Control Strategies for Plenum Optimization in Raised Floor Data Centers

Keke Chen, Clifford C. Federspiel, David M. Auslander, Cullen E. Bash, Chandrakant D. Patel
Enterprise Software and Systems Laboratory
HP Laboratories Palo Alto
HPL-2006-47
March 7, 2006*

data center cooling, smart cooling, dynamic thermal control, energy optimization, thermal management

Contemporary data centers contain a denser aggregation of commodity computing, networking and storage hardware mounted in industry standard racks than before. As a result of high power density in data centers, mechanical engineers face more challenges in cooling system design. One of the objectives for data center cooling systems is to remove dissipated heat efficiently while maintaining the thermal management of hardware. This requires that air conditioning units provide enough cooling resources and deliver them to specific locations. Thus, each link of the cooling system has to be optimized. This paper studies the flow domain of the under-floor plenum in a raised-floor data center. Based on the analysis, flow control strategies were proposed and implemented in Computational Fluid Dynamics (CFD) simulations. The results demonstrate the possibility of optimizing airflow distribution in raised-floor data centers.

* Internal Accession Date Only
Approved for External Publication

Control Strategies for Plenum Optimization

in Raised Floor Data Centers

[1]Keke Chen, [1,4]Clifford C. Federspiel, [2]David M. Auslander, [3]Cullen E. Bash,
[3]Chandrakant D. Patel
[1]Center for the Built Environment
University of California, Berkeley, CA
[2]Dept. of Mechanical Engineering
University of California, Berkeley, CA
[3]Hewlett-Packard Laboratories, Palo Alto, CA
[4]Federspiel Controls, El Cerrito, CA
Phone 510.642.2720, keke@berkeley.edu
Phone 510.418.3392, Fax 510.236.1512, cf@federspielcontrols.com
Phone 510.642.4930, Fax 510.643.5599, dma@me.berkeley.edu
Phone 650.236.2748, Fax 650.857.7029, cullen.bash@hp.com
Phone 650.857.7140, Fax 650.857.7029, chandrakant.patel@hp.com

ABSTRACT

Contemporary data centers contain a denser aggregation of commodity computing, networking and storage hardware mounted in industry standard racks than before. As a result of high power density in data centers, mechanical engineers face more challenges in cooling system design. One of the objectives for data center cooling systems is to remove dissipated heat efficiently while maintaining the thermal management of hardware. This requires that air conditioning units provide enough cooling resources and deliver them to specific locations. Thus, each link of the cooling system has to be optimized. This paper studies the flow domain of the under-floor plenum in a raised-floor data center. Based on the analysis, flow control strategies were proposed and implemented in Computational Fluid Dynamics (CFD) simulations. The results demonstrate the possibility of optimizing airflow distribution in raised-floor data centers.

Key Words: data center cooling, smart cooling, under-floor air distribution, under-floor plenum, CFD

INTRODUCTION

Over the past decade, power dissipated per unit area of a computer chip has increased by a factor of ten; heat dissipation of a microprocessor has also gone up by an order of magnitude. At the same time, with increasing demand by consumers for commodity computing, the number of processors per rack is greater. As a result, the energy consumption and operational costs of a large data center are significant. For example, at $100/MWh, the total cost of a 100,000 ft^2 data center would be around $44 million per year to power racks and an additional $22 million per year in cooling. Mechanical engineers face a critical challenge in data center cooling.

Figure 1 depicts a typical data center with an under-floor air distribution system. The data center contains racks of computer and telecommunication equipment arranged in rows that separate supply-cool-air zones (cold aisles) and return-hot-air zones (hot aisles). Fans in each computer drives cool air from the cold aisles to the hot aisles to remove the dissipated heat. Computer Room Air Conditioning (CRAC) units deliver cool air into under-floor plenums and push it upward through perforated tiles into cold aisles.

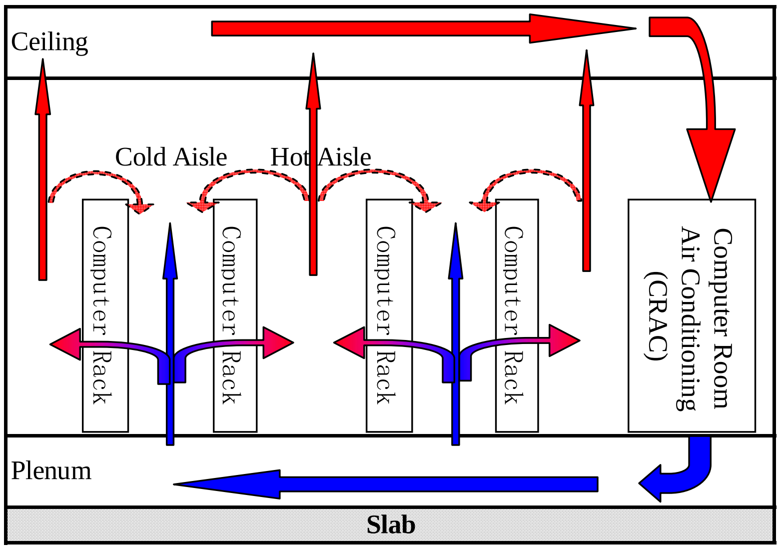


Figure 1: Schematic Section of a Data Center

To increase the energy efficiency of a data center cooling system, CRAC units have to deliver proper cooling resources for each computer system according to its heat load. However, the situation at an individual computer location is usually not ideal. Cooling-resource distribution depends on system layouts and boundary conditions. It is typical that some computers get more cooling resources than required, while others get less. This phenomenon is reflected by an uneven rack inlet temperature distribution. The inlet temperature of a computer system must be lower than the manufacturer's specification for the computer. If the inlet temperature is higher than the specification, the computer doesn't get enough cooling resource and is in danger of failing or shutting down. To satisfy all the computer equipment in a data center, CRAC units have to ensure that the maximum inlet temperature of the computer systems is within manufacturers' specifications.

Uneven airflow distribution across perforated tiles can result in air recirculation from hot aisles to cold aisles. Thus rack inlet temperature distribution is not uniform. Alternatively, the maximum inlet temperature of the computers doesn't equal their minimum inlet temperature. If the maximum inlet temperature becomes higher, the CRAC units have to decrease their temperature set-points or increase their fan speed, which increases the energy consumption of the cooling system. Therefore, we want to make airflow distribution across the perforated tiles and rack inlet temperature distribution uniform. Once the maximum inlet temperature is lower, the energy consumption will be lower. This is a significant saving in cost.

In a raised-floor cooling system, the whole under-floor plenum is used as an open duct though which cool air from different CRAC units is mixed and supplied to computer racks. Thus, airflow distribution across perforated tiles is determined by the under-floor pressure distribution. A bad airflow pattern can introduce vortexes and stagnant regions in a plenum. There may be reverse flow (flow into the plenum) if vortexes occur under the perforated tiles. The temperature above the plenum is usually higher than that under the plenum. Consequently, hot spots may come into being in the plenum. We want to create uniform under-floor pressure distribution to resist reverse flow and ensure that airflow distribution is uniform. A possible solution is to change boundary conditions, thereby breaking down big vortexes under the vent tiles.

EXPERIMENTATION

1) Prototype Plenum

The data center being studied here is located in the Research Lab of Hewlett-Packard, Palo Alto, whose plan view is shown in figure 2. The Grizzly area, Central area and Research area in the plot represent different sections of the data center. The Research area can be separated from the other two areas by a wall between the Research area and Grizzly area, the curtains between the Research area and the Central area and corresponding dampers under the plenum. There is nothing to separate the Grizzly area and Central area physically.

A commercial CFD package, CFX was selected to create a numerical model of the plenum in both the Grizzly area and Central area. This model serves as a basis to test and compare strategies for plenum flow control. Following are the procedures of setting up this model.

(1) Measure the temperatures of selected locations and the airflow from selected vent tiles after the cooling system reaches a steady state. Figure 3 and 4 show the selected locations (numbered in order) with temperature and flow rate measurements respectively. Each square represents a movable 2 feet by 2 feet floor panel. Temperature sensors are placed 2 inches below the top of the plenum.
(2) Collect the boundary conditions of the plenum, such as the size of the plenum, supply air temperature and supply air velocity.
(3) Draw the geometry of the plenum and assign boundary conditions.
(4) Start CFD simulations. Compare the simulations and experimental measurements.

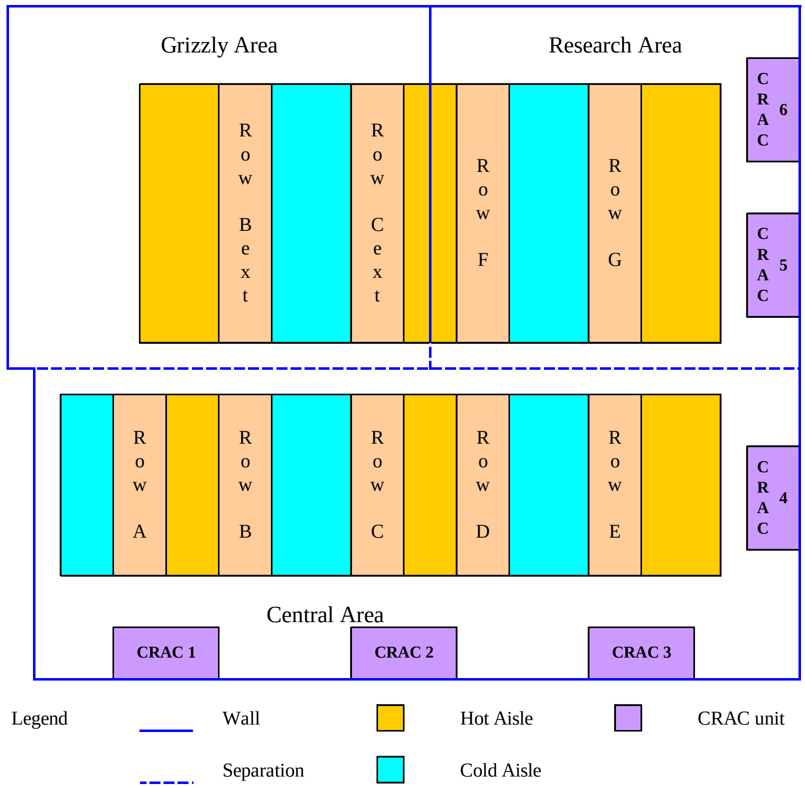


Figure 2: Plan View of the Data Center in HP Labs

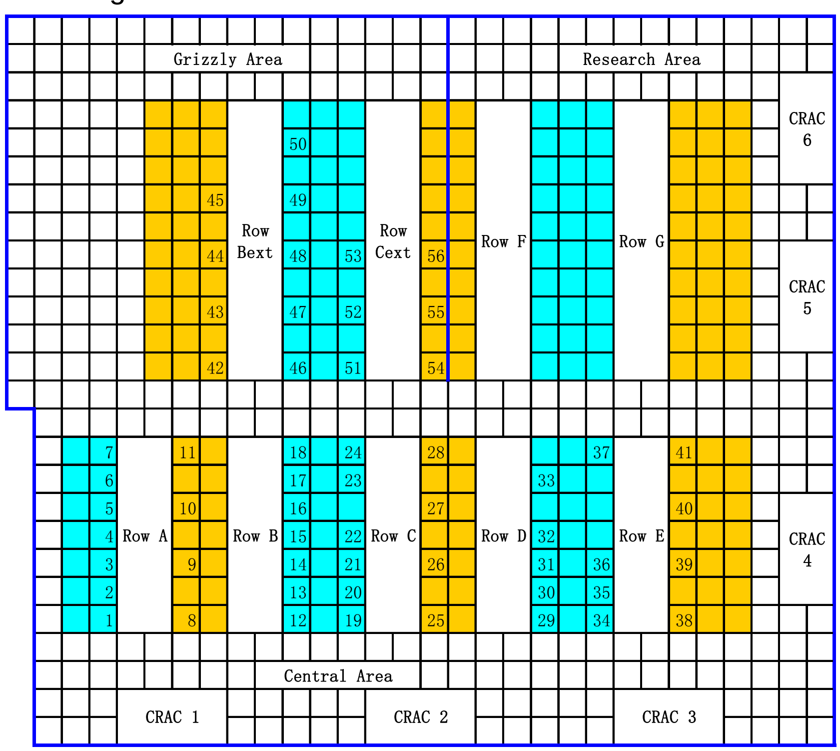


Figure 3: Locations of Temperature Sensors in the Plenum

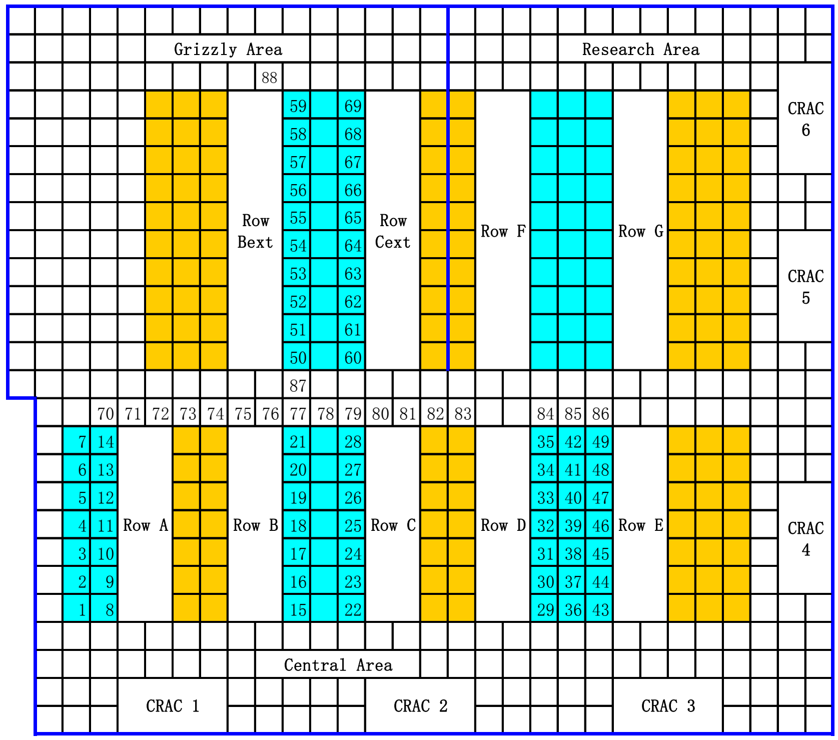


Figure 4: Locations of Measured Floor Tiles in the Data Center

2) Plenum Flow Control Strategies
The objective of plenum flow control is to make the under-floor pressure distribution more uniform. Consequently, we proposed two potential strategies: "Mixing" strategy and "Fire-Fighting" strategy (details will be explained later). The basic principle of these two strategies is to alter the supply air jets in a plenum to desired locations.

Since both strategies require new actuators and designs for cooling systems, it is safer and more efficient to test the strategies in simulations before implementing them in practice. In our prototype data center, CRAC units are designed to blow cool air down into the plenum. A modified CFD model for the plenum was set up, which moved plenum inlets from the top to the side, allowing for control of the supply air jets. The other boundary conditions were the same as that in the prototype plenum. We used the simulation result of this modified model as a base case. After implementing the strategies on the modified model, we compared the results with those of the base case.

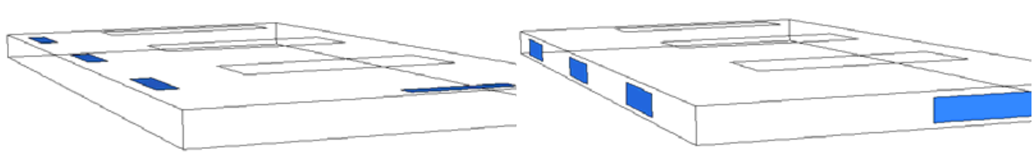


Figure 5: The Inlet Locations of Plenum (Left: Base; Right: Modified)

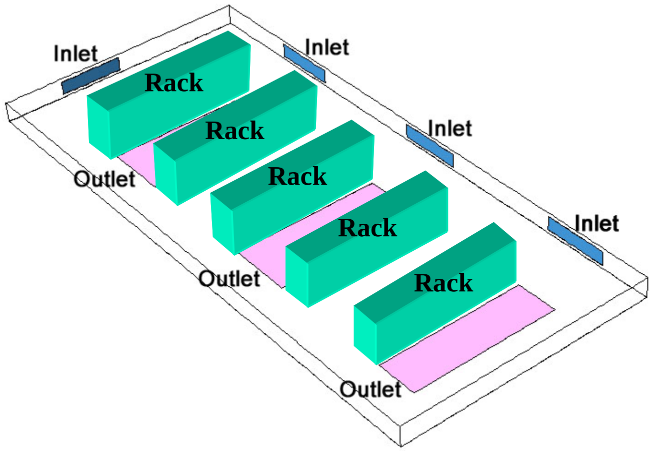


Figure 6: The Layout of Plenum and Computer Racks in the Modified Model

- "Mixing" Strategy

The so-called "mixing" strategy is based on the idea of fully mixing the air in a plenum by randomly changing the direction of supply air jets in the plenum. Non-uniform pressure distribution leads to the formation of vortexes in plenums. If we can change the focus of the air jets, we can break down these vortexes, or at least make them smaller and change their positions.

In this project, we demonstrated the effect of periodically altering the direction of the air jets at a variety of fixed angles. After a specified interval, we switched the air jets from one angle to the next, back and forth, until the end of the simulation.

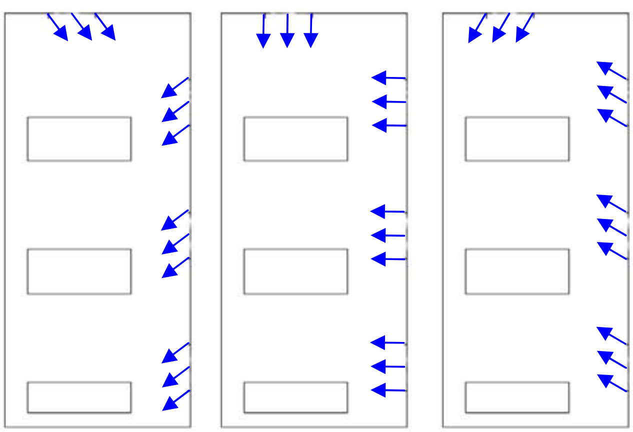


Figure 7: Schematic of "Mixing" strategy

- "Fire-Fighting" Strategy

The "Fire-Fighting" strategy also attempts to alter the direction of supply air jets. If a vortex exists under a cold aisle, its negative pressure will introduce reverse airflow and increase the local temperature. The strategy detects hot spots under the perforated tiles and directs the air jets of the nearest CRAC units to these hot spots. As boundary conditions change, vortexes should be eliminated or moved away from cold aisles, which makes the pressure under vent tiles more uniform and reduces hot spots.

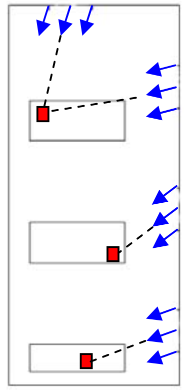


Figure 8: Schematic of “Fire-Fighting” Strategy

RESULTS/DISCUSSION

1) Base Model in CFD Simulations

All CRAC units used the same settings in the prototype plenum of the HP Data Center: temperature set-point was 17.22°C (63F); VFD was 85%. Room temperature was 18.89°C (66F). The vent tiles have two designs with different opening areas: 47% (design 1), 53% (design 2). The column “Open” in Table 1 records the opening ratio of dampers mounted on the corresponding vent tiles. “-“ represents that there was no damper mounted on this vent tile.

Label	Design	Open	Label	Design	Open	Label	Design	Open	Label	Design	Open
1	2	10%	23	1	10%	45	1	85%	67	2	20%
2	2	5%	24	1	0%	46	1	50%	68	2	100%
3	2	5%	25	2	10%	47	1	20%	69	2	0%
4	2	5%	26	2	100%	48	1	10%	70	1	85%
5	2	5%	27	2	10%	49	1	100%	71	1	10%
6	2	5%	28	1	10%	50	2	-	72	2	10%
7	2	5%	29	1	50%	51	2	-	73	1	10%
8	1	75%	30	1	100%	52	2	-	74	1	10%
9	1	100%	31	1	75%	53	2	-	75	2	10%
10	1	100%	32	1	50%	54	2	-	76	2	10%
11	1	100%	33	1	10%	55	2	-	77	1	50%
12	1	-	34	1	95%	56	2	-	78	1	10%
13	1	50%	35	1	85%	57	2	-	79	1	-
14	1	-	36	1	10%	58	2	-	80	1	-
15	1	20%	37	1	10%	59	1	-	81	2	20%
16	1	10%	38	1	10%	60	1	-	82	2	10%
17	2	10%	39	1	10%	61	1	-	83	2	100%
18	2	10%	40	1	10%	62	1	-	84	1	10%
19	2	10%	41	1	10%	63	1	-	85	1	10%
20	2	10%	42	1	10%	64	1	-	86	1	10%
21	2	100%	43	1	50%	65	1	-	87	1	-
22	1	10%	44	1	85%	66	1	-	88	1	100%

Table 1: Vent Tile Opening Ratio for the Base Model of Plenum

Figure 9 is the simulated temperature contour at a cut plane two inches below the floor. Table 2 summarizes temperatures of specific locations from the simulation (“Sim”), experiment (“Measurement”) and their difference (“Diff”).

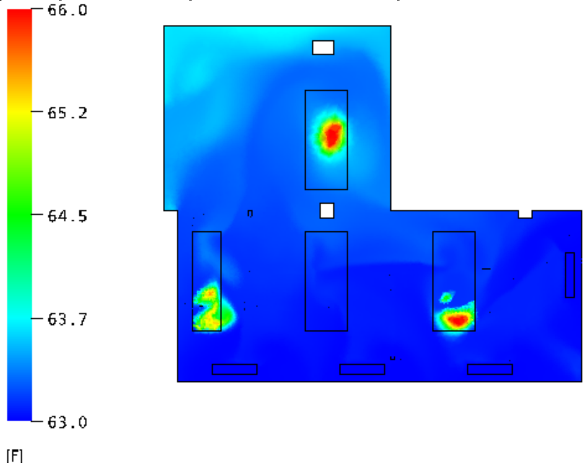


Figure 9: Plenum Temperature Distribution of Data Center from Simulation

Label	Sim	Measurement	Diff	Label	Sim	Measurement	Diff
1	64.6	63.53	1.07	29	63.82	63.38	0.44
2	65.5	63.26	2.24	30	63.21	63.56	-0.35
3	64.85	64.05	0.8	31	63.14	63.21	-0.07
4	63.45	63.84	-0.39	32	63.12	62.78	0.34
5	63.23	64.12	-0.89	33	63.14	62.7	0.44
6	63.27	63.84	-0.57	34	64.2	63.76	0.44
7	63.18	61.89	1.29	35	63.66	63.41	0.25
8	63.09	62.64	0.45	36	63.1	62.25	0.85
9	63.1	61.16	1.94	37	63.12	62.98	0.14
10	63.12	62.89	0.23	38	63.07	60.24	2.83
11	63.14	63.11	0.03	39	63.07	62.25	0.82
12	63.05	64.37	-1.32	40	63.03	63.23	-0.2
13	63.05	64.52	-1.47	41	63.1	63.94	-0.84
14	63.07	64.29	-1.22	42	63.19	63.14	0.05
15	63.12	64.44	-1.32	43	63.23	64.12	-0.89
16	63.12	64.24	-1.12	44	63.25	64.68	-1.43
17	63.16	63.65	-0.49	45	63.25	64.1	-0.85
18	63.19	63.42	-0.23	46	63.27	62.98	0.29
19	63.03	64.47	-1.44	47	63.41	62.63	0.78
20	63.07	64.22	-1.15	48	63.52	62.79	0.73
21	63.09	63.89	-0.8	49	63.3	63.26	0.04
22	63.09	63.61	-0.52	50	63.27	63.42	-0.15
23	63.16	62.12	1.04	51	63.36	63.89	-0.53
24	63.21	61.73	1.48	52	63.7	63.51	0.19
25	63.07	62.4	0.67	53	64.83	63.94	0.89
26	63.05	62.5	0.55	54	63.32	65.56	-2.24
27	63.05	63.23	-0.18	55	63.32	65.05	-1.73
28	63.18	62.68	0.5	56	63.28	65.05	-1.77

Table 2: Comparison of Temperature Distributions (Unit: Degree)

We used Precon thermistors (model ST-R3R) to measure the temperature distribution in the plenum. Their accuracy is ±0.3°C, and A/D resolution is about 0.2°C. So the uncertainty of temperature measurements is about 0.5°C (0.9 F). If the difference between simulation and experiment is not larger than 1 F, we think they are consistent; otherwise they are inconsistent. Table 3 indicates that about two thirds of the simulation results match the measurements.

Category	Number	Ratio
Consistent	38	68%
Inconsistent	18	32%
Total	56	

Table 3: Temperature Simulation Accuracy (compared with experiment)

Figure 10 depicts the temperature simulation in each cell and indicates whether the results are consistent with experiment.

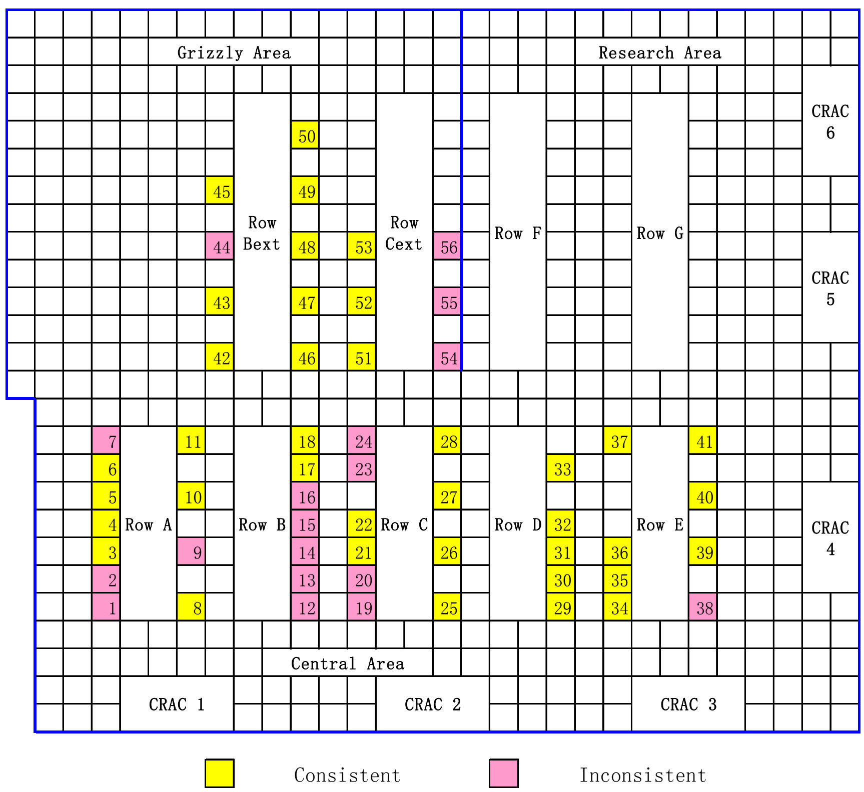


Figure 10: Locations of Consistent and Inconsistent Cells in Temperature

Table 4 lists the airflow of the selected vent tiles from the simulation ("Sim") and experiment ("Measure"), their difference ("Diff") and the ratio of this difference and measurement ("Ratio"). The airflow of each vent tile was measured by flow hoods (backpressure compensated air balanced system model CFM-88), whose accuracy is about ±3% of readings ±7 CFM from 100 to 2000 CFM. Turbulence also results in some variations in airflow measurements, thus we estimate the uncertainty to be ±10% of readings. In addition, we couldn't set boundary conditions perfectly in the CFD simulation. For example, we judged the opening ratio for each vent tile by eye, which could incorporate large bias.

Label	Sim	Measure	Diff	Ratio	Label	Sim	Measure	Diff	Ratio
1	173	222	-49	-22%	45	1025	1049	-24	-2%
2	130	137	-7	-5%	46	754	492	262	53%
3	79	152	-73	-48%	47	276	281	-5	-2%
4	135	155	-20	-13%	48	225	92	133	145%
5	143	131	12	9%	49	1314	1335	-21	-2%
6	148	148	0	0%	50	1198	1086	112	10%
7	152	142	10	7%	51	1268	1032	236	23%
8	704	666	38	6%	52	1128	1142	-14	-1%
9	965	1014	-49	-5%	53	847	1109	-262	-24%
10	973	988	-15	-2%	54	551	1053	-502	-48%
11	998	1023	-25	-2%	55	371	1100	-729	-66%
12	1691	2048	-357	-17%	56	594	1119	-525	-47%
13	621	427	194	45%	57	1072	1174	-102	-9%
14	1782	1900	-118	-6%	58	1394	1178	216	18%
15	253	248	5	2%	59	1614	1275	339	27%
16	194	116	78	67%	60	1035	1205	-170	-14%
17	191	154	37	24%	61	1205	1597	-392	-25%
18	175	183	-8	-4%	62	962	1298	-336	-26%
19	159	123	36	29%	63	403	1162	-759	-65%
20	155	126	29	23%	64	-129	1152	-1281	-111%
21	876	798	78	10%	65	-84	493	-577	-117%
22	220	77	143	186%	66	620	875	-255	-29%
23	203	119	84	71%	67	132	108	24	22%
24	169	103	66	64%	68	888	604	284	47%
25	177	161	16	10%	69	130	82	48	59%
26	921	948	-27	-3%	70	905	577	328	57%
27	177	147	30	20%	71	170	76	94	124%
28	163	139	24	17%	72	167	136	31	23%
29	778	809	-31	-4%	73	168	108	60	56%
30	1448	1396	52	4%	74	169	180	-11	-6%
31	983	1214	-231	-19%	75	79	98	-19	-19%
32	790	564	226	40%	76	145	99	46	46%
33	244	166	78	47%	77	484	266	218	82%
34	1213	1255	-42	-3%	78	184	102	82	80%
35	1073	783	290	37%	79	1827	1668	159	10%
36	242	119	123	103%	80	1361	1498	-137	-9%
37	238	127	111	87%	81	190	162	28	17%
38	250	119	131	110%	82	196	143	53	37%
39	248	216	32	15%	83	1062	986	76	8%
40	235	141	94	67%	84	248	154	94	61%
41	228	111	117	105%	85	244	168	76	45%
42	249	215	34	16%	86	241	160	81	51%
43	745	564	181	32%	87	1007	1293	-286	-22%

44	993	1205	-212	-18%	88	660	578	82	14%

Table 4: Comparison of Airflow Distributions (Unit: CFM)

Thus, we evaluate the accuracy of the flow simulation using the following criteria:

i. If the ratio between difference and measurement is not larger than 10%, the simulation is consistent with the experiment.
ii. If the ratio is between 10% and 50%, but the measurement is lower than 300 CFM, the simulation is consistent with the experiment, where the airflow is small.

Any other situations different from those above are inconsistent cases. Only about half of the airflow simulations are consistent with measurements.

Category	Number	Ratio
Consistent	46	52%
Inconsistent	42	48%
Total	88	

Table 5: Goodness of Flow Rate Simulations

Figure 11 shows that most of the inconsistent locations are in the Grizzly area. The simulation shows a suspected large vortex under the plenum in the Grizzly area (Figure 12). Any inaccurate boundary setting can result in this vortex. Therefore we should treat the CFD simulation with care. No matter how complicated the model is we can never reach a perfect model which is able to replicate every detail of the real plenum. Thus, the simulation results can never be exactly the same as the experimental measurements. However, as long as we obtain reasonable outputs from the CFD model, we can accept them, since the simulation is used to demonstrate plenum flow control strategies instead of accurately predicting the experiments. Most of our simulation results are consistent with the experimental measurements. We only modeled the central area to demonstrate the two strategies for simplicity.

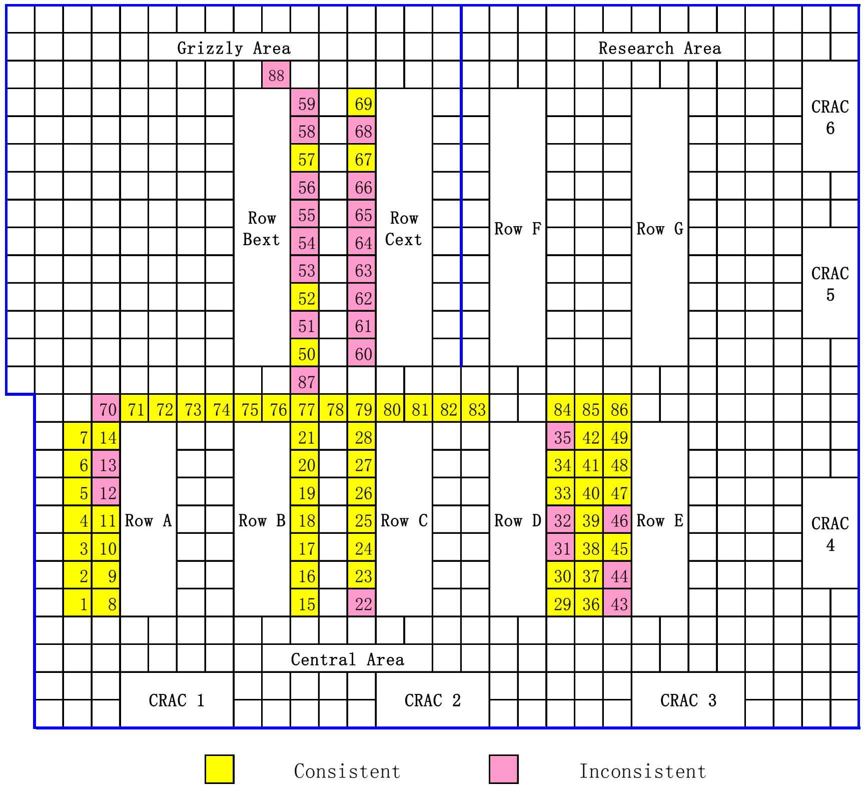


Figure 11: Locations of Consistent and Inconsistent Cells in Flow Rate

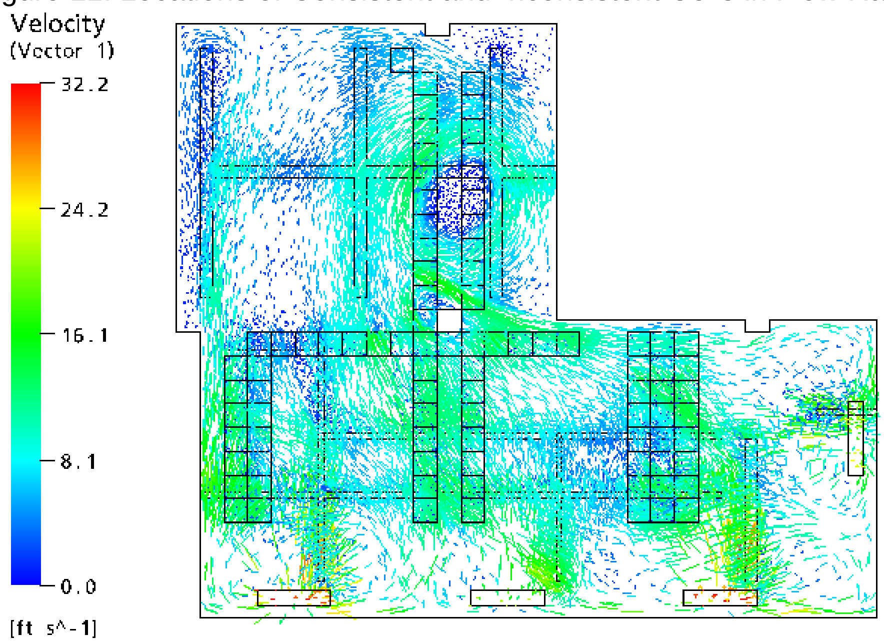


Figure 12: Flow Domain of Plenum from Simulation

Data centers have more intensive heat loads than commercial office buildings. Due to the characteristics of data centers, supply air velocity in under-floor plenums is much higher. In this simulation, heat transfer to and from the slab is not significant, and the horizontal temperature gradients mainly derive from reverse airflow. The control strategies are to eliminate or move vortexes under the vent tiles so that cool air is blown out of plenums rather than being sucked back into the plenums.

2) Plenum-Flow Control Strategies

In this simulation, the temperature set-point was 8°C (46.4 F), room temperature was 25°C (77 F).

(1) "Mixing" strategy

Figure 13 shows the simulated flow domain of the base case, where the system is at a steady state. The vector of velocity is colored with regard to its temperature. The first plot of Figure 14 is the temperature distribution of the base case. Some vortexes exist under the vent tiles.

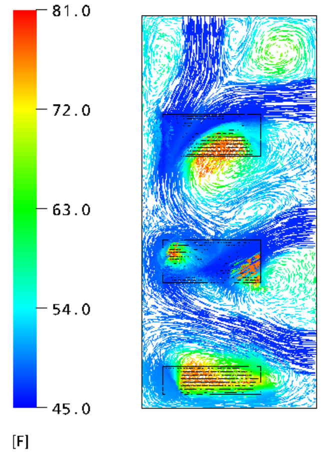
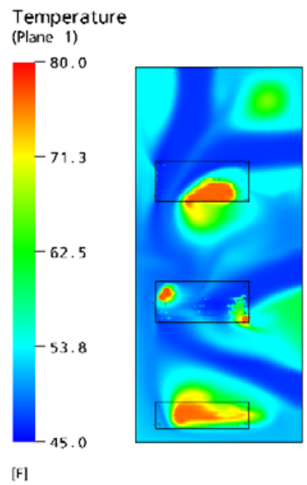


Figure 13: Flow Domain of Base Case



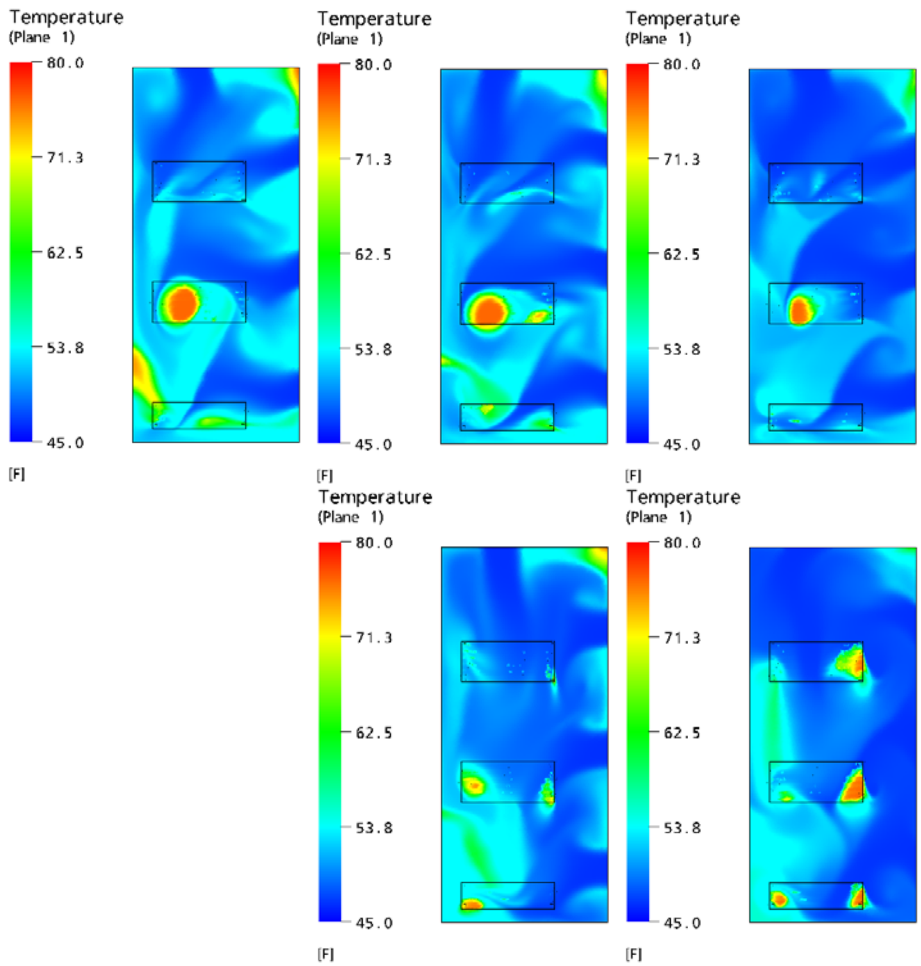


Figure 14: Comparison of Base Case and Two "Mixing" Strategies

Starting with this status, two "Mixing" strategies with different operation frequencies were implemented. Both strategies altered the direction of the supply air jets among three fixed angles: as shown in Figure 7, firstly the angles between the air jets and the walls were set to 45 degrees, then 90 degrees, and 135 degrees after that. And then the angles were changed back in reverse order. Thus, the sequence of the angles should be: 45, 90, 135, 90, 45, 90, •••. The only difference between the two "mixing" strategies here was the operating frequency.

- Strategy I: alter the direction every second;
- Strategy II: alter the direction every 3 seconds.

In Figure 14, the plots in the second row show the temperature distribution at the end of 20 seconds, 26 seconds and 40 seconds of strategy I. Activate strategy II after strategy I has run for 20 seconds. The plots in the third row show the temperature distribution at the end of 26 seconds and 40 seconds of strategy II, respectively. The location and size of vortexes were changed after the "Mixing" strategies started. As a check, Figure 15 shows the flow domains at the end of 40 seconds of both strategies. The vortexes were moved away from the cold aisles.

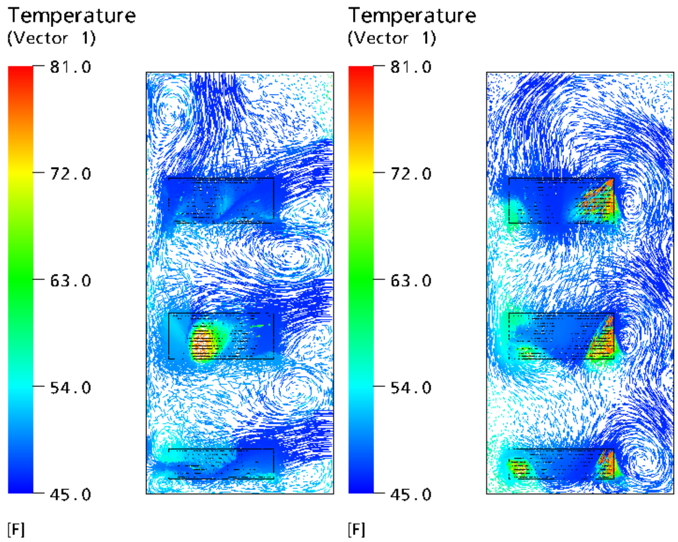


Figure 15: Flow Domain of the "Mixing" Strategies (Left: I; Right: II)

Since we didn't randomize the directions of the air jets and the operating frequency, the flow domain expressed a quite strong pattern. What's more, strategy II could successfully eliminate the vortexes while strategy I couldn't. This is reasonable since the flow domain needs some time to respond to the variations of the air jets. Thus, we can select appropriate operating frequencies and randomize the directions of the air jets in order to maximize the effect of the "Mixing" strategy.

(2) "Fire-Fighting" Strategy

In this simulation, we changed the supply-air jets every 3 seconds. To compare with the "Mixing" strategy, the initial condition of the simulation used the results of the "mixing" strategy after 20 seconds of real-time operation. In Figure 16, the first plot indicates the initial temperature distribution, and the plots in the second row represent the temperature distributions after the "Fire-Fighting" strategy has been running for 20 seconds, 40 seconds and 70 seconds, and the plots in the third row are the corresponding flow domains.

The "Fire-Fighting" strategy was more effective in destroying vortexes than the "Mixing" strategy. We aimed the air jets at the hot spots. At the end of the simulation it effectively moved the hot spots away from vent tiles. In data centers, the major reason for temperature variation under the vent tiles is the existence of vortexes and reverse flow. Thus, the "Fire-Fighting" strategy successfully eliminated the vortexes and made the pressure more uniform. Figure 17 indicates the pressure distribution at the three states in Figure 16.

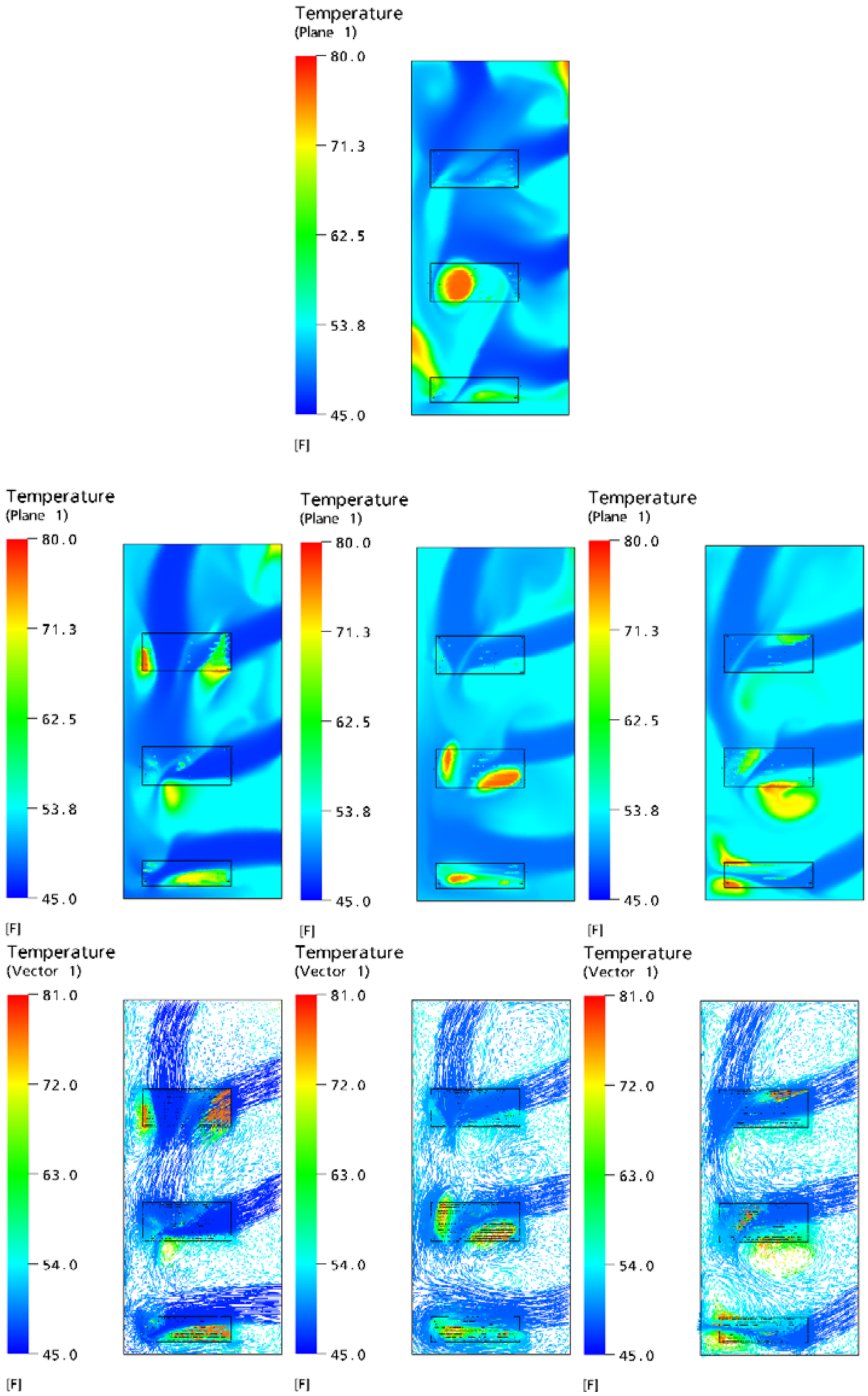


Figure 16: Temperature and Flow Domain of “Fire-Fighting” Strategy

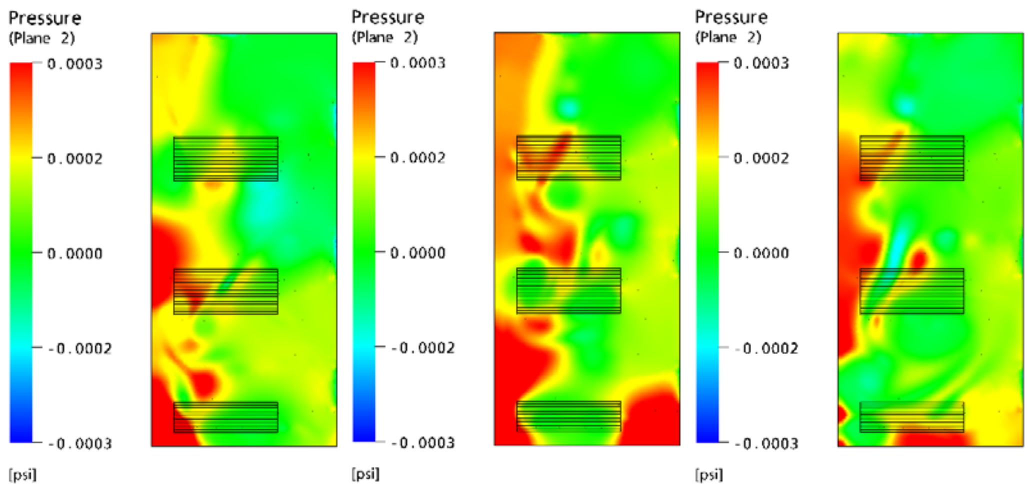


Figure 17: Pressure Distributions of "Fire-Fighting" Strategy

The two strategies are based on a simplified model. In practice they may not work very well since there are many more obstacles in real plenums, which can produce some effects on the strategies. Furthermore, the strategies require new plenum designs and extra equipment to alter the directions of the air jets. The "Fire-Fighting" strategy even requires sensor networks in plenums to detect temperature or pressure distribution. However, the idea of "stirring" can invoke some effective methods of creating a uniform under-floor pressure distribution, i.e. a simple barrier to separate airflow may be effective in eliminating vortexes; or a good design of cable trays may help to make pressure distribution more uniform.

CONCLUSION

The issue of energy efficiency becomes more and more important for contemporary data centers. Uniform under-floor pressure distribution can alleviate hot air recirculation in data centers, thus increasing the energy efficiency of cooling systems. We have analyzed the flow domain in the under-floor plenum of the HP Data Center through CFD simulations, and designed control strategies to make the under-floor pressure distribution more uniform.

- Horizontal temperature gradients are not significant in under-floor plenums of data centers. Their formation mainly results from non-uniform under-floor pressure distribution.

- Both the "Mixing" strategy and the "Fire-Fighting" strategy can be effective to remove hot spots in the plenum with appropriate randomization procedures.

- Both strategies are based on a simplified model. In practice they may not work very well since there are many more obstacles in real plenums.

- Both strategies require installing extra equipment and/or sensor networks in plenums. Since they introduce active control strategies into plenums, more tests are needed to check their reliability.

- The idea of “stirring” can lead to some effective strategies for pressurizing the under-floor plenums, i.e. a simple barrier to separate airflow may be effective in eliminating vortexes; or a good design of cable trays may help to make pressure distribution more uniform.

Although the ideas of “Mixing” and “Fire-Fighting” are innovative, their reliability, feasibility and costs are unknown. We need to compare their advantages and disadvantages before implementing these strategies. The invoked methods may be easier to test in a given data center. For example, if we detect a vortex under a cold aisle, we can try to put a simple barrier in the plenum to eliminate the vortex.

ACKNOWLEDGMENTS

Many thanks to Professor Edwards Arens and Graduate Student Timothy Boucher of UC Berkeley for invaluable advice on these strategies. Finally, thanks to Kathe Gust for reviewing the report.

REFERENCES

Bash, C. E., Patel, C. D., Sharma, R. K., 2003, “Efficient Thermal Management of Data Centers – Immediate and Long-Term Research Needs”, HVAC&R Research, Vol.9, No.2, Apr 2003

Bauman, F., Shirai, R., Webster, T., Zagreus, L., Shi, M., 2003, “Field Study of Capitol Area East End Complex”, Industry Advisory Board meeting of the Center for the Built Environment, University of California, Berkeley, April 2003

Friedrich, R., Patel, C. D., 2002, “Towards Planetary Scale Computing – Technical Challenges for Next Generation Internet Computing”, THERMES 2002, Santa Fe, New Mexico

Hwang, Y., Radermacher, R., Spinazzola, S., Menachery, Z., “Performance Measurements of a Forced Convection Air-Cooled Rack”, Inter Society Conference on Thermal Phenomena, 2004

Kang, S., Schmidt, R. R., Kelkar, K. M., Radmehr, A., Patankar, S., “A methodology for the design of perforated tiles in raised floor data center using computational flow analysis”, IEEE-CPMT Journal, Vol. 24, No. 2, June 2001

Schmidt, R., Karki, K., Patankar, S., “Raised-Floor Data Center: Perforated Tile Flow Rates for Various Tile Layouts”, Inter Society Conference on Thermal Phenomena, 2004

Schmidt, R. R., Shaukatullah, H., “Computer and Telecommunications Equipment Room Cooling: A Review of Literature”, Inter Society Conference on Thermal Phenomena, 2002

Wang, D, “A passive solution to a difficult data center environmental problem”, Inter Society Conference on Thermal Phenomena, 2004

http://www-waterloo.ansys.com/cfx/#