simulation. Specify the types of boundary conditions (inputs, outputs, walls) speeds and temperatures. The fluid properties and boundary conditions were established in ANSYS CFX-Pre.

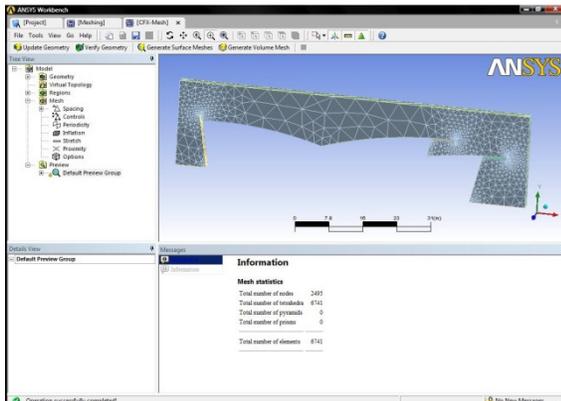


Figure 10-mesh creation in default mode.

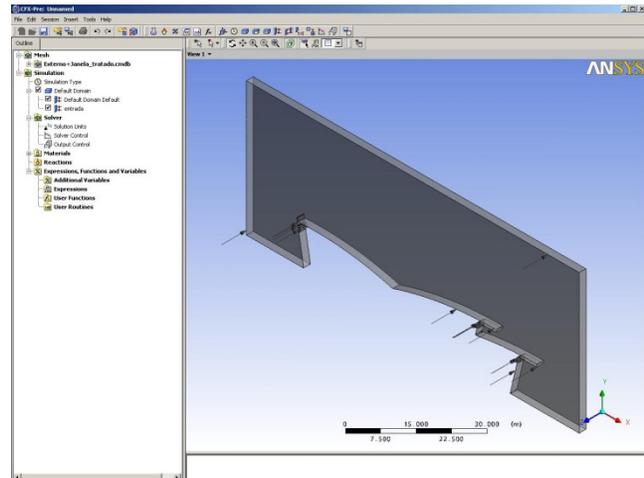


Figure 11 - preprocessing

For purposes more direct, it was interesting to create a new simulation using the "Quick Setup" that allows you to establish other standards. Because of these patterns, some values could not be admitted as negative velocities. In this simulation they were changed after the first simulation set, still in ANSYS CFX-Pre. All options of preprocessing were again accessed. At this stage the inputs are not allowed in the Quick Setup as negative velocities could be applied.

With all the correct conditions, the archive solution can be prepared, taking into account that the stop criteria for the solution of the analysis were chosen according to the required accuracy for the results. The simulation was saved for changing its boundary conditions, so that there was the need to repeat the previous procedures in each calculation. Each new experiment was proposed by analyzing and interpreting the results of previous, always trying conditions as close as possible to reality.

This work carried out three basic types of simulation, regarding the environments considered: Internal and External, External and only Internal only. The input data were based on the prevailing winds in the region of San Carlos, which blow from the northeast (30.2% in February and March), with an average speed of 3.5 m/s. Because the building south façade, the volume of control used here does not get winds perpendicular, but with Cartesian components of 2.47 m / s in X and Y. The coordinate system used was shown in all representations of velocity profiles and pressure. The first simulation considered only air inlets on the right cheek of the control volume and only the velocity component V_x . Symmetry conditions were established on the side of the volume, indicating that the walls of the building has infinite width. This was necessary to observe the velocity profile in the plane of interest, the only dimension that is supposed to flow.

As a result, we obtained a high pressure region at the entrance control volume in front of the first wall of the building and a direction of flow along the ceiling. The first results from the symmetry condition: the air has nowhere to drain after reaching the wall (unlike reality, when it could seep laterally toward Z), leaving only greatly reduce your speed, the second is due to the upper volume control does not allow the output stream. The air then is redirected through the influence of a surface that does not actually exist.

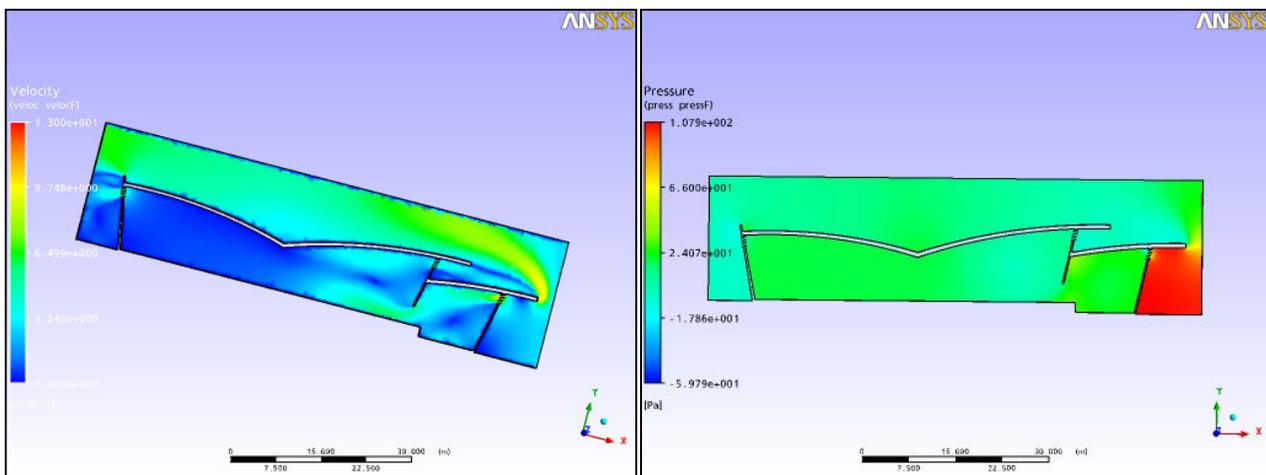


Figure 13 internal and external environment simulations

Figure 14 internal and external environment simulations

To compensate for this distortion, consider the following simulations the upper control volume and horizontal surface immediately in front of the entrance and exit regions. This allowed the air initially stagnated on the building wall flows down (compensating symmetry condition) and it was deflected onto the roof of the building retain its component Y.

Simulations with Internal and External temperature intended to illustrate the effects of ventilation to the temperature difference. First, we considered only the effects of the interaction between flow at constant temperature and its boundary conditions. The result showed the occurrence of a variation profile, but with absolute values not countable (less than 0.1%).

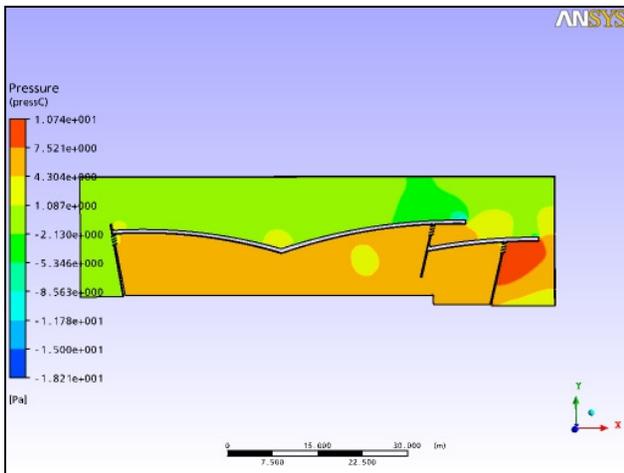


Figure 13 internal and external pressure environment simulations

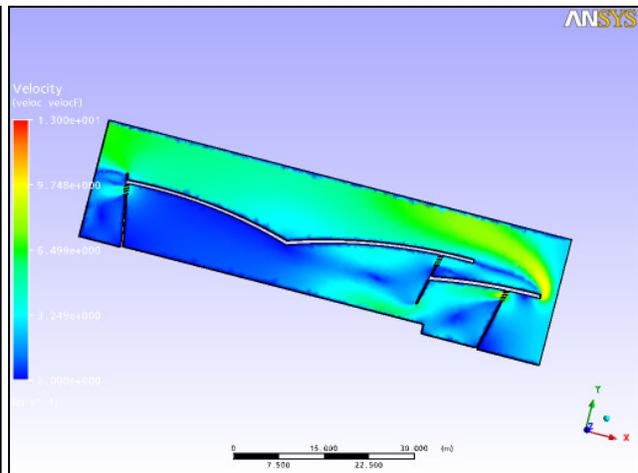


Figure 14 internal and external velocity environment simulations

By applying a fixed temperature in the building floor and ceiling outside both above that of the flow, it can be noted vary widely, near 7%. However, as one of the windows has a restricted air flow is between two covers approaching the temperatures in the environment that were closer to the ground than the flow. Seeking more accurate data environments (external) were separated:

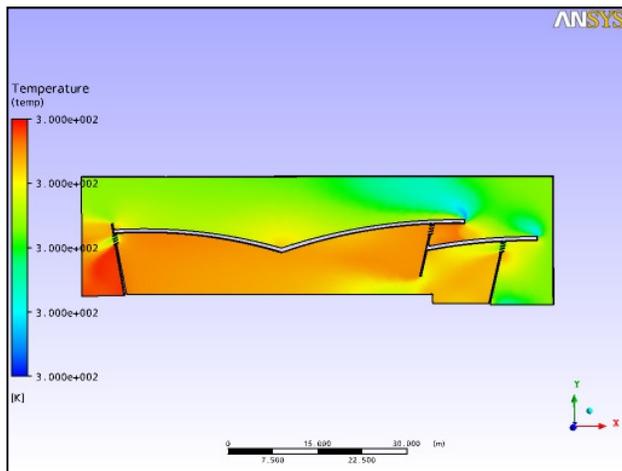


Figure 15 simulations internal and external environment

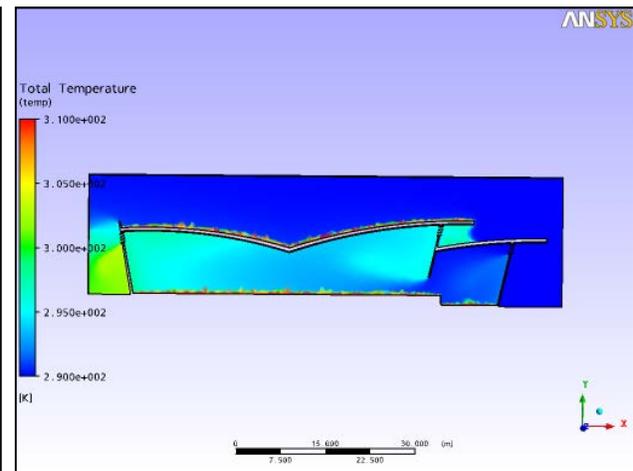


Figure 16 simulations internal and external environment

The first external simulations, as well as the previous input Dimensional considered the only component of V_x input speed and the right surface of the control volume. The outputs were left above and the right bottom surface, the front wall of the building and the windows.

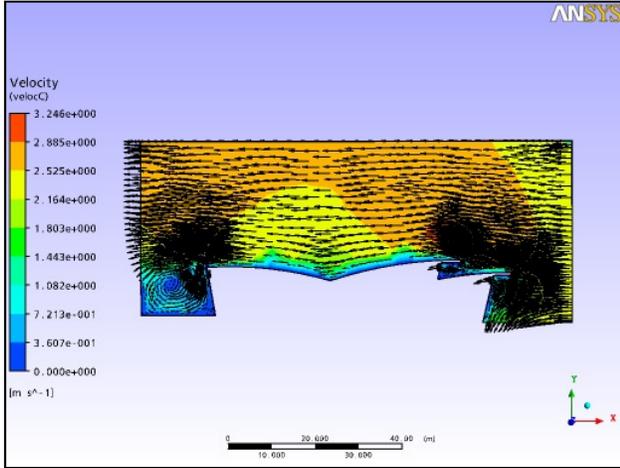


Figure 17 speed external environment simulation

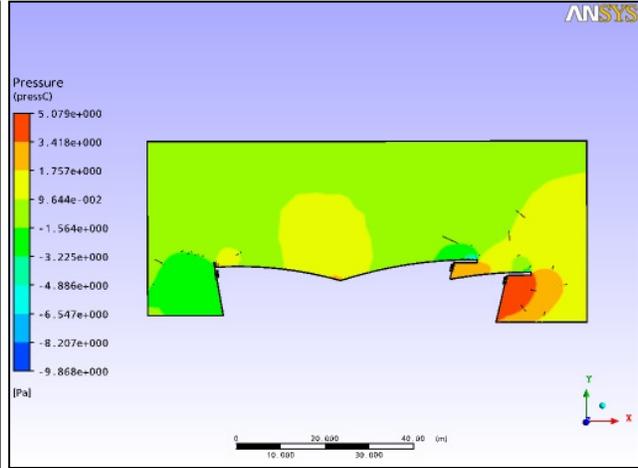


Figure 18 pressure external environment simulation

RESULTS

The results allowed us to detail the behavior of the flow. It may be noted that the mesh was not able to differentiate between the two walls of the flow over windows between them are represented only walls. Thus, the flow rate was zero in the output windows. This result indicates, in principle, there is no flow to the interior of the building known fact incorrect, either by intuition, as based on previous simulations. To correct this effect, the sidewalls of the windows (and only them) have been considered without friction.

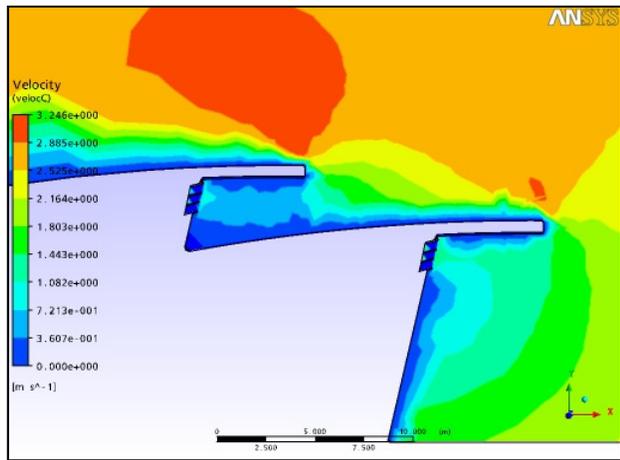


Figure 19 Two-dimensional simulations entries

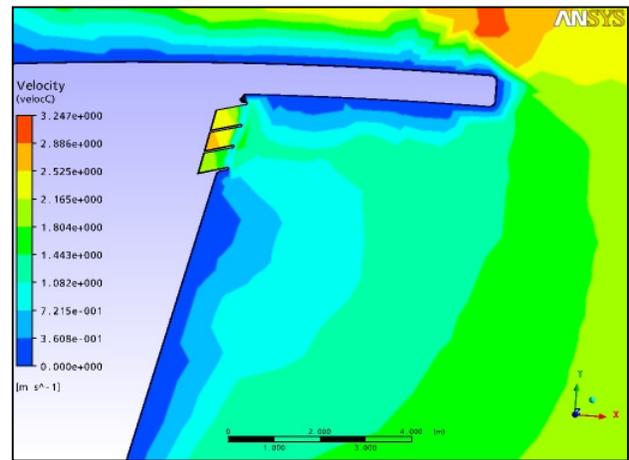


Figure 20 Two-dimensional simulations entries detail

As the inside of the building was not being represented, can adopt new input surfaces. With that, both as component V_x V_y Wind could be used, leaving more real values. Was added, then the entry and exit major faces of the control volume and the wind having passed the two components (Two-Dimensional Input).

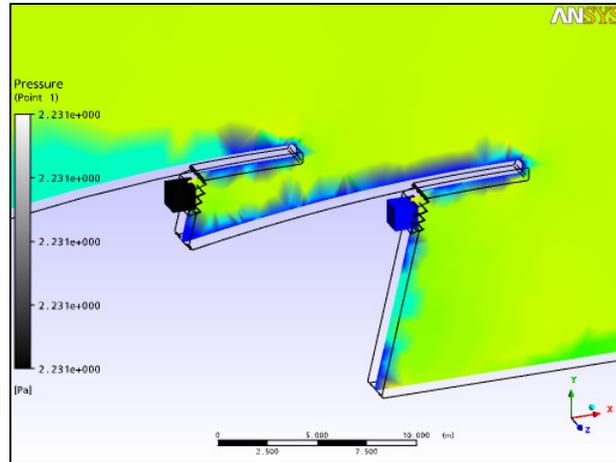


Figure 21 Pressure.

Having actual speeds, the values were measured in the region of the two input window, resulting in components X, Y and Z each. These values were far more accurate analysis parameters used as the inner part. Procedure: To observe the flow as a whole within the building, the volume control portion has become thicker. We considered two windows in the input and output right-left window. The values for the input speeds were those obtained in simulation for each window outside.

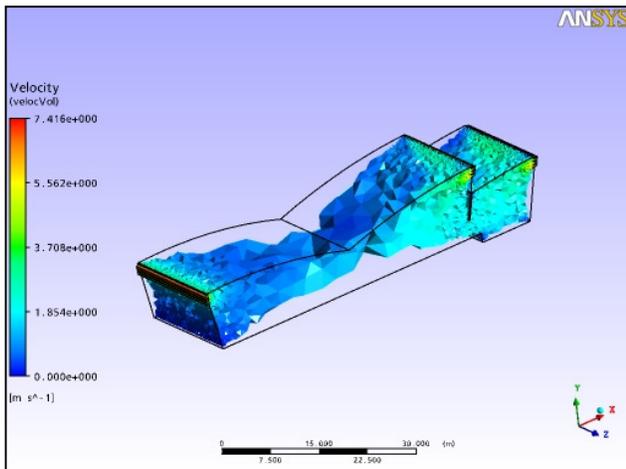


Figure 22. indoor environment simulations

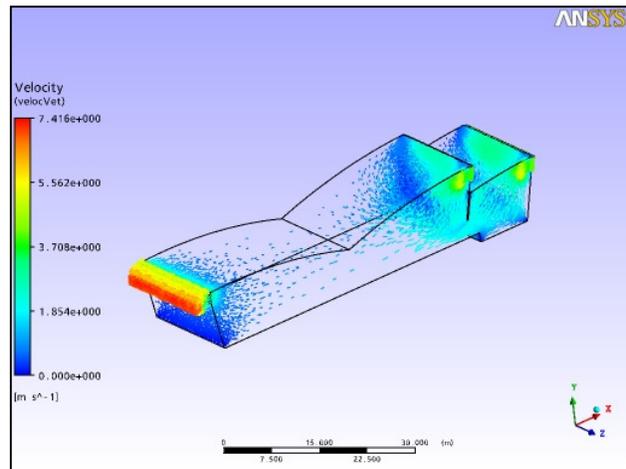


Figure 23. indoor environment simulations



Results show any air flow in all dimensions. Research stage illustrated some of the possibilities of applying CFD analysis of the influence of architectural form seeking to obtain an efficient natural ventilation, with the object still in a building project.

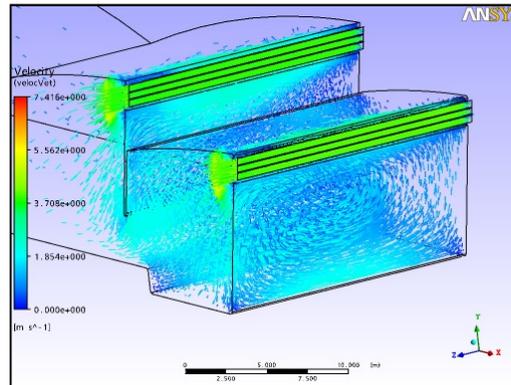


Figure. 44. Final analysis.

The simulations presented natural ventilation, assess mainly aerodynamic aspects, representing the potential for vertical ventilation through roof appropriate form, coupled with vents located in strategic areas. With the temperature distributions and other analyzes throughout the environment resulting from Ansys, changes may be proposed more accurate in the project, from the identification of areas in critical condition. The analyzes carried out throughout the development phase of the project contributed to greater energy efficiency of architectural elements present. The results represent greater sustainability related to the use of energy making the building design more rational in the use of this feature.

The CFX software is rarely used as a tool in the area of projective architecture, although one of its main characteristics is the interaction with CAD systems. Accordingly, such a tool will certainly be the subject of future studies, expanding its use and creating new methods to obtain better results, able to create a more close to reality, helping architectural solutions to be more energy efficient.

In continuation of the research and recommendations for future simulations will be working three-dimensional model on a reduced scale. Will gradually proposal most possible simplification of internal divisions (bulkheads) for these simplifications of geometry compromise as little as possible to the reality proposed project during the simulations.

3. ACKNOWLEDGEMENTS

Coordination Support for Higher Education Program National Post Doctorate_CAPES Students who contributed to the development of this work.

4. REFERENCES

Brazilian Association of Refrigeration, Air-Conditioning, Ventilation and Heating (ABRAVA) <http://www.abrava.com.br/?Canal=44&OperId=2096&Channel=UmVsZWFzZQ==> access on 23/05/2013
 Brazilian Association of Engineering Building Systems (ABRASIP) access on 23/05/2013
 Brazilian Association for Compliance and Efficiency Facility (ABRISTAL) access on 23/05/2013
 Solar Law in São Paulo (Law No. 14.459, OF 3 JULY 2007) accessed on 23/05/2009
 SELEGHIM, P.2002. Creating a line of research in instrumentation and industrial control multiphase flows. Text systematization of career Associate Professor. Department of Mechanical Engineering, School of Engineering of São Carlos - University of São Paulo. EESC / USP. 2002.130p.
 Brazilian Labeling Program (PBE) of Inmetro (<http://www.inmetro.gov.br/consumidor/tabelas.asp>), accessed on 23/05/2010