Analysis of Data Center to quantify the performance in terms of Thermal distribution and Air flow using Autodesk Simulation CFD Software

Raja Surya Nukala
#1 Assistant Professor,
Department of Mechanical Engineering, K L University, Andhra Pradesh, India
rajasurya.nukala@gmail.com

Manuscript History

Number: IJIRAE/RS/Vol.04/Issue06/APAE10099
Received: 26, April 2017
Final Correction: 18, May 2017
Final Accepted: 28, May 2017
Published: June 2017
Citation: Nukala, R. S. (2017), 'Analysis of Data Center to quantify the performance in terms of Thermal distribution and Air flow using Autodesk Simulation CFD Software', IJIRAE:: International Journal of Innovative Research in Advanced Engineering Volume IV (VI), 64-71.
Editor: Dr.A.Arul L.S, Chief Editor, IJIRAE, AM Publications, India

Abstract: CFD technology enables designers to predict operating characteristics, validate performance, and visualize flow and/or thermal phenomena which is difficult to capture in the real world. This is critical for AEC applications as prototyping buildings are not practical and retrofitting existing designs is expensive. The impact of CFD on AEC applications is directly related to the interpretation of simulation results; the ability to assess performance, identifies opportunities for improvement, and quantifies the impact of modification. Results visualization provides the opportunity to optimize design performance characteristics such as energy consumption, amount of contaminant entrained, thermal comfort, solar influence, hot air recirculation, and various other metrics of AEC application design performance.

Key words: Simulation, C.F.D, Data centers, Thermal distribution, Airflow

INTRODUCTION

In this paper, the performance of a data center with sixteen (16) servers will be evaluated by visualizing the flow and thermal distribution of air entering equipment in the room. Extracting data from those views will establish a metric to quantify performance and lead to further interrogation that reveals opportunities for improvement.

I. CLASSIC DESIGN CYCLE (VS) CFD UPFRONT DESIGN CYCLE

Many CFD consultants are contacted at Step 7 (Table 1.0) in a traditional design process when performance is not meeting specifications. Using CFD, they can determine the root cause of the issue, but implementing the fix can be expensive and require the space to be offline for the end client. An AEC design cycle that does not implement CFD can lead to problems during the construction and initial start-up phase. These issues result in a cycle of expensive rework efforts, delays and downtime for the end client. The Table 1.0 shows the seven stages of classic design cycle and CFD design cycle, the major difference is incorporation of CFD technology to the Engineering process at step number four (4). Due to the incorporation of CFD early in the design cycle, potential problems can be addressed before construction.

II. CFD IMPACT ON ARCHITECTURE ENGINEERING & CONSTRUCTION (AEC) APPLICATIONS

Air Management: Controlling air flow is an important aspect of AEC design and has a direct impact on performance and energy conservation. CFD results visualization is used to spot potential problems upfront and refine the location, size, and layout of system components.

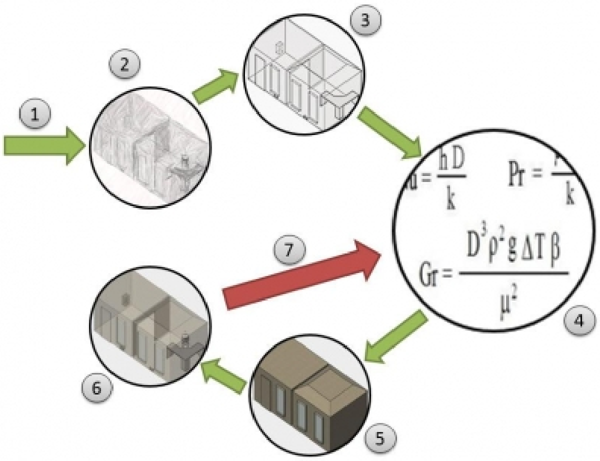


Fig.1 Traditional design process without CFD

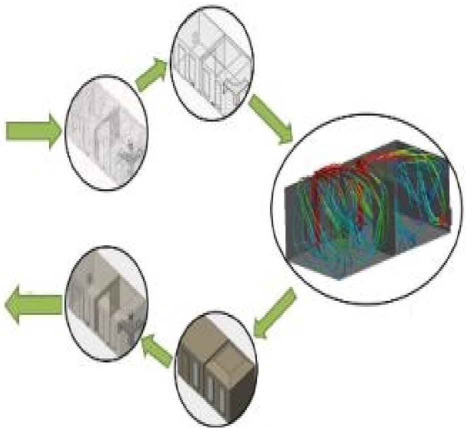


Fig.2 CFD upfront Design Process

Table.1		
Classic Design Cycle	Step	CFD Upfront Design Cycle
Start	1	Start
Concept	2	Concept
Preliminary CAD	3	Preliminary CAD
Engineering	4	**Engineering with CFD**
Final CAD	5	Final CAD
Construction/Testing	6	Construction/Testing
Fail - Rework Required	7	**Pass**

2.1) Energy Consumption

In building design, significant energy is consumed by equipment which moves and conditions air. The amount of air and conditioning required can be optimized by assessing thermal stratification and air flow paths in CFD.

2.2) Human Comfort

Human comfort is an influential consideration when developing a ventilation system. Design performance relating to air temperature, velocity, humidity, clothing, and metabolic rate can all be assessed in CFD to impact human comfort.

III. CFD WORK FLOW

Table 2.0	
Pre-processing	Post-processing
• CAD	• Results Visualization
• Materials	
• Boundary Conditions	
• Mesh Definition	• Validation
• Processing	
• Solving	

The typical CFD workflow follows a consistent and repeatable sequence of equally important steps to complete a simulation. The Simulation CFD workflow follows a consistent and repeatable sequence and it has been classified into two segments namely Pre-processing and Post-processing, the steps involved in them summarized briefly in the below Table 2.0

IV. CFD SIMULATIONS OF DATA CENTERS

The technological advances of computer hardware and software have created a need for advanced storage facilities operating at their maximum potential. Cooling a data center can consume a lot of energy. CFD can be used to model and improve the heat transfer and fluid flow in these facilities, to help optimize the overall performance and prevent equipment from exceeding maximum operating temperatures.

Objectives of simulation:

- Characterize datacenter components to accurately capture their physical behavior in Simulation CFD.
- Interpret flow and thermal simulation results to improve datacenter layout and configuration.

V. CREATING A SIMULATION-READY DATA CENTER MODEL

Data centers can be large spaces with a lot of equipment. The data center needs to be characterized well in Simulation CFD to properly represent its performance. Here the process of preparing a data center for simulation will be discussed.

5.1. Define Data Center Design Goals:

Simulation CFD can be used to help design an efficient air management system in data centers. The high energy density of a data center requires extra consideration while developing the air management system. Our objective is to avoid recirculation and conditioning hot exhaust air as soon as possible to lower operating temperatures and reduce energy costs.

5.2.Characterize Data Center Components:

Data centers all share the following primary components which need to be considered in the characterization strategy, and are discussed in detail below:

- Air Handlers,
- Server Racks,
- Diffusers and Tiles,
- Walls, Ceiling, and Floor.

The complexity of these components, combined with the large number of components, will require strategic characterization methods as detailed in the Component Characterization section.

VI. COMPONENT CHARACTERIZATION

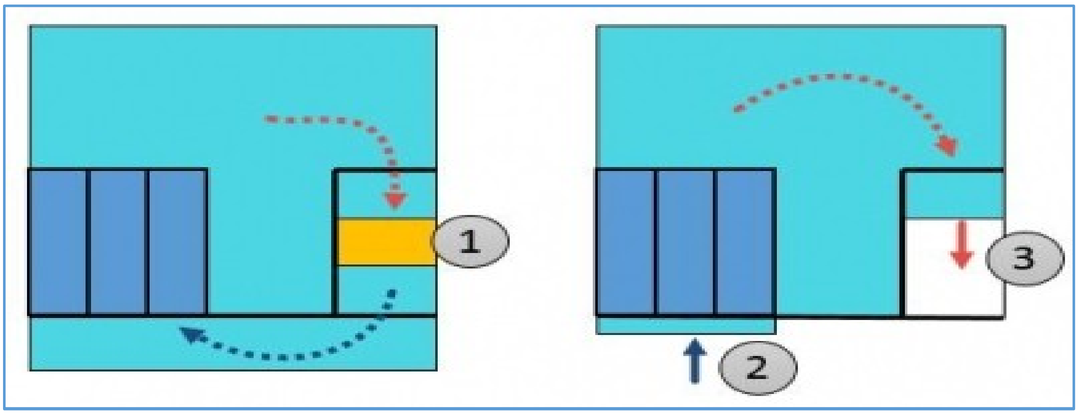


Fig.3 Closed loop Fig.4 Flow boundary

6.1 Closed loop configuration vs a Flow boundary configuration:

In this closed loop configuration Fig.3 (left), a heat exchanger device (1) is used to remove energy and circulate the air through the sub-floor and into the cold aisle inlets. A simpler alternative Fig.4 (right) uses a flow boundary condition (2) and a pressure opening (3) to avoid modeling the full sub-floor air domain.

This can be by circulating air and removing the thermal energy from the system. Can be a centralized air conditioning system or dedicated Computer Room Air Conditioner (CRAC) units. It is simulated by representing with either with heat exchanger devices or with boundary conditions. The first step in characterizing the air handler is to determine if a closed loop model will be simulated or if a simpler set of boundary conditions will be used instead. A closed loop model is more complex since it requires modeling the air space below the sub-floor or above the drop-down ceiling. When using boundary conditions (a process also known as short-circuiting), these areas do not need to be explicitly modeled, reducing overall complexity. However, the boundary conditions method will not directly account for any losses in the sub-floor space between the heat exchanger and the cold aisle inlets.

6.2 Server Racks:

From a thermal and flow standpoint, the server racks (fig.5) both circulate air and add thermal energy to the system it is simulated by representing internal fan devices with a heat generation boundary condition. Server racks contain server blades, each of which have individual fan flows and heat loads.

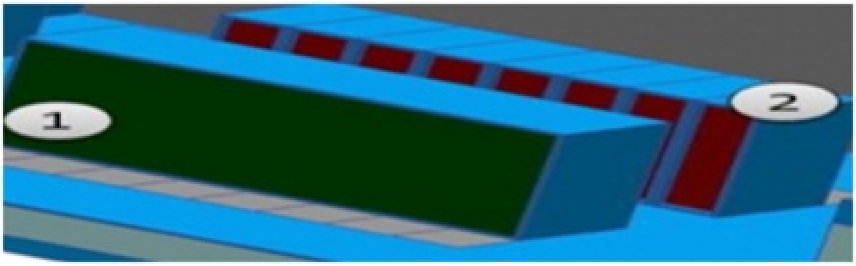


Fig.5 -(1) Entire row of server racks characterized as a single fan device.
(2) Individual racks characterized as fans.

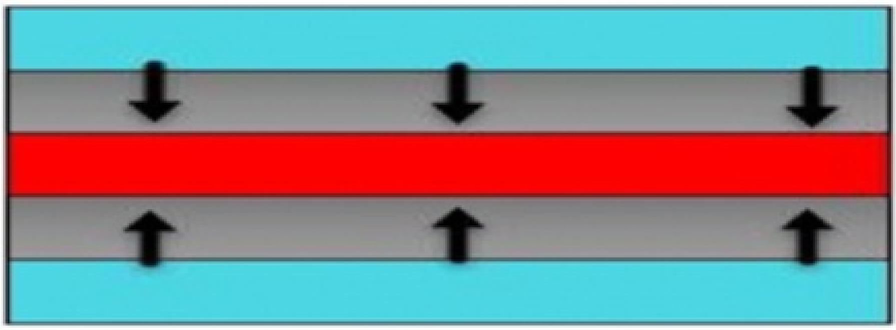


Fig.6- When simulating data centers, at least 2 internal fan devices will be needed to account for the opposite direction of flow for each row of servers. When looking down from the ceiling, the servers (grey) pull air from the cold aisles (blue) and push it into the hot aisles (red).

For simulating a data center system, the blades are all added up to determine a total flow rate and heat load for each rack. The server racks are characterized using an internal fan device which also has a heat generation boundary condition. Note that a minimum of 2 internal fan devices (fig.6) will be required since each row of servers will move air in opposite directions to create the typical alternating hot aisle/cold aisle pattern. Rows of servers can be represented with either single or multiple fans. Using a single fan reduces complexity and would be valid if all of the server racks have the same flow rate and heat load. Multiple fans would be the better choice if individual racks have varying flow rates or heat loads. Note that in actual data centers, some racks are intentionally kept empty to store wiring and other components.

6.3 Diffusers and Perforated Tiles:

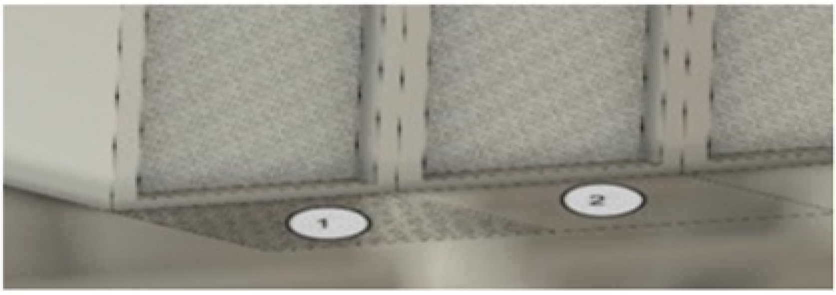


Fig.7 A detailed data center floor tile (1) next to a simpler characterization (2) using a resistance material. Note that a large data center can contain hundreds of these tiles.

Some data centers may have diffusers in the walls or ceiling to supply air from a central conditioning system. Perforated tiles are a type of diffuser used in the cold aisle supplies to evenly distribute the air across the server inlets. Diffusers should be simplified and represented with boundary conditions, especially if there are large quantities of them. Perforated tiles can be represented with a resistance material. Perforated tiles contain a lot of small holes or slots which let air pass through. The combination of small features and the overall number of tiles make them impractical to model explicitly.

To account for the pressure drop of air moving through this restriction, a resistance material can be used. Since the tiles are relatively thin compared to the overall model, the best practice is to use surface resistances for tiles to reduce mesh requirements.

6.4 Walls, Ceiling, and Floor:

These are the elements that bound and enclose the data center. The simulation of these is done by suppressing their geometry but they define the shape of the air volume. External walls can be represented with U-factor boundary conditions on the air volume. If the bounding objects (e.g., walls, ceiling, floor) of a data center are not expected to transfer thermal energy, then they can be omitted from the simulation. The primary air domains can be created directly in CAD, or if the bounding objects are modeled, they can be suppressed from the simulation.

VII. PROBLEM STATEMENT

In this paper, the performance of a data center (fig.9) with sixteen (16) servers will be evaluated by visualizing the flow and thermal distribution of air entering equipment in the room. Extracting data from those views will establish a metric to quantify performance and lead to further interrogation that reveals opportunities for improvement. The temperature of air entering computer equipment will impact its performance and reliability. Delivering a consistent amount of cool air is very important in data center design. Accounting for the path of air and its temperature as it moves through a data center enables designers to predict and optimize equipment placement (fig.8). A typical data center room (fig.9) is comprised of computer equipment in rows of racks (servers in cabinets) along with an air delivery and conditioning system.

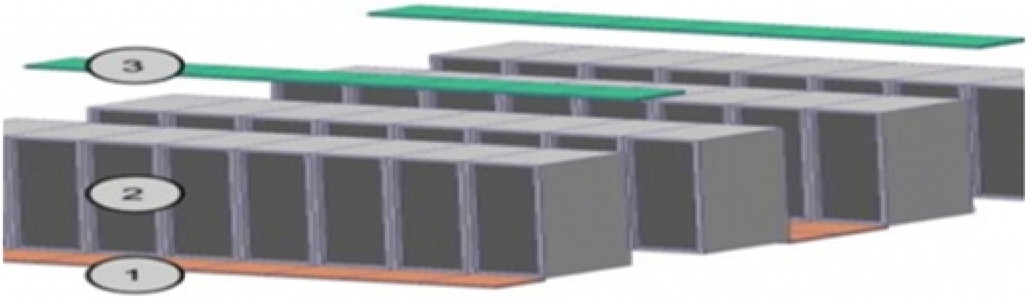


Fig.8

(1) Cool air entering from floor Tile (2) Equipment Racks (3) Exhaust air exiting from ceiling

The energy used to move and condition air is wasted when the cooler air does not move directly through equipment. The recirculation of air around the room or equipment will:

- Increase equipment air intake temperatures.
- Reduce the air delivery and conditioning systems efficiency.
- Degrade equipment performance and lifespan.

7.1) Simulation Parameters:

- Heat exchanger capacity: 15,000 CFM each
- Supply temperature of 61F
- Server racks: 800 CFM each & 10,000 Watts each

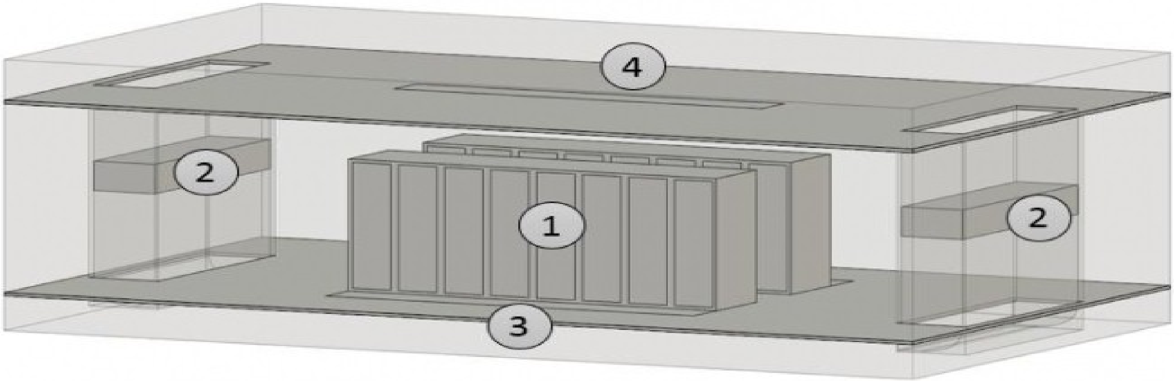


Fig.9

The CAD Model of Data Center considered for simulation is comprised of the following components: Server racks (1), CRAC units (2), Cold aisle supply tiles (3) and hot aisle exhaust tiles (4).

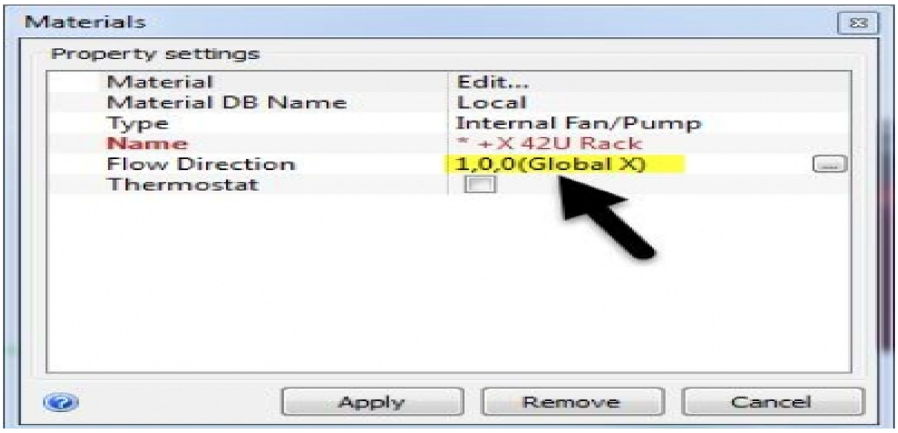
7.2) Simulation Process:

- The initial step for initiating the simulation process starts with loading the CAD file consisting the CAD model of Data Center. The Autodesk AutoCAD (2013) and Autodesk Fusion Inventor (2013) are used for creating the CAD model of Data Center.
- Later the file from Fusion Inventor is loaded into the Simulation CFD software for simulation purpose.

Table.3

Assigning Internal Fans

Save to Database	**Local**
Name	**+X 42U Rack**
Variation Method	**Constant**
Value	**800 ft3/min**
Rotational Speed	**0**



Materials

Property settings

Material	Edit...
Material DB Name	Local
Type	Internal Fan/Pump
Name	* +X 42U Rack
Flow Direction	1,0,0(Global X)
Thermostat	☐

Fig.10

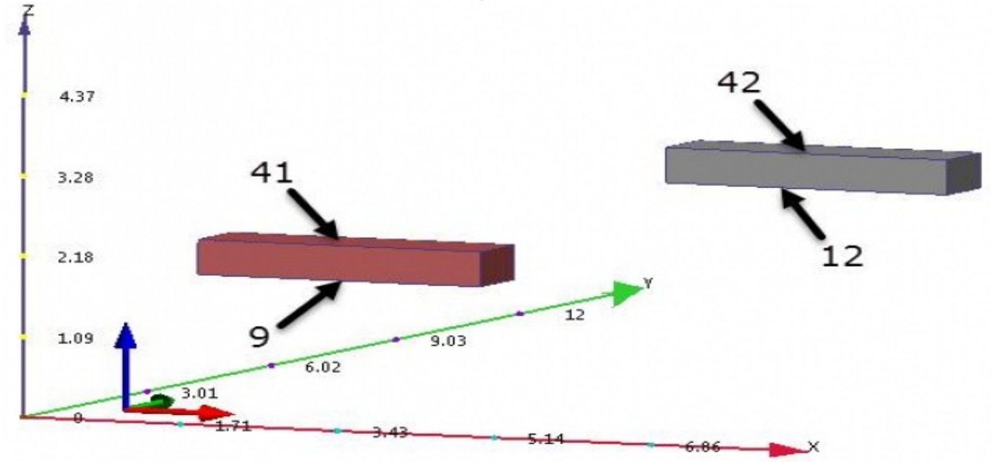


Fig.11 Surface ids of the top (41, 42) and bottom (9, 12) surfaces of the heat exchangers.
The positive Z axis points up to the ceiling of the data center

- Defining internal fan devices for the server racks: Assigning an Internal Fan device to the selected volumes with the attributes mentioned in the Table 3.0.
- After Internal Fans, we have to make certain to set the Flow Direction to 1,0,0 (+X) before clicking on the Apply button (fig.10).
- The above previous steps are repeated for the servers on the opposite side of the hot aisle but the Flow Direction is set to -1,0,0 (-X).
- Assigning Heat Exchanger devices to represent the CRAC units (fig.11): The Flow is set to Constant at 15000 ft3/min, **Variation method** of Heat Transfer is selected and **Air conditioner** mode is selected from Variation method drop down menu. The **Set Point Temperature** is assigned with a value of 61 Fahrenheit.

- Assign surface resistance regions to the floor tiles to represent the pressure drop going through the actual tiles. The 2 selected surfaces (fig.12) are assigned with surface resistance regions to represent the restriction of flow through the cold aisle tile. **FreeArea Ratio** of **Variation Method** is selected and the value is set to **0.4**. The same procedure and parameters are assigned for the ceiling tiles but the **FreeArea Ratio** value is set to **0.8**.
- Assigning a heat generation boundary condition to each server to represent the heat load from the server components (16 servers with 10,000 Watts each for a total of 160,000 Watts added to the system).

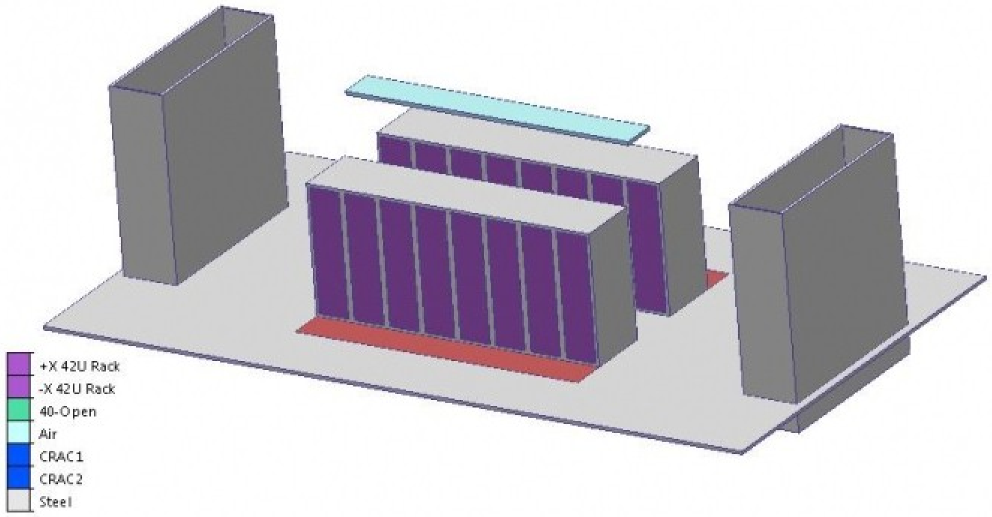


Fig.12
The 2 selected surfaces above are assigned with surface resistance regions to represent the restriction of flow through the cold aisle tile.

VIII. RESULTS

After Assigning/setting of all the parameters and values the simulation is solved for results for 100 iterations. On completion of iteration we had obtained an energy balance sheet which is presented in the Table.4

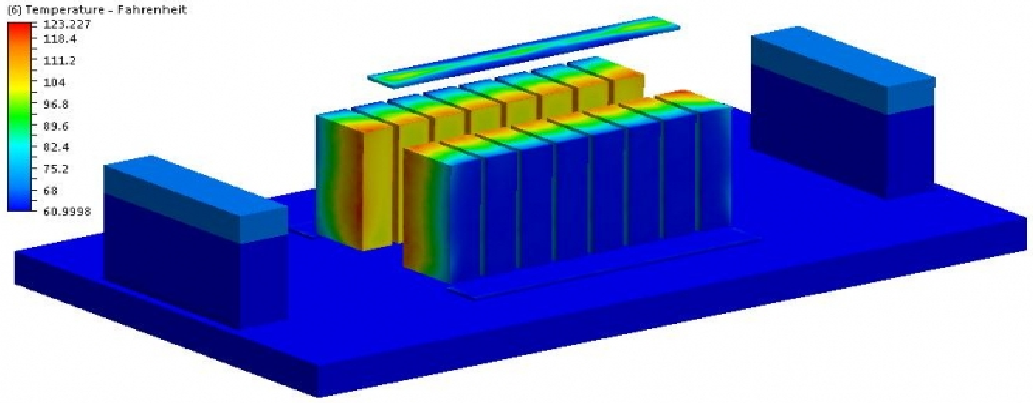


Fig.13

Table.4	
ENERGY BALANCE SHEET DATA	
Heat Transfer Due To Sources In Fluid	1.6e+005 Watts
Flow Rate for each heat exchangers	25,000 m^3/hr (15,000 cfm).

- **Checking the Global Thermal results**: The minimum temperature in the model should be 61F (16.11C), which was the set point for the CRAC units. The hot aisle should be in between the 2 rack aisles. From the Global Thermal result image (fig.13) below we can say that the highest temperatures in the model are in the hot aisle.

- **Check for recirculation in the server racks**: On zooming in on the server racks (fig.13)we had noted some areas of potential recirculation which appear at the edges of the end units. These isolated areas of higher temperatures at the server indicate that there is possible recirculation in the data center.
- The results are further interrogated, using Planes and Particle Traces (depicted below, fig.14), to confirm the extent of recirculation around the servers or in other areas of the data center.

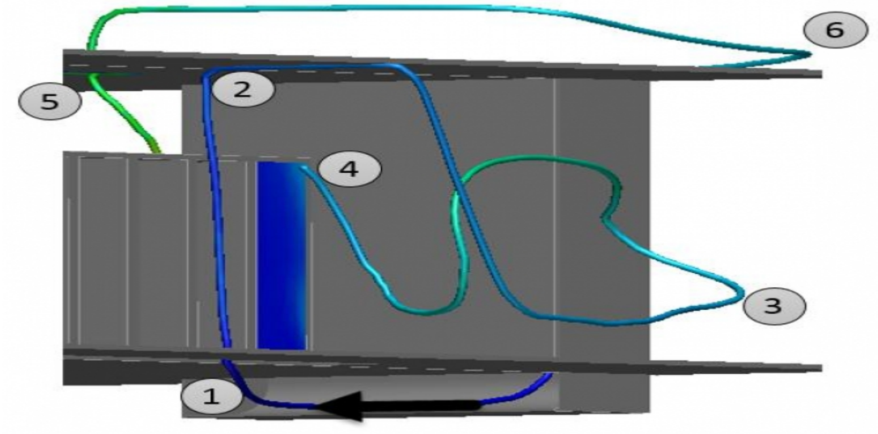


Fig.14

1. Cold air particle from the CRAC unit enters the floor tiles.
2. Air particle goes right past the server inlets to the ceiling.
3. Air particle goes over to the corner of the data center.
4. Warmer air particle is sucked into upper corner of server. This was the seed point of the particle trace which was defined by locating a Plane on the server inlet.
5. Particle exits server and goes through the ceiling tiles. Particle is pulled back into CRAC unit.

VIII. CONCLUSION

The primary focus of this paper is to ensure proper setup of a data center in terms of desirable minimum temperature (61 Fahrenheit) to be maintained and thermal distribution of air entering the room through simulation. From the Global Thermal result, Figure.13 it is clear that desirable minimum temperature is obtained and particle trace (fig.14) shows even distribution of air circulation. Even though as per the CAD design, the data center is having 16 servers, the same fundamental best practices demonstrated here could be directly applied to the largest of data centers, which may contain hundreds of servers. With larger data centers, the higher number of components will increase the probability of user input errors, making model validation a top priority. Finally, as the number of servers increase, so does the mesh size and solver run time.

RAJA SURYA NUKALA

Was born in Guntur on 8th of August 1990, he had completed Diploma in Mechanical Engineering from C.R.Polytechnic, Ganapavaram, Chilakaluripet in the year 2009, Later worked as CAD Designer from 2009-2010. Later completed his B.Tech (2010-2013)in Mechanical Engineering from Guntur Engineering College (Affiliated to J.N.T.U-Kakinada) then completed his M.Tech (2013-2015) in CAD/CAM from L.I.T.A.M(Affiliated to J.N.T.U-Kakinada), while pursuing his M.Tech he worked as a service provider to I.T.C projects division Guntur. He is also recognized as an Education Expert Premium member by the Autodesk Education community. At present he is working as Assistant Professor at K.L.University. His research is inclined towards Non-Conventional resource, Energy efficient Designs and Green Energies.