
Computer Simulation Optimizes Displacement Ventilation System in a School Auditorium

By Hee-Jin Park, Ph.D., P.E.
hjpark@flonomix.com
Flonomix, Inc.

Computer simulation played a key role in the optimization of a displacement ventilation system in a school auditorium by helping to reconfigure the locations of the diffusers and the exhausts in order to eliminate unnecessary air flow. Displacement ventilation systems, which locate the air supply on the floor instead of the ceiling, are gaining popularity in North America because they are more efficient and require less energy to maintain a given level of comfort. However, displacement ventilation systems are generally more difficult to design due to several inherent issues like the need to avoid temperature gradients and drafts with the air supply located near the people.

In a recent application, these challenges were addressed by simulating a school auditorium using the computational fluid dynamics (CFD) technique to evaluate airflow velocity, pressure, and temperature. The simulation showed the existence of a flow pattern that had both hot, cold spots as well as a stagnant area in the auditorium. The diagnostic information provided by the simulation results made it easy to improve the flow pattern by changing the location of some of the diffusers and exhausts.

Displacement ventilation - a brief overview

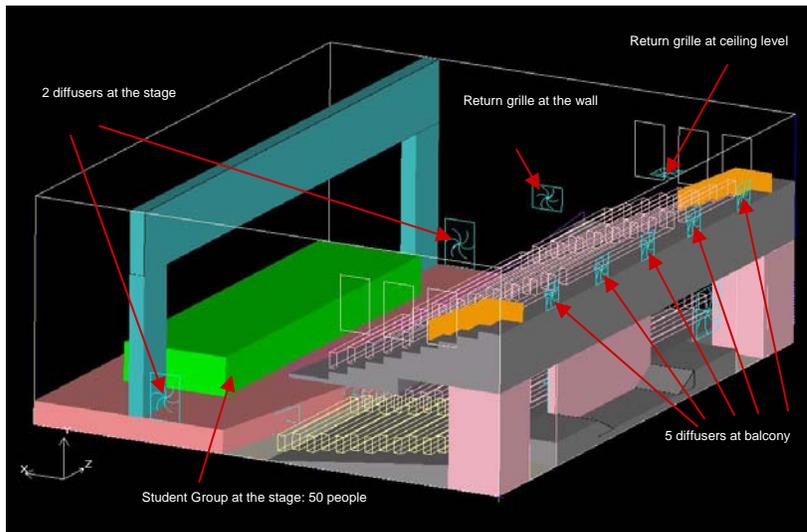
Displacement ventilation is a unique concept for ventilating buildings and supplying conditioned air. It uses the natural buoyancy of warm air to provide improved ventilation and comfort. First developed for industrial buildings, displacement ventilation now enjoys an increasing market share in many applications throughout the world. Although relatively new to the United States, displacement ventilation has been in use in Europe, since the 1970s.

In a conventional HVAC system, air is supplied at a relatively high velocity towards the ceiling at a temperature about 20°F below the design temperature. The supply air mixes with the room air to provide a nearly uniform temperature throughout the space. The mixing slows the rate at which the room air recirculates, resulting in relatively low ventilation efficiency.

In a displacement ventilation system, on the other hand, supply air is introduced to the space at or near the floor level at a temperature only slightly below the design temperature at a low velocity. The cool, clean air spreads and forms a pool of conditioned air over the floor in the occupied space, displacing the warmer room air. When the cool air meets a heat source, the temperature difference creates a buoyant force and a convection plume is generated. This plume acts as a channel through which warmed and dirty air moves upward to a ceiling area where it exits through the exhaust. Due to entrainment of the surrounding air, the volumetric flow of the plume gets larger as the plume rises. When the flow rate of the plume is equal to that of the supply air, thermal and containment zones form a level or area that distinguish the upper (warm and polluted) and lower (cool and clean) levels of air. This level or area is termed as a stratification level through which there is no flow exchange between lower and upper levels. The warm and contaminated air and contaminants are then exhausted from the space near the ceiling.

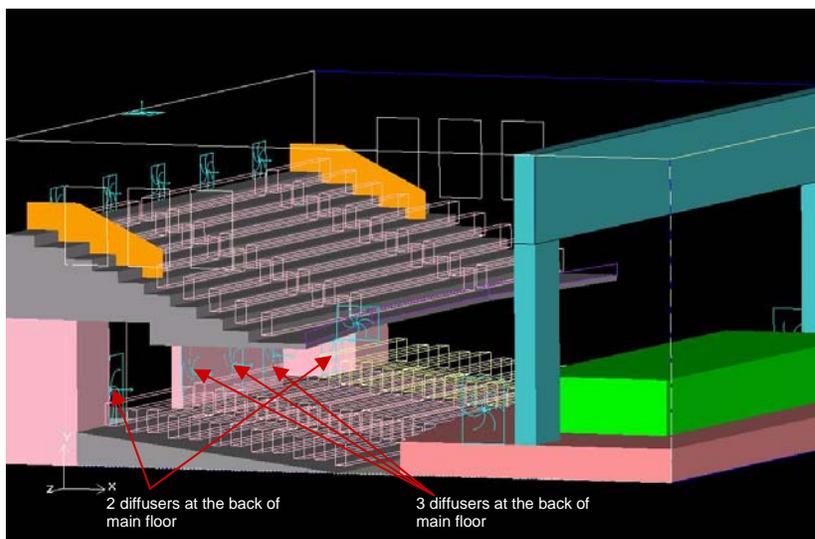
This stratification is one of the greatest advantages of thermal displacement ventilation systems over conventional mixing type ventilation systems. Cool and clean air is where people need to breathe it and hot, dirty air is at the ceiling being exhausted. In addition to the fact that the displacement ventilation approach improves indoor air quality in the lower level by separating contaminated air from clean air through the stratification, many investigators have noted the advantages of displacement ventilation in lower energy costs. It was recently reported by Park *et al.* (H.J. Park and D. Holland, " Effect of Location of Heat Source on Displacement Ventilation", *Building & Environment*, Vol. 36, pp.883-889, 2001) that when a location of heat source is higher, heat gain from a heat source to the lower level decreases significantly, which results in a reduction of the cooling load in the lower region.

Geometry description



Room geometry and original ventilation system design (1)

The school auditorium shown above is designed to seat a total of 680 people. The main floor has 12 rows of seats designed for 460 people while the balcony has 9 rows that seat 220 persons. The stage is designed to accommodate an additional 50 persons. The original design of the HVAC system had two 1600 cfm diffusers and three 1200 cfm diffusers at the back of the main floor, four 400 cfm diffusers at the front of the main floor, two 600 cfm diffusers on the stage, and five 1000 cfm diffusers at the back of the balcony. There were two 3400 cfm returns on the balcony ceiling, one 2800 cfm wall exhaust, and one 5000 cfm exhaust at the balcony.



Room geometry and original ventilation system design (2)

Challenge of design optimization with computational fluid dynamics

The challenge of designing a ventilation system for a complicated space such as this is that the performance is highly dependent upon a number of variables, such as the sizing and placement of supply diffusers and return grilles, location of heat sources etc. It is impractical to measure the flow related parameters to any significant degree of accuracy. Hence, the best that engineers can do using conventional methods is to make a rough hand calculation or educated guess as to which configuration will work best. The accuracy of hand calculations is reduced by several factors. First, these calculations do not account for the geometry details. Second, they do not account for the localized effects of the supply diffusers and exhausts.

CFD can dramatically improve this process by predicting airflow, pressure, temperature, and contaminant concentrations throughout a region with a high level of accuracy. CFD is a very potent, non-intrusive virtual modeling technique that uses numerical methods to solve the fundamental nonlinear partial differential equations that describe fluid flow (the Navier-Stokes and allied equations) for predefined geometries, boundary conditions, process flow physics, and chemistry. A key advantage of CFD is that engineers can evaluate the performance of a wide range of exhaust system configurations on the computer without the time, expense, and disruption required to make actual changes on site.

In this application, Flonomix engineers created an unstructured grid using hexahedral cells with 594,702 elements. They used a standard $k-\epsilon$ turbulence model and ideal gas law for a density calculation. The calculation converged (reached a preset minimum error) on a dual 2.4 GHz Xeon processor personal computer with 2 GB of RAM.

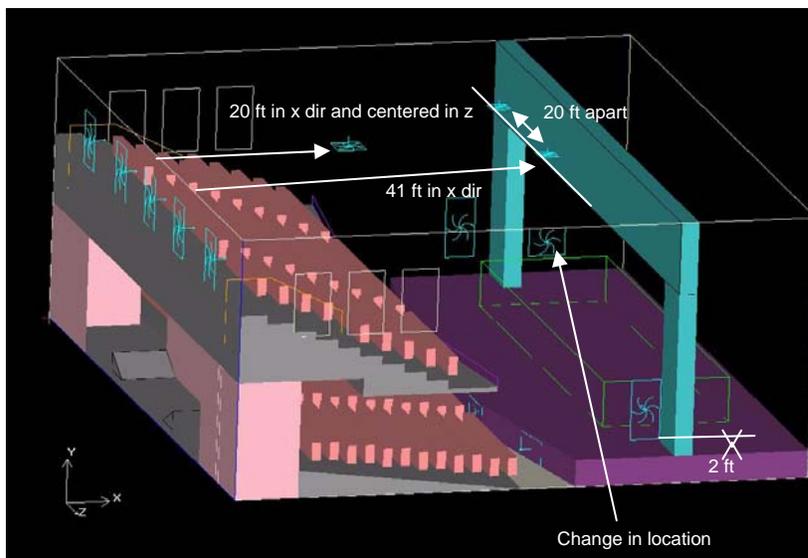
Evaluating original case

The CFD simulation of the original design showed problems with recirculation, which may destroy a stratification level that displacement ventilation system designers are aiming at for indoor quality. Mainly that's because the exhaust grille on the ceiling level is located too close to the back wall of the balcony. Another problem occurs in the air on stage. Because there is no exit for the warm and contaminated air above the stage, the air needs to travel far to exit the space which results in stagnant area over the stage. The flow velocity distribution showed that the plume of air on the

balcony hit the ceiling and was redirected towards the floor, instead of leaving the room through the exhausts. Tracking the path of a particle through the auditorium showed that it moved on a very convoluted path, moving up and down and around the auditorium.

Optimizing the design

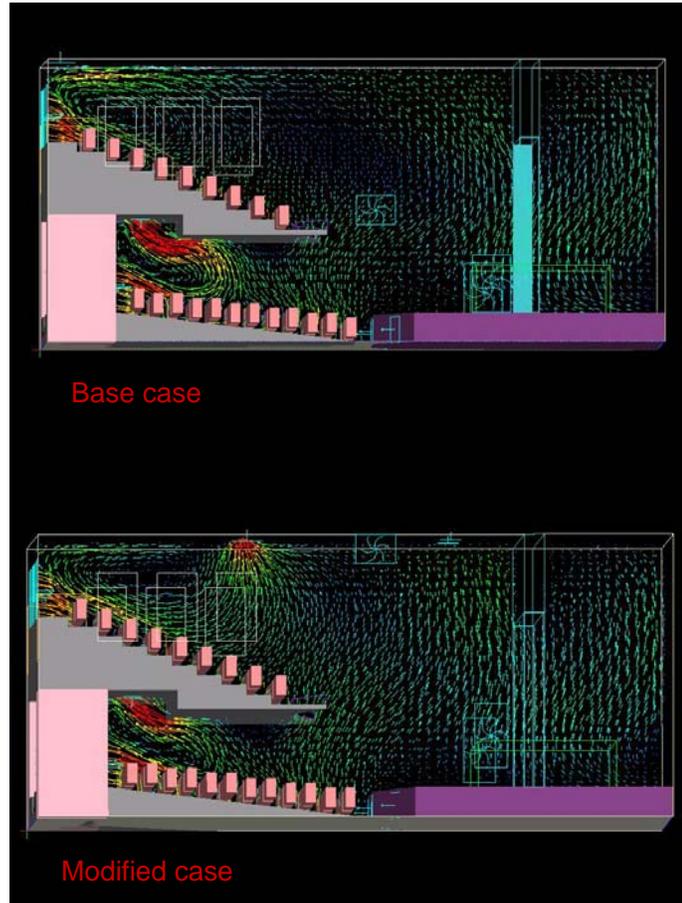
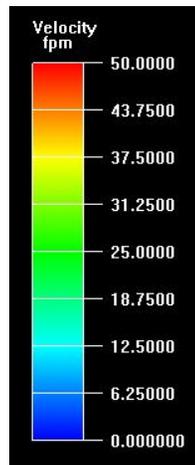
Several modifications were proposed. First of all, in the main floor and around the back of the audience seats, the locations and CFM of the return grilles and diffusers were changes based on the original case simulation results. These changes eliminated the recirculation mode in the balcony level.



Changes in ventilation system that optimized design

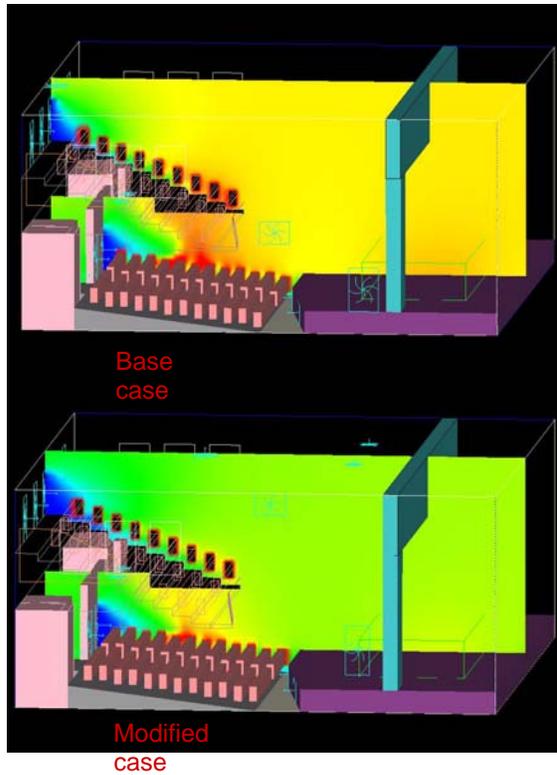
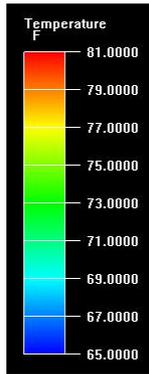
The visual depiction of airflow in the room helped engineers modify the design to correct these problems. Basically, they could see from the simulation results exactly where air was hitting the ceiling and being deflected back to the floor, so they moved the exhausts toward these areas in order to capture the air. The engineers replaced the two 3400 cfm returns on the balcony ceiling with two 2400 cfm returns in the same location and reduced the air flow rate of the wall exhaust to 1400 cfm and moved it to a new location. They reduced the size of the balcony exhaust to 3100 cfm and changed the location and added two new 2750 cfm exhausts in the auditorium ceiling. The engineers also raised the two diffusers on the stage by two feet and increased the size of the one on the left side to 800 cfm because the simulation showed it was much more effective in cooling the heat on the stage.

Velocity Distribution
z @ 38'



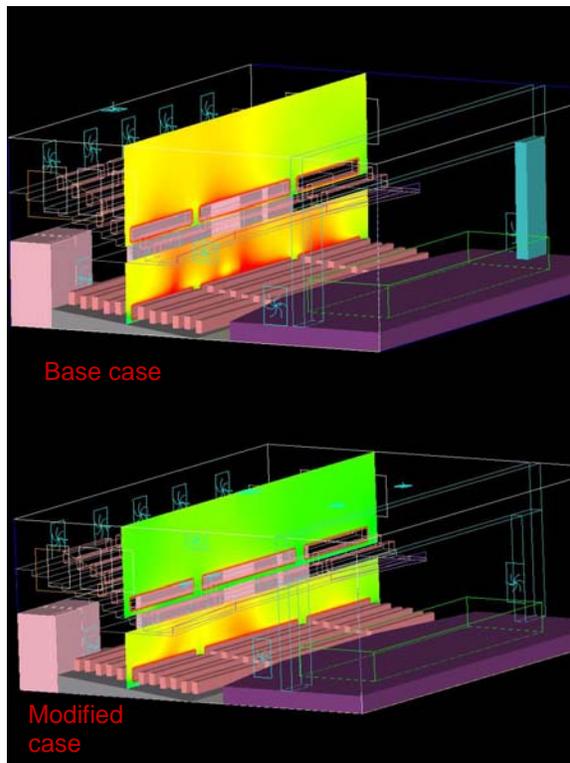
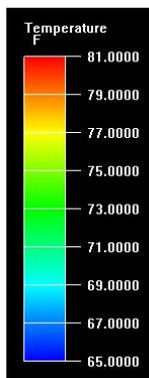
Velocity vector distribution in original and modified design

Temperature Distribution
z @ 38'



Temperature distribution in original and modified design in z direction

Temperature Distribution
x @ 20'



Temperature distribution in original and modified design in x direction

The detailed understanding provided by the CFD results of the original model made it possible to eliminate all of the problems with this design with a single set of changes. The simulation of the modified design showed that the recirculation problems and hot and cold spots were eliminated and that fluid particles now move effectively to the exhausts without making any stagnant area. The modified design also dramatically increased the quality of the air by reducing the mean age of air, especially on the stage.

About the author:

Dr. Hee-Jin Park has been leading the Computational Fluid Dynamics (CFD) division at Flonomix, Inc. He has extensively applied computational airflow simulation techniques to various HVAC/IAQ applications, fire protection analysis, and toxic chemical spreading simulation for last 15 years. He also has researched on advanced HVAC systems including thermal displacement ventilation and adapted computational simulation as a tool to verify the effectiveness of displacement ventilation system. He published numerous technical papers and has been frequently invited for presentation of his work at various technical conferences in his fields. He holds a bachelor degree from Seoul National University and a master degree from Korean Advanced Institute of Science & Technology (KAIST) and a Ph.D. from University of Michigan (Ann Arbor) in mechanical engineering. He is a registered professional engineer in Minnesota and an active member of ASHRAE.

Contact: hjpark@flonomix.com, 503-648-0775 (USA)