24th COBEM - 2017

24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2076
AIR CONDITIONING SYSTEM: A COMPUTATIONAL NUMERICAL ANALYSIS

Lincon Ulian Tramontano
Reginaldo Ribeiro de Sousa
Eduardo Manfredini Ferreira
Federal University of Grande Dourados (UFGD), Rodovia Dourados-Itahum, km 12, PO Box 533, 79804-970, Dourados-MS
lincon.lut@gmail.com; reginaldosousa@ufgd.edu.br; eduardomanfredini@ufgd.edu.br

Abstract. *The present work has the purpose of analyzing the cold airflow coming from air conditioners of professors' offices selected as models of the upper floor of the Engineering Building of UFGD. Using tools and Ansys Fluent software, a model was made for each office and relevant model data were obtained regarding air behaviour as a temperature and velocity function, taking into account the internal thermal load of each model and the heat generated due to solar incidence on models with windows. Thus, it was possible to analyze whether or not the air conditioners of each model were correctly dimensioned, as well as to recommend the necessary velocity to obtain thermal comfort at 24 ºC. Finally, a proposal for improvement was made using a photovoltaic solar system, as well as the generation and energy saving parameters analyzed with the proposed system for the energy sustenance of the air conditioners.*

Keywords: *Fluid Dynamics, Thermal Comfort, Ansys Fluent*

1. INTRODUCTION

The internal and external environment structures must favour the people and their comfort, which leads to the concept of thermal comfort. The best conditions of life and health are present when the human body is not exposed to thermal stress or fatigue. For this to be avoided, the thermal comfort has been studied progressively.

Some thermal variables such as the temperature, the solar radiation incident (at some point in the study), and the air velocity in the professors' offices are utilized to the thermal comfort definition. The human body is homoeothermic, keeping a constant temperature and producing heat according to its operation as if it was a thermal machine. The body works to lose heat, keeping its constant temperature, generally at 37 ºC (FROTA; SCHIFFER, 2005, p.12-15).

Computational Fluid Dynamics (CFD) is an iterative calculation method, widely utilized with a basic precept of the governing Partial Differential Equations (PDEs) substitution. Iterations are performed according to the profile described by the user, which result in equations as a function of time, space, or a custom variable, making possible to obtain a numerical description of the analyzed control volume (WENDT, 2009, p.3-6).

An interesting fact is that computational analysis reduces prototype creation necessity. Calculations have good accuracy and, as consequence, lower cost to get results from desired situations.

Discretization means partitioning an element from a more complex form. The complex element is transformed into simple elements of infinitesimal size and, when summed, a solution with considerable precision can be obtained.

The distance between these infinitesimal elements composes a calculation mesh. The lower the distance between these elements are, the more accurate the results of the governing PDE will be. Discretization residual errors occur because of the small difference between the PDE's analytical solution and its exact solution (WENDT, 2009, p. 15-18; 87-98).

The number of iterations should be enough to reduce the error to an established level, allowing the linearity of the given result. In other words, the difference between the results of the subsequent iterations is minimal, as shown in Fig. 1 (FOX; MCDONALD; PRITCHARD, 2011, p.195).

2. METHODOLOGY

Most of the inserted data in the Ansys Fluent were obtained either analytically or through the ABNT (2008) and ASHRAE 55 (2013). Thus, the data were collected from the environments to be simulated in different office models. Characteristics and models of air conditioners, the average number of people in the professors' offices, the upper floor plan of the Engineering Building, walls temperatures, as well as the exit velocity of cold air from the air conditioners are the initial parameters to proceed to the simulation.

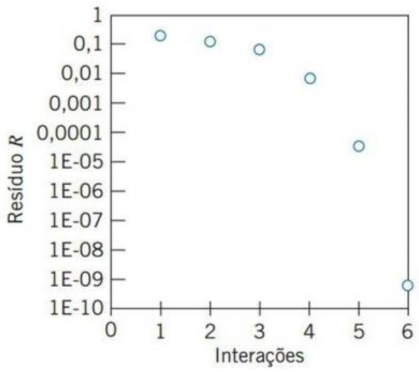

Figure 1: Residue R as a function of the number of iterations.

Velocity measurement was possible by using a hot wire anemometer (Instrutherm, model TAFR-180). A sensor composes this equipment where two needles are fixed. A sensitive element (5 micrometers in diameter and 1,25mm in length) is welded to the end of each needle. This sensor is connected to a circuit, monitoring the electrical resistance variation due to the flow incidence. The device makes correlations to identify which flow velocity would be required to cause this variation. As result, the velocity and temperature of the air passing through the sensor are shown in a digital display attached to the device (MORAES, 2017, p.2-3). The hot wire anemometer utilized for the measurements is shown in Fig. 2.

Figure 2: Hot-wire anemometer utilized in the measurements of velocity.

2.1 Software Used

CFD can be present in day-by-day situations, for example, verification of drag forces and investigation of improvement in the geometry of some materials. More complex analysis includes aeronautical aerodynamics, optimization of internal combustion engines, and analysis of HVAC (Heating, Ventilating, and Air Conditioning) systems in ecologically sustainable structures. Some problems require more attention for the dynamic parts presented, requiring transient analysis; others, only stationary analysis is necessary (ANSYS, 2016).

The Software Engineering, identifying the physical forces and flow characteristics of the situation, revolutionized the fluid dynamics by automating these calculations, even for numerically large problems. Ansys has the Fluent, a powerful computational fluid dynamics software, recognized by the precision of the calculations and how quickly they are made. It is possible to create meshes of different geometric shapes. This offers a smaller number of elements to reach the desired object format, requiring a smaller time and fewer calculations to discretize the element of the study.

Fluent includes solutions for finite elements, used together with known fluids flow modelling, as well as finite elements for viscous fluids or more complex fluids (ANSYS, 2016).

In this work, data from the environments modelled were inserted in order to obtain nontrivial results such as graphs and images, referring to the airflow behaviour along the environment and temperature fields, as well as energy loss due to external factors (influence of the sun, internal thermal load, and occupants).

An important precision application to be cited is the analysis of pressure in the pumps and turbines blades, which may cause failure of the equipment. Currently, with the system analysis by computational fluid dynamics, it is possible to maximize the equipment lifetime and to avoid this problem.

2.2 Thermal load calculation

Sensible and latent heat compose an environment thermal load. The first is responsible for the temperature variation and the second promotes the change of the physical state, which is important for the environment relative humidity. In the studied environments, most of the important equipment for thermal load calculations do not release latent heat, contributing mainly to the internal temperature variation.

The heat lost to the environment due to evaporation is latent, and the sensible heat occurs as a result of the dry exchanges. Many variables are taken into account for the heat production calculation, which is very dependent on the individual's clothing, the activity intensity, the surface temperature of the environment internal elements, and others.

Considering dry exchanges, the conduction has a very small value when the individual is dressed and wearing shoes. In turn, convection depends on the difference in temperature between the air and the body of the individual, as well as the velocity that the air passes through it (FROTA; SCHIFFER, 2005, p.21- 23).

ABNT (2008), "Air-conditioning installations - Central and unitary systems", is divided into three parts: (a) the first is about the facilities design, followed by (b) thermal comfort parameters, and finally (c) indoor air quality. For this study, the ABNT will be indispensable for calculations and acquisition of the necessary parameters for fluid dynamics simulation, such as thermal load data from lighting, occupants, solar factor of windows and walls, and finally, machines present in the professors' offices. Tables 1 and 2 show data areas and total thermal load in the professors' offices.

The cold air velocities were obtained experimentally with the hot wire anemometer that resulted in 2.6m /s, 4.1m /s and 5.2m /s, respectively. Thus, it is possible to start the simulations in the Ansys Fluent.

Table 1. Areas considered for thermal load calculation.

Name	Area (m²)
Model 1	25.15
Model 2	14.00
Model 3	29.95

Table 2. Thermal load values resulting from the models considered.

Name	Occupants	Power (W)	Lighting Area (m²)	Power (W)	Equipment	Power (W)	Total (W)
Model 1	12	1,560	25.15	402.4	12	1,740	3,702.4
Model 2	8	1,040	14.00	224.0	6	870	2,134.0
Model 3	6	520	29.95	479.2	4	580	1,579.2

3. SIMULATIONS RESULTS

Three simulations are required to verify the temperature and flow profile along each model. It will be considered the most critical case of thermal load in each office. The conditioner air temperature was set up at 18ºC, due to the fact the analysis is stationary. This is the average value obtained with the hot wire anemometer.

3.1. Model 1

The mesh generated for the environment discretization of this model contains 702,823 elements of calculation, with 737,986 nodes between these elements. The maximum size of the element, chosen with hexa-unstructured geometry to allow better discretization, is 2 cubic centimetres. The number of current lines was variable between 50 and 100 for all models, always adjusted for a better visualization.

In velocity 1 (2.6 m/s), it is shown the solar thermal charge incident on the window glass and the air energy gain as it passes through the office (Fig. 3).

It is possible to verify the temperature at the beginning and at the end of the simulation environment, with a range from 25 to 28 ºC.

The next simulation is performed, using cold air at 4.1m/s (velocity 2), being visible the temperature difference in the adjacent regions to the model window (Fig. 4).

Velocity 2 reduces the internal temperature of the office in about 3ºC comparing when it is used the air conditioner at velocity 1. Finally, using the maximum air-conditioning velocity of 5.1 m/s, the temperature field is shown in Fig. 5.

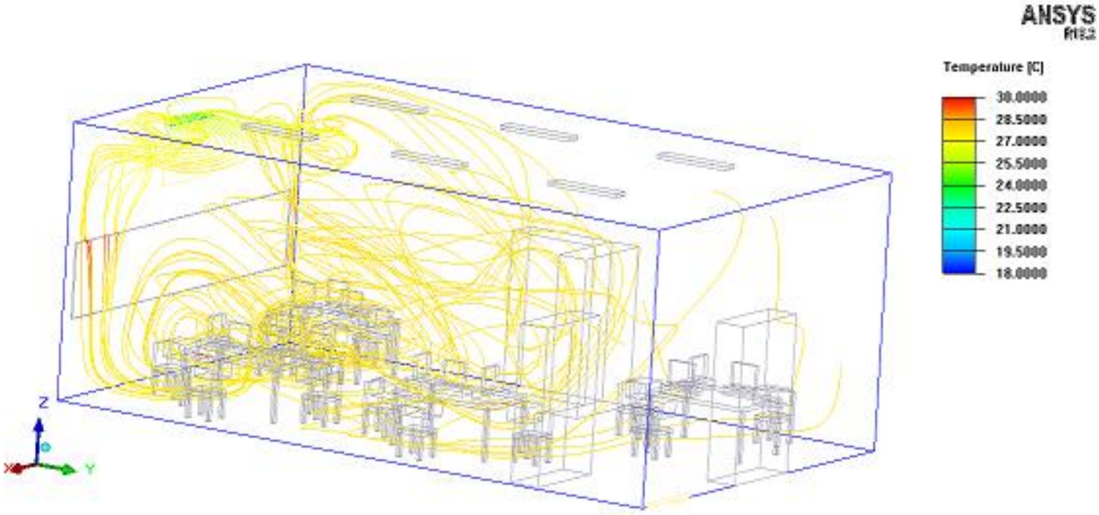

Figure 3: Temperature field of Model 1 to velocity 1 of the air conditioner.

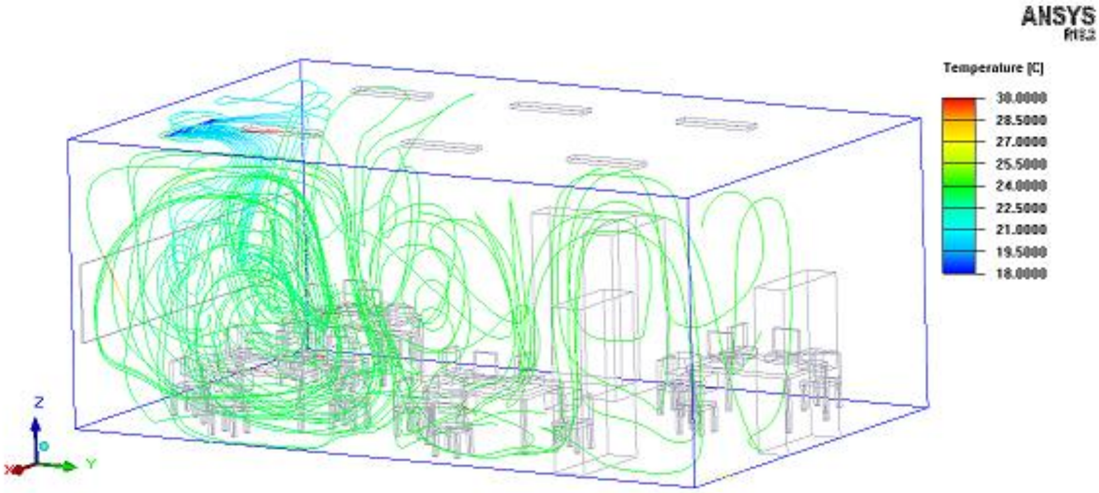

Figure 4: Temperature field of Model 1 to velocity 2 of the air conditioner.

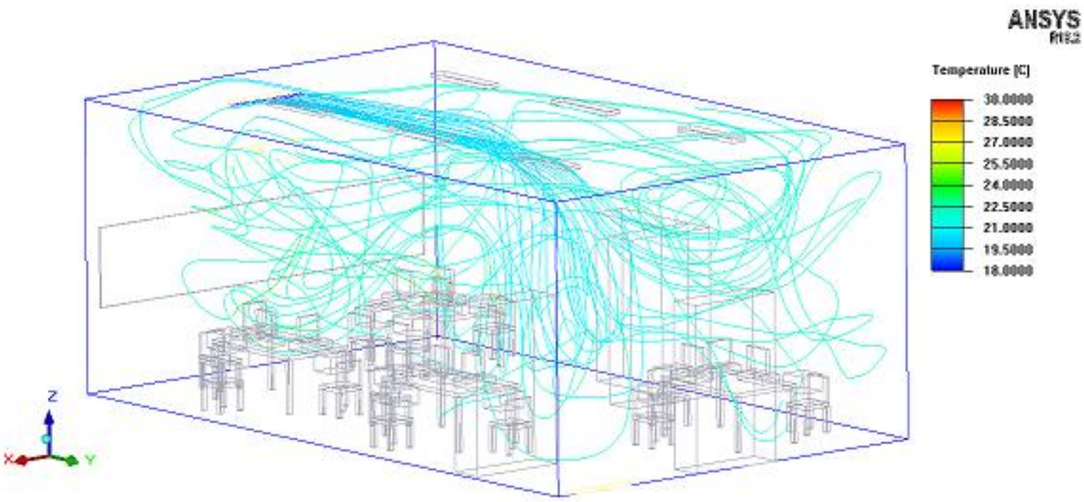

Figure 5: Temperature field of Model 1 to velocity 3 of the air conditioner.

3.2. Model 2

In this model, 86 objects were constructed and the mesh contains 316,454 elements and 335,223 nodes. As in the Model 1, this model has the same maximum size of each element. This was used to establish a pattern in all simulations. Figures 6 to 8 show the results of the simulation temperature field for velocities 1, 2 and 3, respectively.

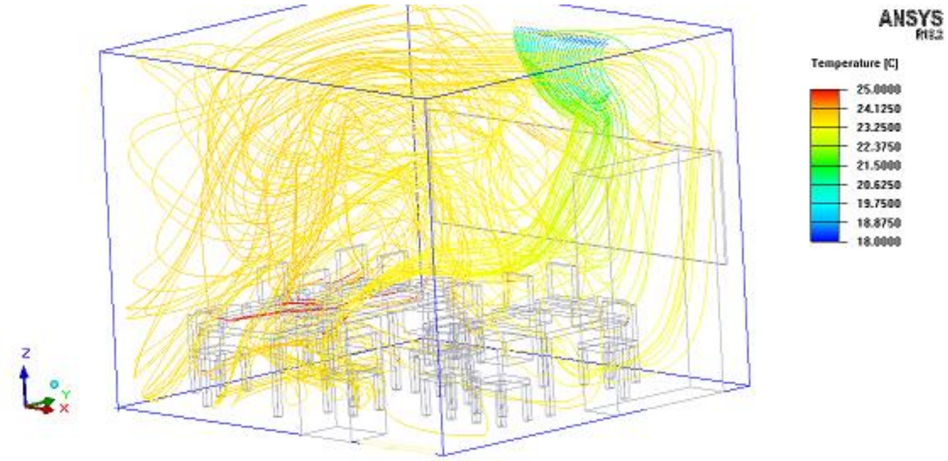

Figure 6: Temperature field of Model 2 to velocity 1 of the air conditioner.

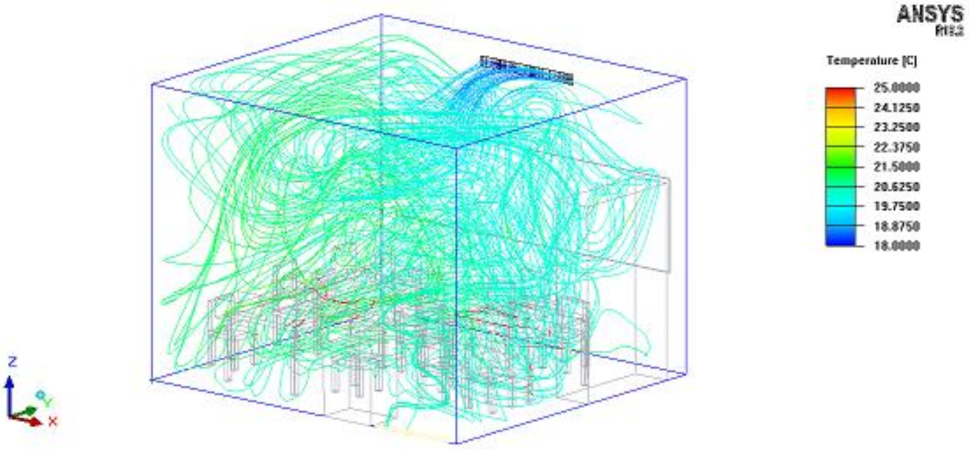

Figure 7: Temperature field of Model 2 to velocity 2 of the air conditioner.

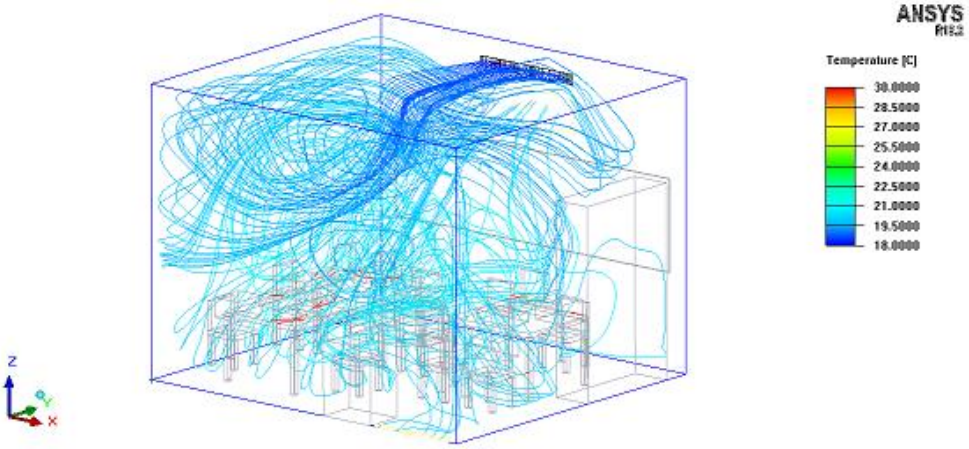

Figure 8: Temperature field of Model 2 to velocity 3 of the air conditioner.

3.3. Model 3

This simulation contains 75 objects, 458,835 elements, and 480,675 nodes. The wall temperature where the air conditioner is installed was set at 31.2ºC (according to ABNT, 2008), due to the solar incidence. The temperature field, in relation to velocity 1, can be seen in Fig. 9. Figures 10 and 11 present the temperature fields for velocities of 4.1 m/s and 5.2 m/s respectively.

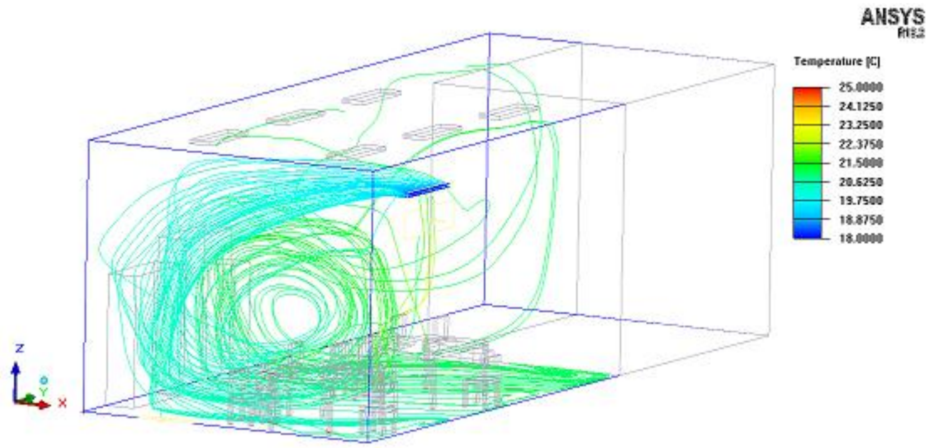

Figure 9: Temperature field of Model 3 to velocity 1 of the air conditioner.

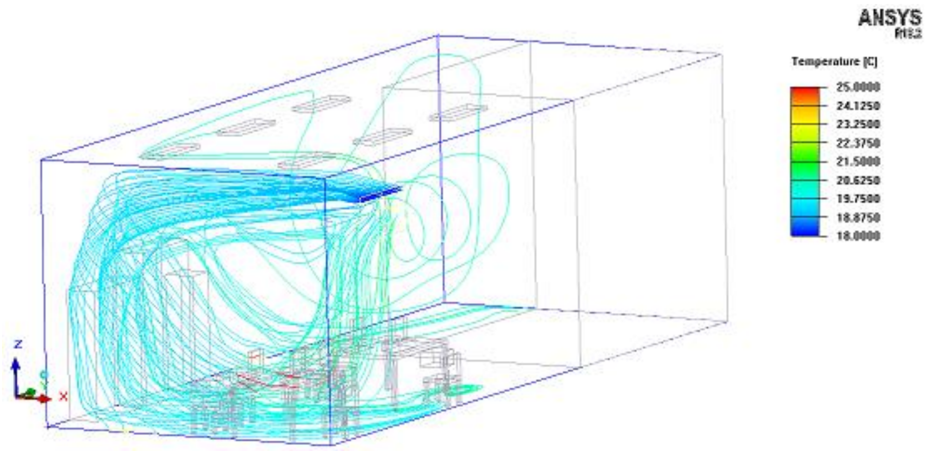

Figure 10: Temperature field of Model 3 to velocity 2 of the air conditioner.

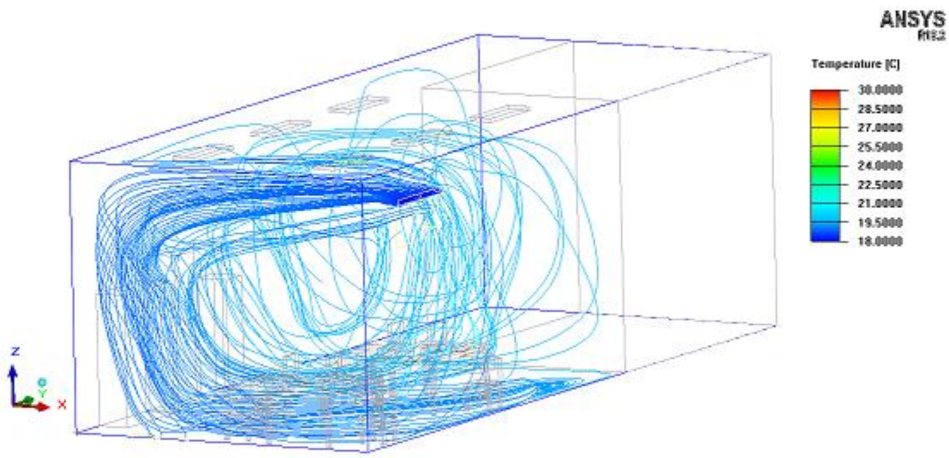

Figure 11: Temperature field of Model 3 to velocity 3 of the air conditioner.

4. DISCUSSION

Considering the ideal internal temperature as 24ºC (ABNT, 2008), the thermal comfort and the worst thermal load scenario of each model, the average internal temperature of each model, as a function of the air conditioner velocity is shown in Tab. 3.

Table 3: Average internal temperature in function of air velocity for simulated models.

Name	Velocity 1 (2.6 m/s)	Velocity 2 (4.2 m/s)	Velocity 3 (5.2 m/s)
Model 1	27 ºC	24 ºC	21 ºC
Model 2	23.5 ºC	21.5 ºC	19 ºC
Model 3	21.5 ºC	20 ºC	19 ºC

For Model 1, velocity 2 in the air conditioner is recommended, while for Model 2 velocity 1 can be utilized. This is sufficient to maintain the office's internal temperature in a range that does not cause noise or low temperatures, bothering their occupants.

For Model 3, the air conditioner can be considered oversized, once there is no solar incidence due to the absence of windows. This is perceived when it is used the lowest velocity, so the conditioner usage results in a temperature out from the thermal comfort range. As consequence, thermal discomfort affects the occupants, being recommended the replacement of the air conditioner.

The air conditioners of Models 2 and 3 in velocity 1 presented a temperature below the ideal. As the result of the situation of higher thermal load, these devices are oversized for these environments, since people and internal equipment generating thermal load in the considered assumption is rarely real. As a proposal for improvement and energy savings, the ideal condition would be changing the air conditioning units by others with lower cooling capacity, which requires previous studies and new simulations.

5. IMPROVEMENT PROPOSAL

To improve the system installed in the Engineering Building, there are viable options, such as *brise-soleil* placement, but these elements only decrease the solar incidence on the windows and walls. In search of a generalized improvement, it was considered a photovoltaic system for electric generation support of the upper floor air conditioners.

To realize a good sizing in this system, it is necessary to know the air conditioners electrical data. There are 24 identical conditioners, 24,000 BTU/h each. Through the design data available on the manufacturer's website, the total rated power of the RPC24CP evaporator unit in concomitant use with the RAP24BL condenser unit is 58.56 kW.

Considering an 8-hour daily use of the air conditioners with the compressor working 20 minutes per hour, the average monthly consumption is 3446.74 kW. The photovoltaic system to supply this demand has its manufacturing data described in Tab. 4.

For comparative purposes, the photovoltaic system analyzed enables energy saving of 97.23%, according to the solar calculator (Neo Solar, 2017). Since this sizing is just an estimate, it is recommended to make a budget with specialized companies to guarantee these values. However, the values of this website provide a good approximation for this study. As the purpose of this paper is not an economic analysis, cash flow and payback will not be evaluated. A more detailed analysis of this system is presented in Tramontano, L.U. (2017).

Table 4: Description of equipment for an auxiliary photovoltaic system for the Engineering Building.

Description	Quantity	Power capacity (kWp)	Required area (m²)
Yingli Solar Panel 275Wp	128	35.2	217.6
Fronius Inverter 15kW	2		

6. CONCLUSION

Thermal comfort is an important issue when using air conditioning, aiming at the lowest power consumption. Thus, manufacturers try to produce their air conditioners with the maximum possible cycles and systems efficiency. Distributed and localized head losses are inevitable as they are related to variables such as roughness and length of pipes present on the system.

Pre-studies of the environment becomes useful when generating a reliable simulation, saving time and money. The simulation makes unnecessary to build prototypes; however, computational processing is required to finalize iterations of complex systems with a high number of mesh elements. CFD is an important area which tries to produce realistic simulations involving flows, vibrations, thermodynamics, and aerodynamics.

Several region graphs of the simulated models can be plotted and animation can be made through series of sequential photographs that shows the interaction of the elements with the analyzed stream. Since discretization is governed by a computational mesh where PDEs are used, projections can be made to make the transition from analytical to experimental as reliable as possible. As energy management concepts become more pronounced over time, studies are conducted to develop methods to reduce the energy consumption of a system. The use of renewable energies and load modulations are among the techniques used to reduce energy billing.

Other auxiliary systems for the photovoltaic system presented in this study can be used to generate electricity to air conditioners. Among these systems, wind turbines or even biodigesters can be sized, depending on the ease of local resources.

One hypothesis is the replacement of air conditioners by a single central air conditioning system, which can be made using Ansys Fluent software. As a result of this analysis, it can be said that air conditioners are oversized since the situations are the worst possible and rarely happen.

At certain simulated speeds, due to the high flow of cold air, air pockets are formed at the boundaries, leaving the occupants uncomfortable. A solution would be to use the air-conditioning oscillating function to direct the airflow in a distributed way. Finally, the possibility of using an auxiliary photovoltaic system proved to be efficient, generating large energy savings.

7. ACKNOWLEDGEMENT

Federal University of Grande Dourados (UFGD) supported this research.

8. REFERENCES

ABNT – *ASSOCIAÇÃO BRASILEIRA DE NORMAS TÉCNICAS*, 2008. "NBR 16401: *Instalações de ar-condicionado - Sistemas centrais e unitários: Projetos das Instalações*". Rio de Janeiro.

ASHRAE 55 Standard, 2013. "Thermal Environmental Conditions for Human Occupancy". 4 Fev. 2017. <https://www.ashrae.org/resources-- publications/bookstore/standard-55-and-user-s-manual>.

Fox, R. W., McDonald, A. T and Pritchard, P. J, 2011. "*Introdução à Mecânica dos Fluidos*", LTC, 7ª ed.

Frota, A. B. and Schiffer, S. R., 2005. "*Manual de Conforto Térmico*". Studio Nobel, 1ª ed.

Moraes, C. C. F., 2017. "*Anemômetro a fio quente*". 10 Mar. 2017 < http://www.dee.ufrn.br/~luciano/arquivos/ins_ele/Apresenta%E7%F5es_2008_2/CAYO%20CID/Instrumentao_Eletrnica_Cayo_Cid_200321285_AFQ.pdf >.

Neo Solar, 2017. "Simulador Solar: Calculadora Solar Fotovoltaica". 5 Mar 2017. <https://www.neosolar.com.br/simulador-solar-calculadora-fotovoltaica/>.

Tramontano, L.U., 2017. "*Análise numérica computacional de um sistema de ar condicionado*". Course Final Paper. *Universidade Federal da Grande Dourados*.

Wendt, J. F., 2009. "Computational Fluid Dynamics: An introduction". Springer.

9. RESPONSIBILITY NOTICE

The authors are only responsible for the printed material included in this paper.