BACKGROUND OF PROJECT

Computer simulation helped ensure low carbon monoxide levels in a new parking structure by making it possible to evaluate the performance of different ventilation system designs without the expense of actually building and testing them. The main concern in the design was ensuring that carbon monoxide would remain below specified levels even when 125 cars were waiting to exit the garage with their engines running for a long period of time. Engineers evaluated the performance of the ventilation system diffuser with an easy-to-use computational fluid dynamics (CFD) tool that lets the user accurately model airflow, heat transfer, contaminant transport and thermal comfort for internal as well as external building flows. Five different diffuser configurations were evaluated while the total capacity of the ventilation system was maintained at a constant level. Engineers used the simulation to select the most efficient diffuser configuration, making it possible to build a cost-effective ventilation system that met all performance requirements without any modifications.

CARBON MONOXIDE CONCERNS

The operation of automobiles indoors presents many concerns such, as the emission of carbon monoxide, nitrous oxides, and oil and gasoline fumes. However, it is generally accepted that the ventilation required to dilute carbon monoxide to acceptable levels can control the other contaminants satisfactorily. A secondary concern is maintaining sufficient draft on each underground floor to avoid mold growth. The traditional approach to designing the ventilation system would be to use hand calculations, whose accuracy is reduced by several factors. First, these calculations don't take the geometry of the structure into account. Second, they determine only average carbon monoxide content but not the spatial distribution or gradients in the distribution, which can have an important impact. The result is that engineers are unable to be certain about the performance of the design until the ventilation system is installed and tested. The possibility exists that expensive changes will have to be made after testing is performed.

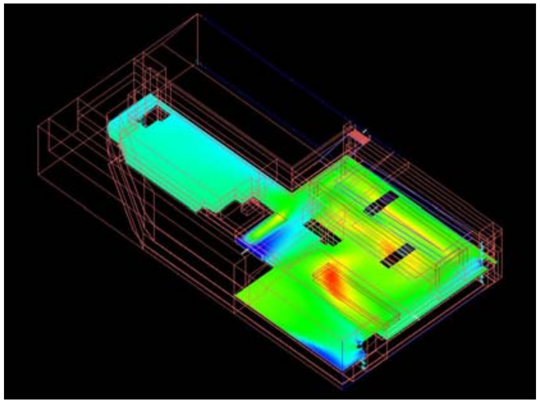
In recent years, engineers have begun using CFD to analyze ventilation system designs and predict indoor air quality in advance. CFD is a tool for analyzing fluid flow and transport phenomena. CFD uses computers to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier-Stokes and allied equations), for predefined geometries and a set of initial boundary conditions, process flow physics, and chemistry. The result is a wealth of predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs. CFD is a very potent, nonintrusive, virtual modeling technique with powerful visualization capabilities.

A CFD TOOL DESIGNED FOR VENTILATION PROBLEMS

The underground parking structure for which the ventilation system was designed is fully enclosed and has seven levels. Its total size is 614 feet by 224 feet by 98 feet. Four sides of the structure are surrounded by the ground and the top is covered by a residential facility. The seven levels are labeled from A – the highest - to G - the lowest - and each has a different floor plan. Level A is on the ground level. There is also an excavation area along the parking levels that creates a vertical flow channel. There are three entrance and exit areas in this structure, on levels A, C and D. The 125 cars were simulated in seven groups. Each car is assumed to generate 4.27 grams of carbon monoxide per minute, based on Environmental Protection Agency estimates, as well as 5 kW of heat. The primary purpose of the analysis is to determine the carbon monoxide levels at all points within the structure. The EPA specifies a maximum allowable level of 35 ppm for one hour while the American Conference of Governmental Industrial Hygienists specifies a maximum of 25 ppm for 8 hours. The American Society of Heating, Refrigeration, and Air Conditioning Engineers standards, which combine both of these requirements, were used as the design specifications for this analysis.

SELECTING THE DESIGN ALTERNATIVES

Another criterion for the design was to minimize the areas with air velocity above 200 fpm in order to avoid draft discomfort and to reduce the area below 20 fpm to avoid possible contamination buildup. At this moment, there is no clear understanding of the relation between the speed of the air and the microbial growth. But it has been widely accepted that low velocities provide a more comfortable environment for microbial buildup. For this reason, special efforts were made to avoid stagnant air in the excavation area. Five different diffuser configurations were evaluated in the analysis. Each of these cases maintains the same total supplied air volume and the same exhaust system. Only the locations and size of diffusers were modified in each case. Case 1 has two columns of diffusers on the north and south side. Case 2 has split diffusers (each diffuser split into two smaller ones) on both the north and south sides. Case 3 has a split diffuser oriented in the 6:00 and 7:30 directions on the north side. Case 4 also has split diffusers that are oriented in the 4:00, 6:00 and 10:30 directions. The idea in Case 4 is to reduce the high draft area by directing the flow into an excavation area. Case 5 maintains the split configuration but diffusers are laid side by side and flow directions are 10:30 and 7:30.

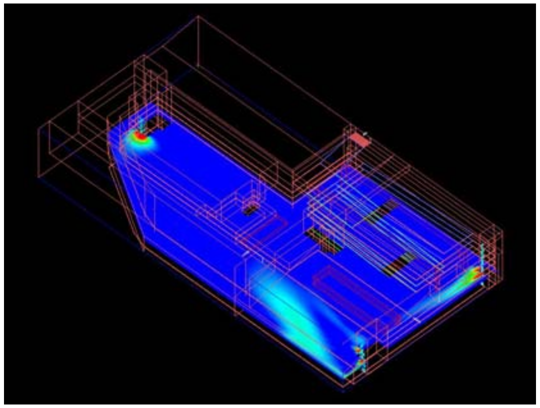


CFD image showing CO distribution on level D

Engineers modeled the five designs by creating an unstructured hybrid grid system that includes hexahedral, prism, and tetrahedral elements. The total number of cells was about 770,000. The standard k-epsilon turbulence model with standard wall functions was used. The ideal gas equation was used to calculate density. The model converged after more than 2000 iterations in about 50 hours with an 800 MHz Pentium III personal computer. Pressure boundary conditions were used for three openings on the A, C and D levels. Outdoor air conditions were assumed to be windless, with a temperature of 55ºF and at a carbon monoxide level of zero.

The results of the simulation included colored contour plots of the carbon monoxide concentration in each area of the facility. On floor F, which was assumed to have no operating cars, cases 1, 3 and 5 showed more areas with carbon monoxide concentration in the 20 to 30 ppm range while cases 2 and 4 were within acceptable levels. In the analysis results for the D floor, which does have operating cars, cases 1, 2 and 5 showed larger areas with a high carbon monoxide concentration than cases 3 and 4. The contamination level in the east side of the structure seemed to be effectively diluted by the flow direction of the split diffusers in cases 3 and 4. On the B level, case 1 yielded more carbon monoxide concentration above the floor while cases 4 and 5 yielded lower carbon monoxide levels near the floor. Looking at the air velocity on the different levels, it was noted that none of the cases exhibited draft problems except for Case 1 on the E level and Case 3 on the B level. Cases 1, 2 and 5 exhibited a stagnant flow region around the elevator lobby.

After a thorough comparison of all of these results, engineers selected Case 4 as the best design in terms of carbon monoxide distribution and airflow. The ventilation system was built to this design and testing showed that it met all the design requirements without any modifications.



CFD image showing CO distribution on level E

FLONOMIX, INC

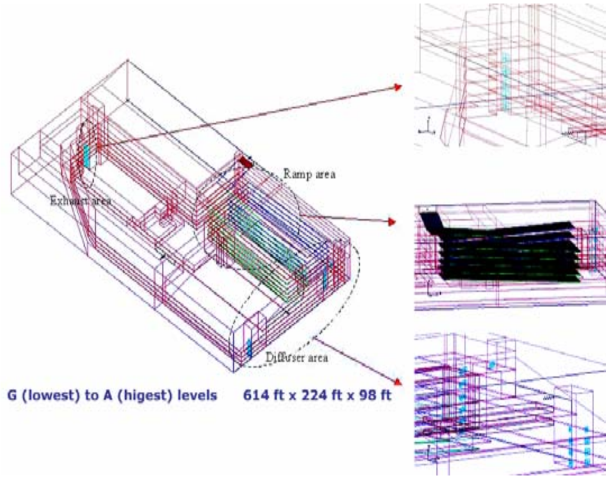
Flonomix, Inc. has been offering computational fluid dynamics (CFD) analysis for various engineering applications, especially for the HVAC/IAQ industry to achieve better indoor air quality through optimization of designs and enhancement of ventilation effectiveness. Among projects in which Flonomix has been involved are computational air flow simulations for smoking lounges, clean rooms, operating rooms, computer cluster rooms, institutional environments, underground parking structures, casinos, hotels, atriums, theaters, and shopping centers.

CONTACT INFORMATION:
Tel) 503-648-0775
Fax) 503-648-0777
Heejin Park at hjpark@flonomix.com
www.flonomix.com

COMPUTATIONAL SOLUTIONS FOR ENGINEERING EXCELLENCE

FLONOMIX ARTICLE SERIES 104

COMPUTER SIMULATION ENSURES LOW CARBON MONOXIDE LEVELS IN PARKING STRUCTURE



Computational domain for CFD simulation for underground parking structure

FLONOMIX, INC

More.info @ www.flonomix.com