DNS

DESIGN and NUMERICAL SIMULATIONS

# MULTIWAY – R01

<u>Technical Report:</u> Determination of temperature field and velocity in racks of computers.

## 1. INTRODUCTION

Fluid flow simulations using CFD techniques (computational fluid dynamics) are currently employed in issue where there is a need to meet important physical parameters for dimensioning the components used, for example, in the implementation of computer centers (data center). In air-conditioned rooms, which temperature control is necessary, the fluid-mechanical design becomes an important phase of the project, where the determination of the performance of each component is crucial for thermal calculations and fluid dynamics involved in projects of this nature. In this regard, the CFD techniques are well suited for such evaluations, since it is possible to employ physical models that describe the fluid dynamics in order to determine the fields of temperature, pressure and velocity within the air conditioned room.

In this work, CFD simulations for thermal analysis and flow through racks of computers for the purpose of determining the physical parameters of interest, are presented: velocity field, pressure and temperature coming from the flow through racks of computers.equipped with heat sources, contained in a heated room type Data Center.

## 2. DESCRIPTION OF WORK

This study aims to verify, through CFD, three proposals for geometric configurations for the rack as follows:

- Perforated Screens;
- Shutters the front door, back door and punched the screen;
- Shutters on the front and rear doors;

- Electronic equipment modeled as boundary conditions the output of the flow (in the front air intake of the server) and flow entrance (entrance at rear air server);

- Check front and rear air modeled as an area corresponding to 55% of the frontal area of a 1U server passage;

- Continuity of mass flow imposed between the boundary conditions of exit and entry of air to each of the servers;

- Operating Curve Fan's extracted server manufacturer catalog;

- Thermal Power dissipated by server imposed as a boundary condition at the air outlet in the rear of the server.

Was chosen for the present analyzes, the Dell PowerEdge R610 (Fig2.1). The characteristics of the equipment are listed below:

- Geometry 1U x length 28"

- 2 Intel ® Xeon E5645 processors, each with 6 colors

- Intel ® 5520 Chipset

- 64 GB memory

- 4 SAS hard drives

- Source 500W

We also intend to evaluate the influence of the direction of the cold air from the air conditioning system using grid floor at an angle of entry of 15 degrees.

Therefore, it was requested to Multiway the geometries of the rack and the various port configurations as well as the dimensions and layout of the racks in a room typical IT (data center), where these geometries will be evaluated numerically environment. It is important to note that the characteristics of the flow in the racks are strongly influenced by the geometry and arrangement of elements of the environment where they are located.

Ideally, it is also necessary to characterize - geometry, layout, thermal dissipated power - the electronic components as well as the specification of the performance of the fans responsible for ventilation of the same curve. Since the specification of such equipment is the responsibility of the end customer Multiway products, and also that one of the main purposes of using racks is allowing great flexibility for the installation of different types of equipment, it becomes difficult to specify a configuration equipment universally contemplate all possible options, so it was decided by the arbitrary choice of a typical setup for the analysis in question - namely, populated rack for 42 bi-processor 1U servers dimension, operating at full load. The characteristics required for the analysis of CFD were then obtained and datasheets provided by the server manufacturer. Some of the data required in the absence of more accurate sources of information were estimated following the best criteria available in the literature and technical catalogs.

It should also be noted that the scope of the present work is the evaluation of the aerodynamic and thermal motions of the different geometries of racks Multiway features, and not exactly the flow inside the electronics contained in the rack. Thus, the electronics were modeled as follows for the analysis of CFD.

contatos@dnsimulations.com.br
http://www.dnsimulations.com.br
CNPJ 14.004.160/0001-13

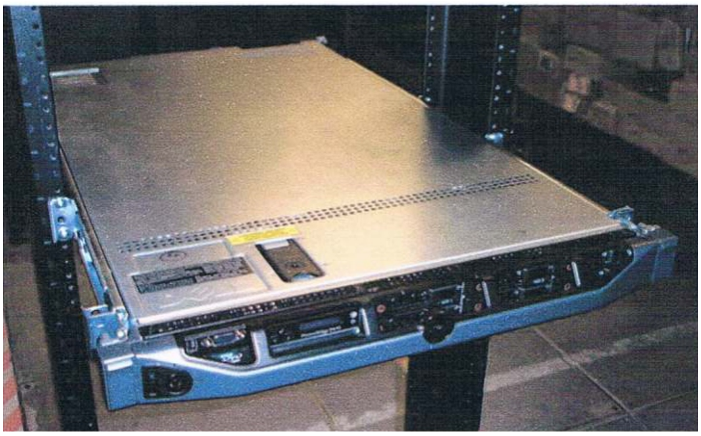


Fig2.1 Servidor Dell PowerEdge R610

To determine the power dissipated by the device, the web tool available was used at the manufacturer's site, from where[1] you obtained the following information:

- Dissipated power: 320W (maximum load)
- Air Flow: 26.2 CFM / server
- Temperature variation between inlet and outlet: 21 ° C

To determine the characteristics of the ventilation server - essential for evaluation of the heat exchange thereof - it is necessary to model the variation of the air flow according to the variation of the operating conditions (pressure, temperature).

Typically, this can be done by modeling the ventilation fans if the server by a performance curve (pressure x volume flow). This information is provided by the manufacturers of fans through technical catalogs.

However, this information alone is not sufficient to completely describe the behavior of the server mechanism ventilation. It is also necessary to determine the levels of loss and impedance vent server (chassis and components), depending on the flow of ventilation air that passes through it.

[1] Dell Energy Smart Solution Advisor - http://essa.us.dell.com/DellStarOnline/DCCp.aspx

This information is not typically provided by the server manufacturer , since it depends on a number of characteristics , such as the geometry of incoming and outgoing air into the chassis , quantity and arrangement of internal components ( hard drives , processors, memory , power power ) . The lifting of the impedance curve can be done experimentally by scanning the entire operating range of fans and measuring velocities and pressures at the entrance and exit of the server , or by means of CFD simulations in the full geometry of the cabinet and server components, for various operating conditions of the fans. However, these options are beyond the scope of this work.

Thus , we chose to apply a total impedance curve for a typical full rack,as shown in Fig.2.2. This curve was obtained through technical publication of Dell manufacturer for a model similar to Multiway rack . The complete impedance curve shows [2] the overall behavior of the rack,comtemplating all losses and impedances due to runoff and its components in the rack( perforated doors , wiring , etc) as well as on own servers .

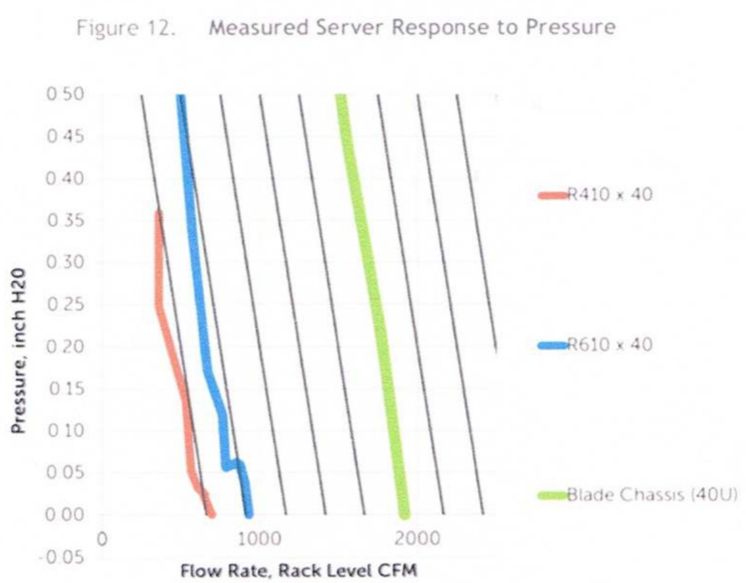



Fig2.2 Curva de impedância combinada para servidor e rack

[2] Dell White Paper: Thermal Design of the Dell PowerEdge T610, R610 and R710

Therefore, specified through the combined impedance curve flow corresponding to the maximum air flow ventilation possible for the server at a given operating point, and changes in the flow of air through the server are due only to variations in the pressure field and velocities in the Ti room (data center) environment. The effect of such variations in air flow vent server will be computed by imposing this operation curve in the CFD model.

The geometries of the IT room and the racks were provided by Multiway in typical CAD formats. Figure 2.3 shows an overview of the Ti room, with only one of the racks positioned.

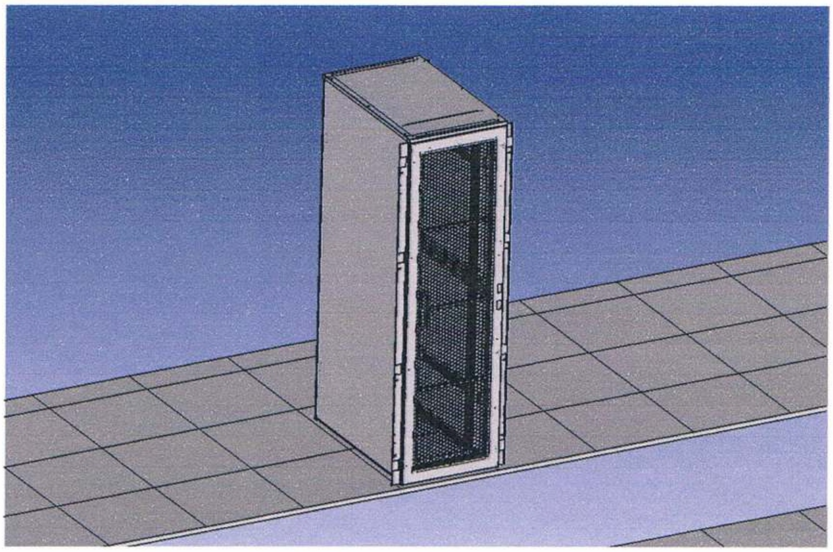


Fig. 2.3 Rack 44U Multiway

The geometries of Ti room and the racks were then converted and processed for the purpose of generating the domain discretization (generating computational mesh). It is noted that the preparation of the geometry for meshing step consumed a large part of the workload for the present analysis because the geometric model provided was not in an appropriate setting (excessive details like rivets, screws, thin sheets, etc.) for CFD analysis. Thus, a simplification of the geometry was necessary to continue the generation of computational mesh (Fig. 2.4) step.

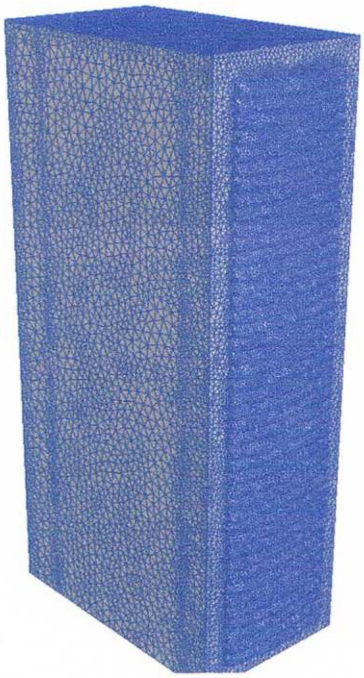


Fig. 2.4 Malha Computacional de superfície

After preparation of the computational mesh for each of the proposed geometrical, then it moved to the numerical modeling of the problem under analysis, in terms of boundary conditions, fluid properties, and mathematical modeling of the equations that describe the phenomenon - the Navier-Stokes. I will now briefly present the numerical models used in the analyzes.

## 3. APPROACH MATH, NUMERICAL AND BOUNDARY CONDITIONS

In this section, we describe briefly the mathematical and numerical models used, as well as the condition imposed on the contour problem.

## 3.1 Mathematical Model

The full Navier-Strokes (conservation equations of mass, momentum and energy) are solved permanently.

## 3.2 Boundary Conditions

The boundary conditions used were supplied by the client (input data). The boundary conditions for the room were specified by IT Multiway and cover two as the boundary condition of entry of air from the air conditioning system situations:

- Inlet temperature of 12 º C;
- Plate with high flow normal to the surface of the floor outlet;
- Plate with high flow tilted 15 ° toward the exit doors of racks.

The other conditions are the contours of related equipment (fans, servers, etc.) and have been reported in Chapter 2. For modeling the operation of servers, it is assumed that they work at full load, with total power dissipated assumed according to the data provided by the manufacturer. It is also assumed an upper limit for the temperature of air equal to 60 ° C, above which the logic servers exhaust thermal protection of the processor spends to reduce the clock rate of the same (speed step mechanisms). Thus, temperature levels of this order grandez are assumed as a failure to the server and therefore undesirable.

## 3.3 Method of Solution

The solution of partial differential equations that model the problem was determined by the geometry of discretisation applying the technique of finite volumes. The resulting system of equations was solved by a second-order polynomial interpolation and the pressure-velocity coupling was done through the scheme known as SIMPLE.

For that, we used a mesh unstructured for analysis of computer rack. The system of algebraic equations was then solved by a segregated iterative process using a multigrid solver applied to the equations of momentum and energy. Convergence was taken on the review of various wastes. In addition, the evolution of temperature and velocity was monitored at various points within the computational domain and convergence was determined when such values of temperature and velocity indicated steady state.

A package of software produced by CFD OpenCFD Ltd. This toolset called OpenFOAM ® has an extensive range of resources to solve complex flow problems involving chemical reactions, turbulence, heat transfer, multiphase flow, among others was used.

## 4. RESULTS

This chapter presents the results and promotes discussions about the equipment analyzed. For didactic purposes, these results will be presented in three parts. The first provides a comparative analysis of the three proposed geometries.

The configuration shown in Figure 4.1 a - Geometry named 1 - shows the perforated doors front and back screens of the rack, as depicted in Figure 4.1-b (referred Geometry 2) has shutters on the front door and the rear door perforated screens the rack. Already shown in Figure 4-1-c (Geometry 3) has shutters on the front and rear doors

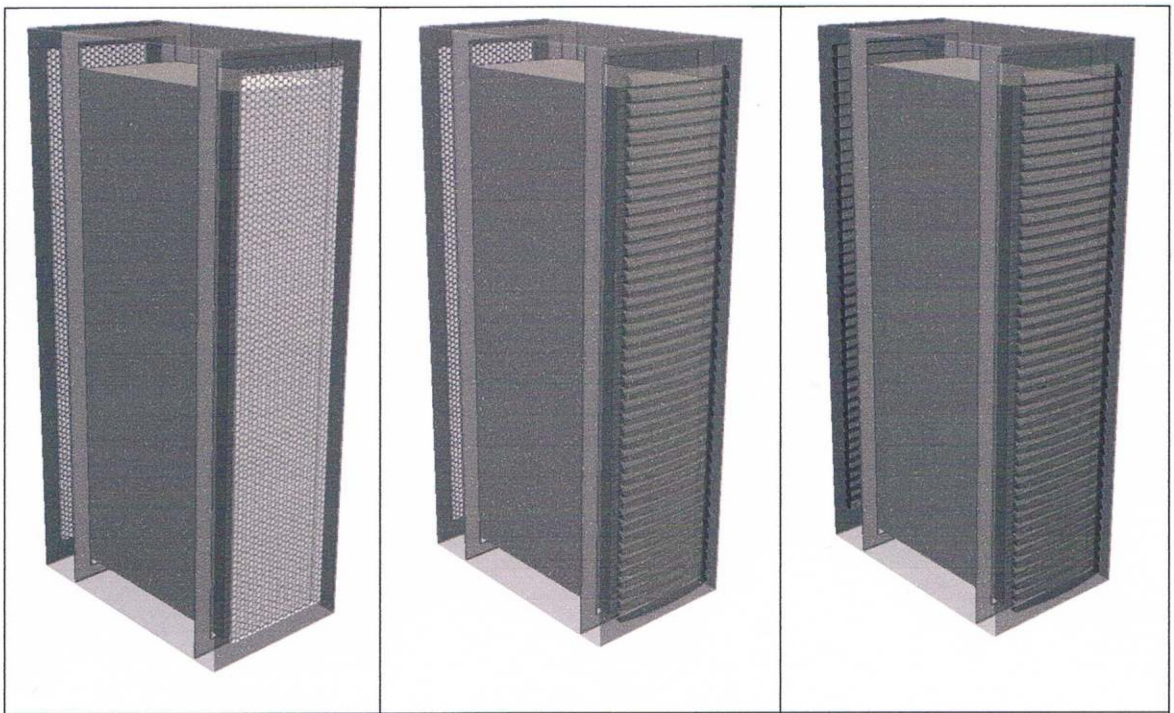


Fig. 4.1 a- Geom. 1 | Fig. 4.1 b- Geom. 2 | Fig. 4.1 c- Geom. 3

In the second part of this chapter the results obtained concerning the configuration of the grid supply air are presented, ie, generic grid with angle normal air supply to the floor of the data center room, and grid proposed by Multiway, which imposes the drainage holes one angle of 15 degrees inclination.

In the third and last part is discussed under the light of the results of CFD, the possible causes of overheating generated at the bottom of the rack and suggests simulations to evaluate changes that bring improvements in the thermal performance of your equipment analyzed.

## 4.1 PARTE 1 - Comparação entre geometrias das portas do rack

The figures shown in this sub-section show a comparison between the three geometric configurations analyzed. It is hoped this part analyze the flow by physical parameters that can be inferred about the thermal performance to geometric variations used in ports on both the front as the rear rack. For this, all results presented in Part 1 of this chapter were obtained from CFD simulations, performed on the basis the same operating .

conditions, so with the same boundary conditions. As the only difference between the simulations are of geometric nature, the analysis developed here will only evaluate the geometric effect in flow concerned.

All figures are presented in threes, where the letters a, b and c correspond to the geometries described in figures Figure 4.1 abc, respectively, Geometry 1, 2 and Geometry Geometry 3. The boundary condition used for the input stream (floor-grille) is the angle of incidence equal to zero.

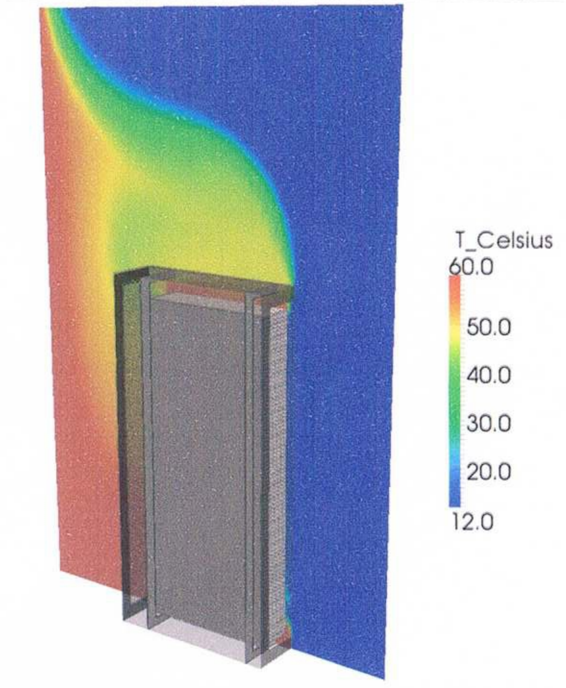
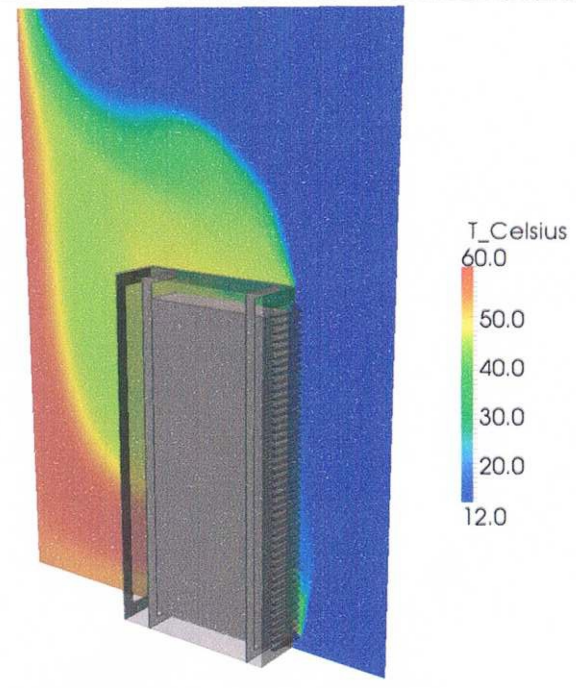
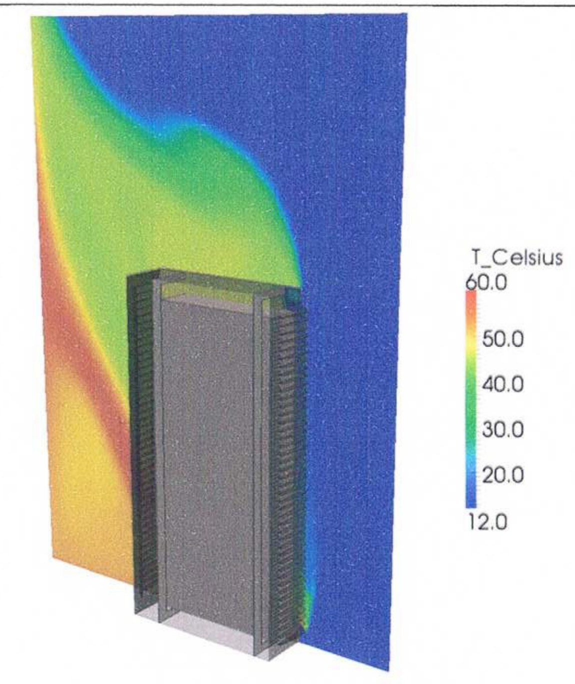
The abc Figure 4.2, show the temperature field recorded in a median plane perpendicular to the doors of racks. You can see that the temperature field on the warm side of the rack (external to the rack) has less severe distributions for Geometry 3, showing a gain in performance as the use of shutters on the doors (front and rear) equipment. The same occurs with Geometry 2 (Venetian Screen +), compared with only 1 Geometry, composed solely of screens in front and rear doors.

Figs 4.3abc show parallel to the floor of the data center that supports the conclusion obtained in the preceding paragraph cuts, that is, Geometry 3 is the better from the standpoint of cooling (lower temperature on the hot side of the rack), followed by geometry 2, and one end of the geometry, which is the least efficient of the three geometries analyzed.

Figs 4.4abc show the temperature profile obtained on the front doors. It can be seen better temperature distribution geometries for 2 and 3, with temperatures very similar fields. Geometry 1 shows a large expanse of regions with levels of temperature around 60 º C on the front, which indicates a possibility of overheating of servers. Further details of the phenomena that lead to the occurrence of these high levels of temperature will be raised in the next topics.

In 4.5abc figures , shows the temperature distributions in planes parallel to the rear doors . These figures show a decrease in the levels of temperatures when using shutter is to Geometry or Geometry - 2 - 3 - although these levels may still be considered high especially at the bottom of the rack , which demonstrates the possibility of occurring again overheating of servers . Through these figures is not possible to quantify the relative gains Geometries 2 and 3, but it is clear that the use of blinds , especially on the front door , promotes gains in thermal performance . It can be inferred that the major contribution to cooling is due to the use of shutters on the front, and that the use of screens or blinds in the rear doors do not promote significant changes in the flow .

Finally, Fig 4.6abc , show the temperature field on the walls of the servers . In this case , there is an evident decrease of temperature levels on the walls of the servers . In this case , it is evident that decreased levels of temperature using shuttered settings . These figures show the gains obtained with the use of blinds , especially on the front rack door .

| | | |
|---|---|---|
|   |   |   |
| Fig4.2 a Field Temperatures, Geom1 | Fig4.2 b Field Temperatures, Geom2 | Fig4.2 c Field Temperatures, Geom3 |

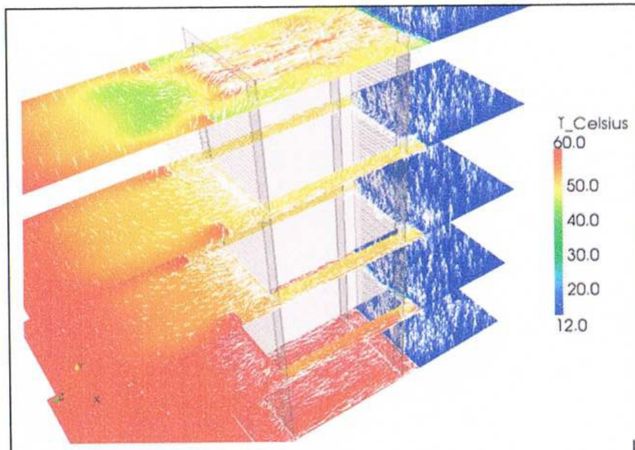



Fig4.3 a Field Temperatures, Geom.1

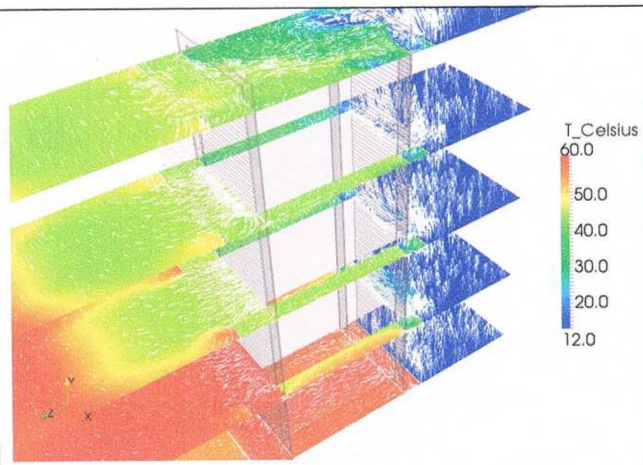



Fig4.3 b Field Temperatures, Geom.2

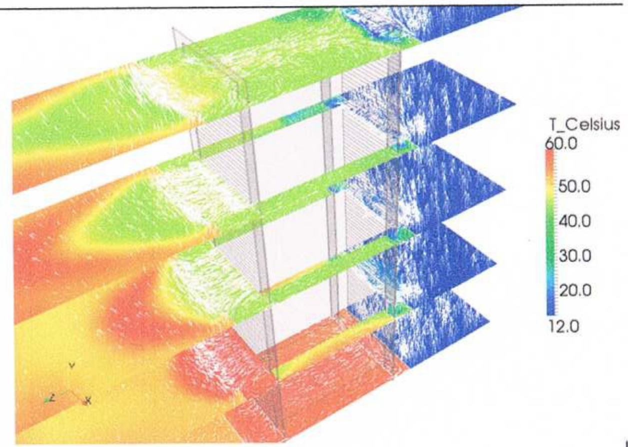



Fig4.3 c Field Temperatures, Geom.3

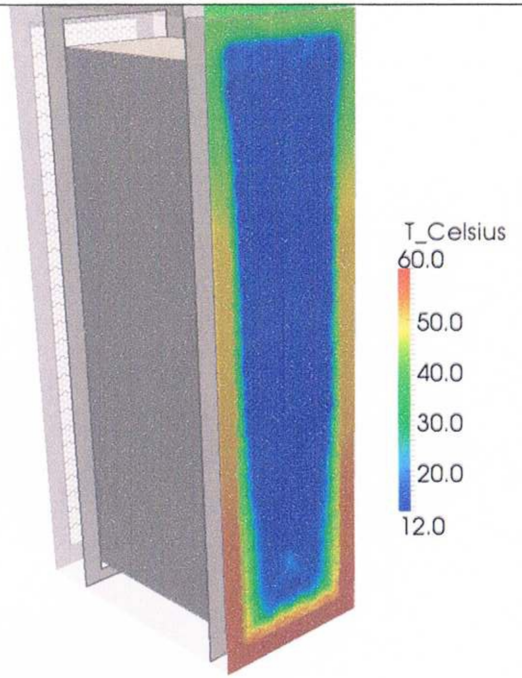



Fig.4.4 a - Field Temperatures. Front Door. Screen

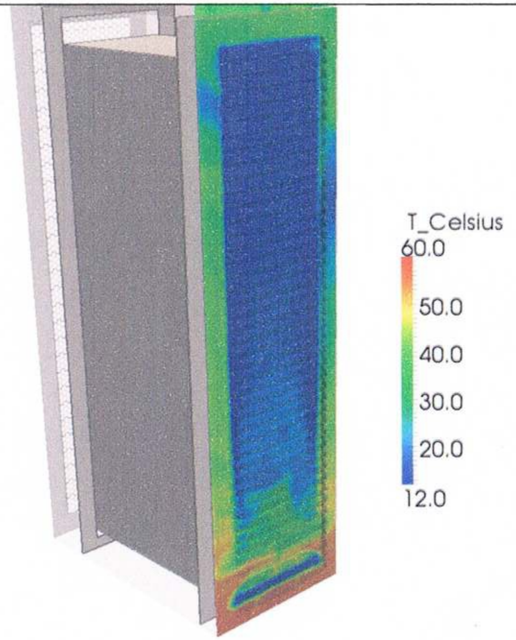



Fig.4.4 b - Field Temperatures. Front Door. Shutters

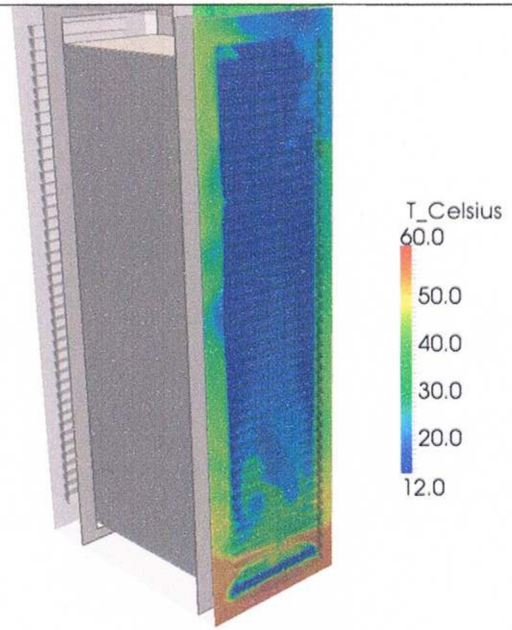
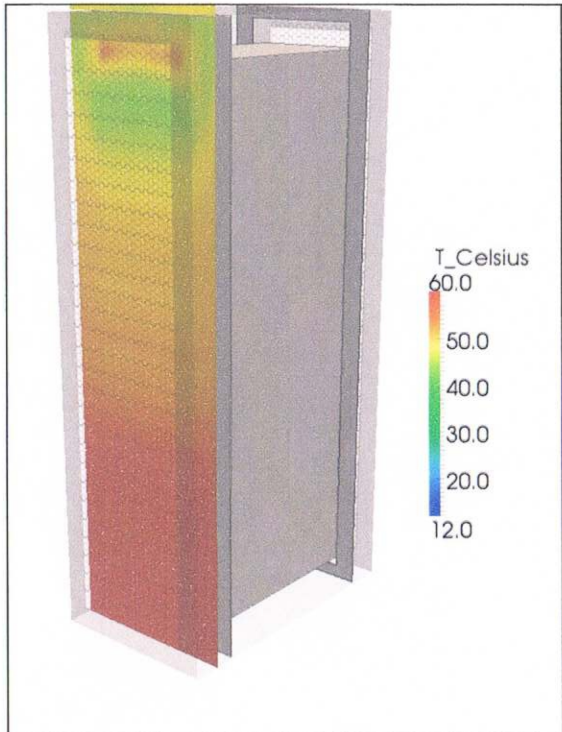
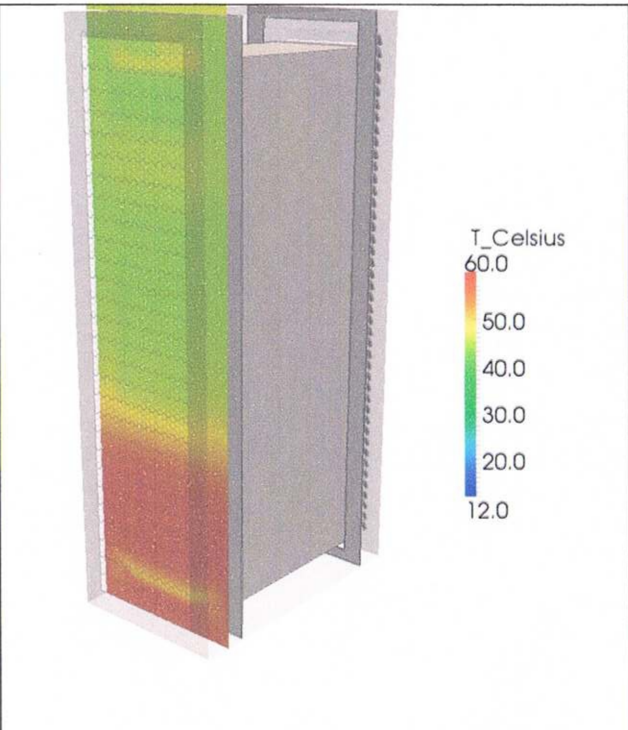


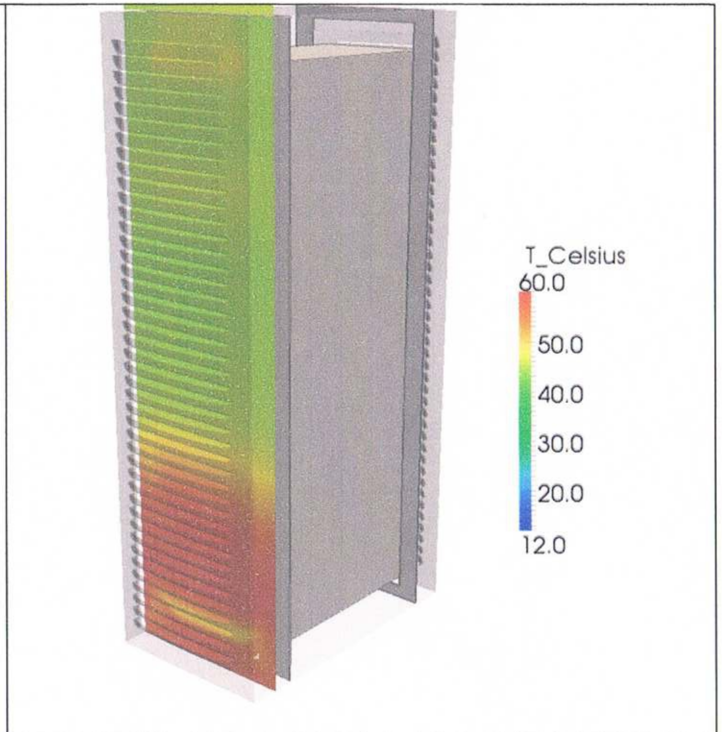

Fig.4.4 c - Field Temperatures. Front Door. Shutters



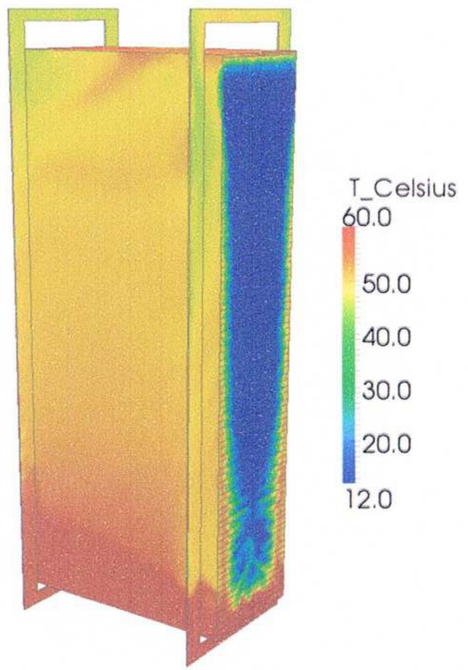

Nas Fig.4.5 a - Field Temperatures. Near the Back Door. Screen



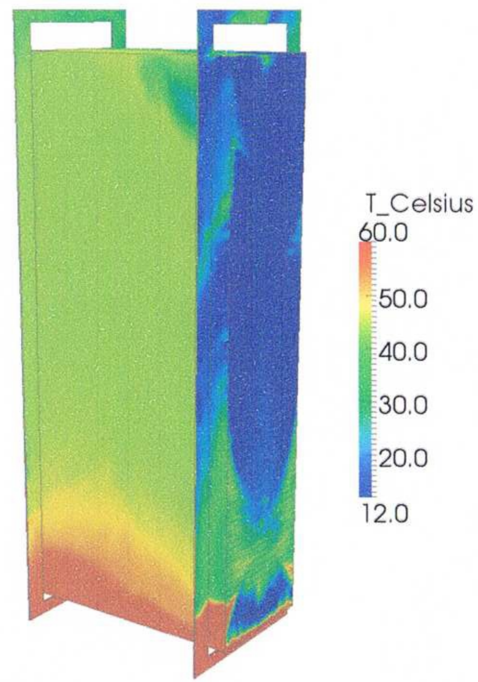

Nas Fig.4.5 b - Field Temperatures. Near the Back Door. Screen



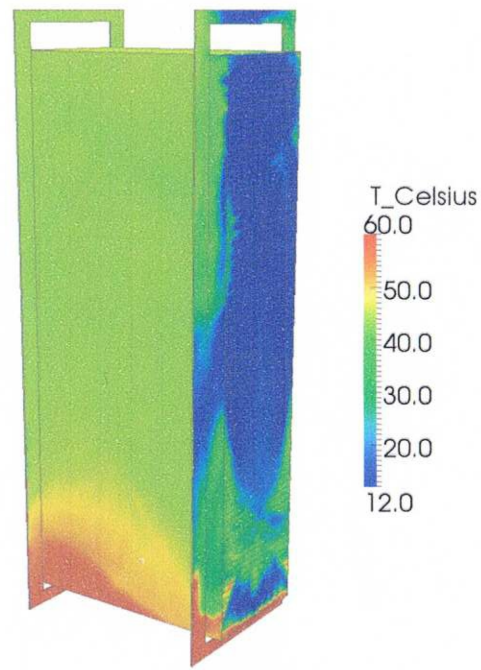

Nas Fig.4.5 c - Field Temperatures. Near the Back Door. Shutters




Nas Fig.4.6 a - Field Temperatures Wall of the Servers. Geom.1




Nas Fig.4.6 B - Field Temperatures Wall of the Servers. Geom.2




Nas Fig.4.6 c - Field Temperatures Wall of the Servers. Geom.3

## 4.2 Part 2 - Comparison between the cooling supply air

The results presented in this section are intended to identify potential performance gains related to the use of grids of common ground , which cold air is blown vertically in the Data Center , in relation to employment of the grid proposed by Multiway with cold air blown with an inclination of 15 degrees with respect to the vertical plane.

Figures 4.2 abc , then represent the results for the three geometries studied , with the flow of cold air coming through the conventional grid floor with zero incidence angle ( perpendicular to the flow entering the floor of the room Data center ) . Already in Figures 4.7 abc , simulation results using the same three geometries discussed here are presented , but with the entry of cold air flow with angle of incidence of 15 degrees , thus simulating the proposed Multiway the floor . A comparative analysis between the results obtained with zero angle of incidence ( figure 4.2 abc ) and incident angle 15 degrees ( figure 4.7 abc ) , lets say that the effect of using the floor with inclined flow (floor Multiway ) , promotes a strong reduction levels of outlet temperature of the rack. Figures 4.8 abc 4.7 abc represent the figures in different scales for better viewing.

For comparison purposes, the following figures 4.5 abc represent the temperature distribution in planes parallel to the rear case where the injection of cold air occurs at zero angle to the normal to plans floor doors. Equivalent to these Figures, but with the boundary condition requires that mass flow of cold air with an inclination of 15 degrees relative to the normal to the floor by means are shown in Figures 4.9 abc. It can be seen that the temperature level in the region in question has fallen by around 30%, showing the thermal efficiency arreficimento using the inclined grate (proposed by Multiway).

I

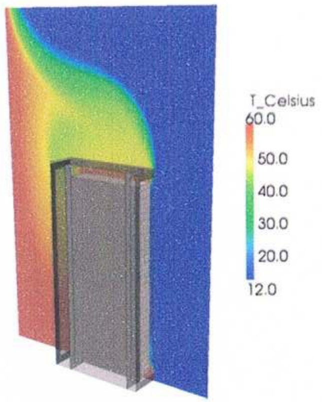


Fig.4.2 a - Temperatures.Geom.1, Grid 0º

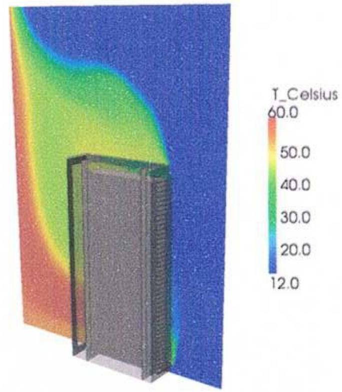


Fig.4.2 - Temperatures.Geom.2, Grid 0º

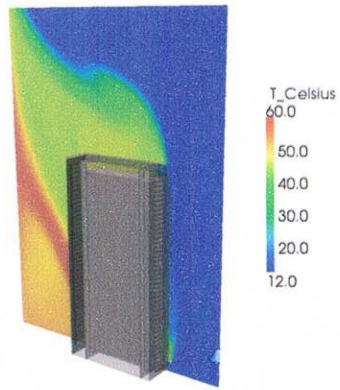


Fig.4.2 c - Temperatures.Geom.3, Grid 0º

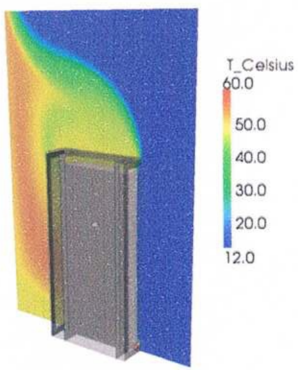


Fig.4.7 a - Temperatures.Geom.1, Grid 15º

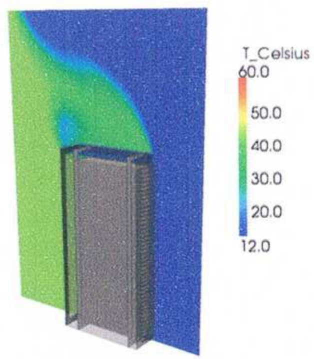


Fig.4.7 b - Temperatures.Geom.2, Grid 15º

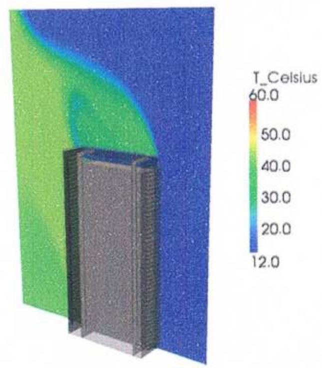
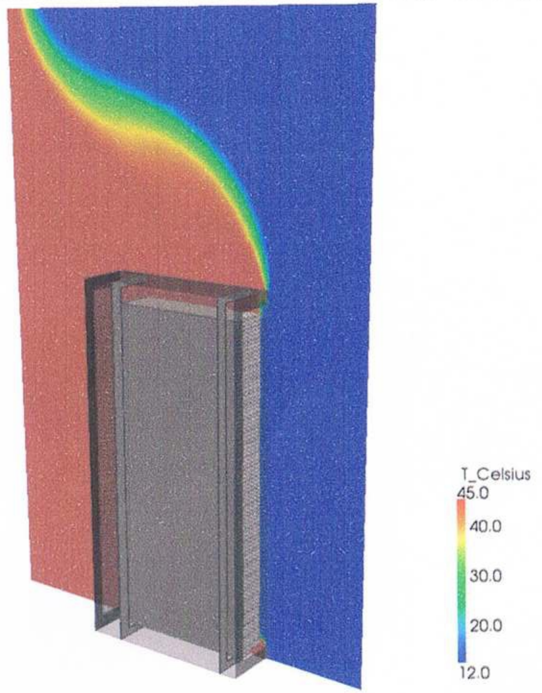
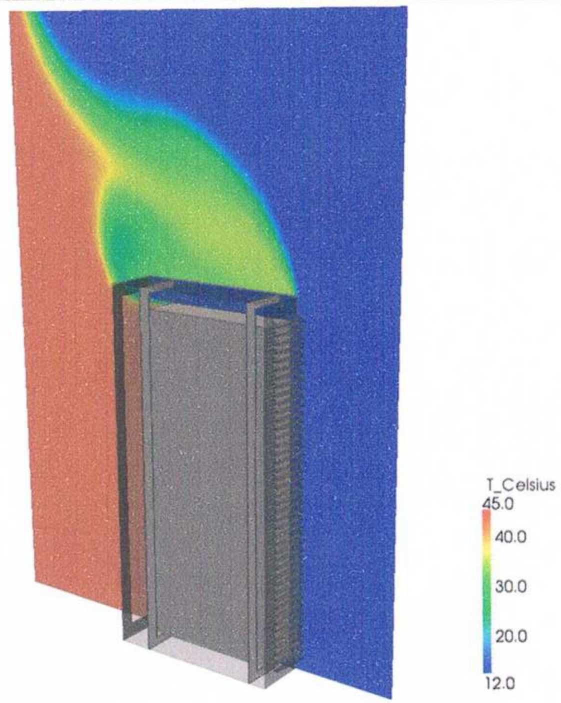
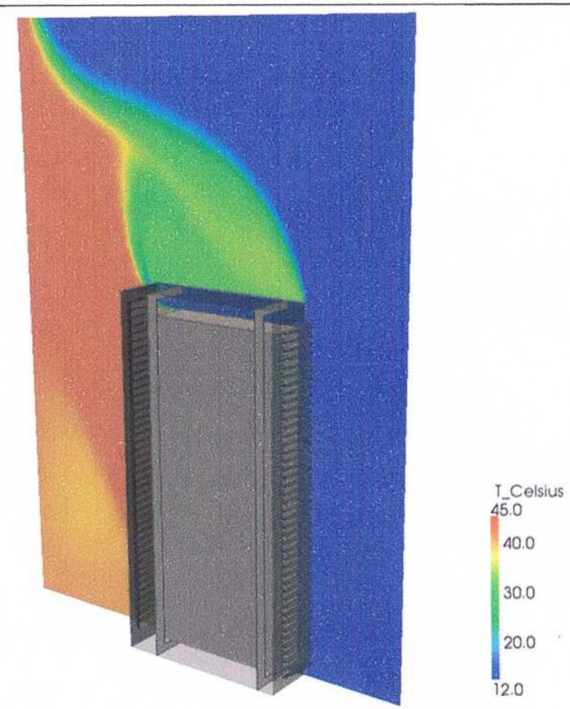


Fig.4.7 c - Temperatures.Geom.3, Grid 15º

| | | |
|---|---|---|
|   |   |   |
| Fig.4.8 a - Field Temperatures.Geom.1 | Fig.4.8 b - Field Temperatures.Geom.2 | Fig.4.8 c - Field Temperatures.Geom.3 |

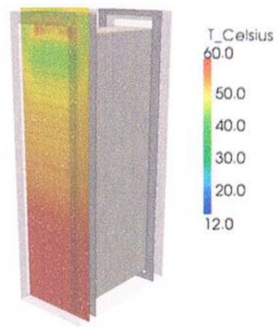
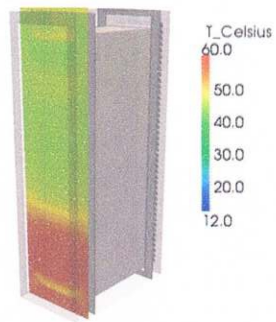
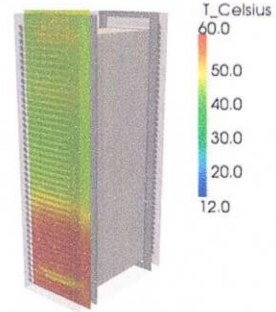



Fig.4.5 a - Field Temperatures.Geom.1, Grid 0º




Nas Fig.4.5 - Temperatures.Geom.2, Grid 0º




Nas Fig.4.5c - Temperatures.Geom.3, Grid 0º

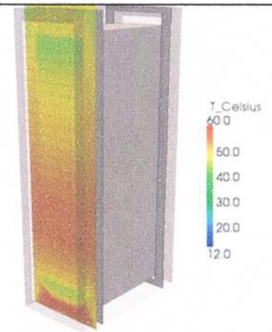



Fig.4.9 a - Field Temperatures.Geom.1, Grid 15º

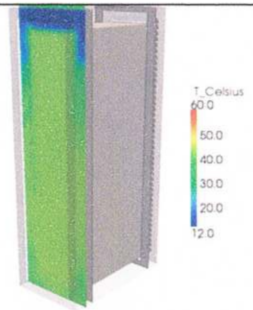



Fig.4.9 b - Field Temperatures.Geom.2, Grid 15º

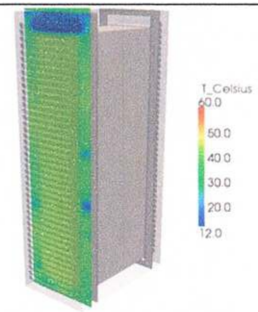



Fig.4.9 c - Field Temperatures.Geom.3, Grid 15º

## 4.3 Part 3 - Analysis of overheating in the bottoms of the racks

In this subchapter we discuss the possible causes of overheating generated at the bottom of the rack and suggests simulations to evaluate changes that bring improvements in the thermal performance of your equipment analyzed .

The abcdef Figure 4.10 shows the velocity field in the transverse plane to the rack doors . You may notice a recirculation in regions where there is communication between the front and rear of the rack . This recirculation is mainly caused by the lack of tightness between the front and rear section of the rack . As the rear section is a region with a higher realtiva pressure, the hot air coming out of servers tends to penetrate the spaces livrese eventually reach the front of the rack , where the front of cold air occurs . Because there is communication between the back and front , so there is a mixing between the hot and cold air , raising the temperature of the inlet air servers and consequently increasing the possibility of overheating. As the intake air has a higher temperature , the energy dissipated by the components entails a higher level of temperature on the output of these servers , and this warmer air eventually al front, and the cycle repeats until temperature levels reach critical values. Furthermore, the increase of the specific volume (due to increased temperature) causes the mass flow of the fans is smaller, further aggravating the problem of cooling of electronic components.

Thus, communication between the front and rear of the rack creates a pocket of warm air, which transfers the energy flow to the lower servers.

Even simplified geometric model used indicates that a reduction in the gap, ie, a seal such communicating parties could minimize this effect of overheating.

It can be seen that using louvers sloped floor type pressure increases slightly from the front side of the rack, the effect of decreasing hot air recirculation.

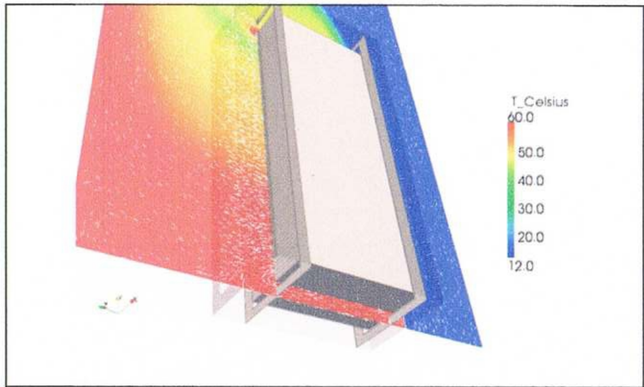


Fig.4.10a - Flow Field - Entry 0º

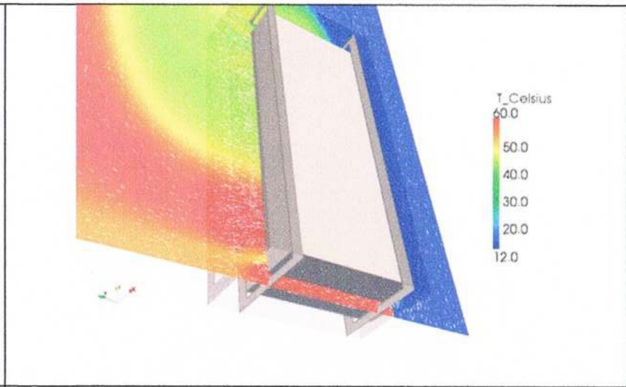


Fig.4.10b - Flow Field - Entry 0º

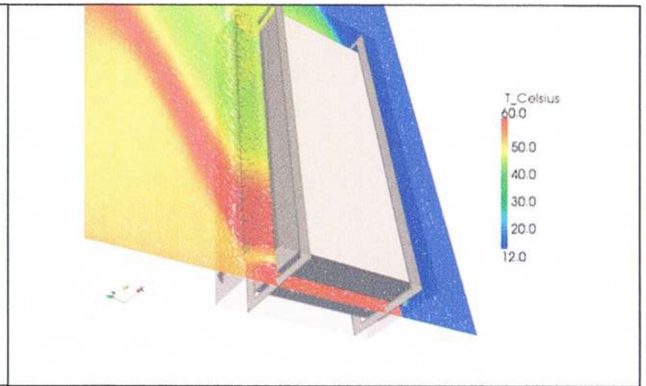


Fig.4.10c - Flow Field - Entry 0º

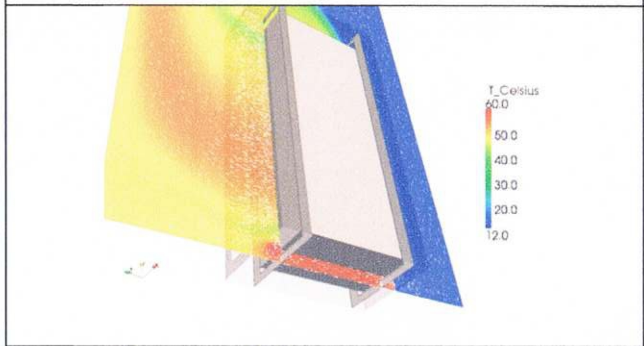


Fig.4.10d - Flow Field - Entry 15º

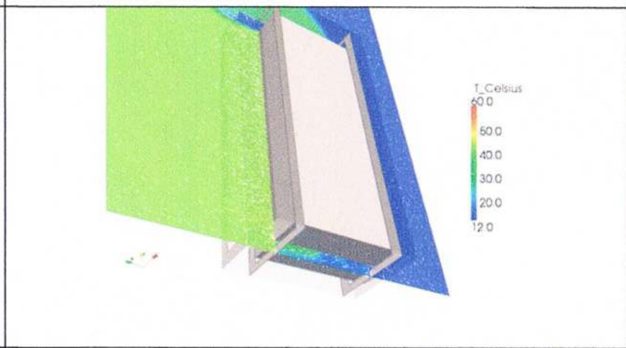


Fig.4.10 e - Flow Field - Entry 15º

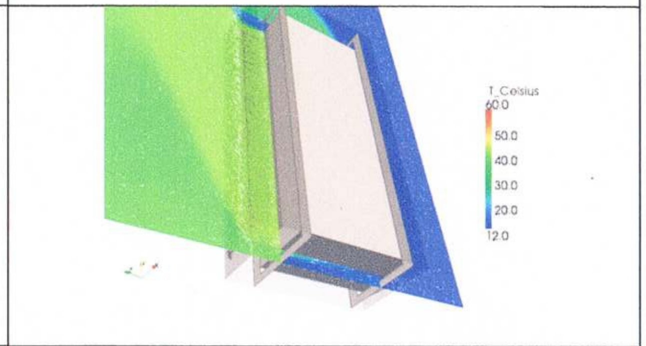


Fig.4.10 f - Flow Field - Entry 15º

## 5. Conclusion and Recommendations

The resulting analysis in Part 1 , found in Chapter 4 , it can be concluded that:

- The use of shutters in equipment promotes a redistribution analyzed in the temperature in the hot outer side of the racks.

- Although found maximum temperature in the hot zone of the machine remains the same in all three configurations analyzed in geometries composed exclusively screens, these regions with higher temperatures are greater than in configurations composed of shutters.

- The analyzes carried out in Part 1 of Chapter 4 suggest that the use of shutters are effective when placed on the front doors. To the rear doors, the simulations show that the use of shutters or screens promote similar effects in the flow.

The main conclusions in Part 2 of Chapter 4 are as follows:

- The use of sloped surface (15 degrees from normal), together with the geometry shutters generates a significant improvement in the temperature profile, providing a thermal cooling efficiency of approximately 30% compared with conventional floor whose angle of blowing with normal is null.

The conclusion and suggestions arising from the analysis described in Part 3 of Chapter 4 are:

- It was observed that lack of sealing between the front and rear portions of the rack causes recirculation zones hot air, which mixes with the cold inlet air raises its temperature, causing overheating of the lower servers. A detailed hands-analysis is suggested in order to establish such evidence and solve the problem of overheating of the lower servers.

contatos@dnsimulations.com.br
http://www.dnsimulations.com.br
CNPJ 14.004.160/0001-13

## Observation

One of the major challenges for the deployment of data centers is the correct configuration and arrangement of elements within the cold room. An unfavorable provision could lead to the creation of hot spots inside the room, leading to low efficiency of the air conditioning system, as well as possible damage to equipment due to the possibility of pockets of hot air in certain parts of the environment . DNS was developed by staff a program for automatic generation of geometry refrigerated room, with the arrangement of racks and major equipment. This program automatically proceeds to the preparation of the mesh setup and boundary conditions of CFD, and can be a useful tool in the process of optimizing the configuration of the cold room.

AluisioViais Pantaleão, Dr.
Diretor Tecnológico
aluisio.pantaleao@dnsimulations.com.br
Fone: (12) 9745-1245

Ricardo BechtFlatschart, Dr.
Diretor Presidente
ricardo.flatschart@dnsimulations.com.br
Fone: (11) 98168-8246