CFD ANALYSIS CHALLENGES IN BUILDING SIMULATION FOR SIMBUILD2004 CONFERENCE

Ferdinand Schmid and Galen Burrell
Architectural Energy Corporation

ABSTRACT

This paper discusses the capabilities and challenges of computational fluid dynamic analysis (CFD) in the architectural engineering field. It is intended to provide a user's perspective of CFD challenges and not a scientific report on cutting edge simulation techniques and algorithms. The combination of large dimensions with often non orthogonal geometry poses a unique challenge to CFD software. Models quickly become very large and require more memory than some operating systems and workstations can provide. In addition to these model size issues architectural models need to properly consider ambient weather conditions. Solar loading and the resulting buoyancy driven convection can completely change airflow patterns and comfort conditions in a building. Equally important is proper consideration of exterior wind pressure on building openings. Methods for determining these and other boundary conditions are discussed. Changing weather conditions throughout the year require CFD analysis for several days of the year. Some problems cannot be solved in steady-state mode and require a transient analysis. Economic and project time limitations in the real world demand smart problem simplifications. A variety of methods for simplifying CFD problems without compromising the validity of the results are discussed.

INTRODUCTION

Computational Fluid Dynamic simulation is currently the most detailed building ventilation and comfort analysis method. Air movement, building construction, exterior and interior boundary conditions, and even air composition changes are part of the simulation. This level of detail allows a precise analysis of building comfort beyond thermal comfort. Unlike other building simulation systems, such as DOE2 (http://www.doe2.com), CFD does not assume full air mixing. All air properties are calculated for thousands to millions of locations throughout the simulated space, which allows for analysis of thermal stratification. Humidity, mean age of air, and pollutants are part of the analysis. All of this detail comes at the expense of complexity and computational cost. However, there are many situations where the benefit of a CFD analysis far outweighs its cost.

Historically, CFD analysis was focused on fairly small geometries, such as automotive engines; and material sciences, such as semiconductor manufacturing. The cost of CFD analyses in these situations was small compared to the resulting value in the manufacturing environment. Larger scale CFD analysis was primarily found in academic and high-tech environments, such as aerospace engineering. Recent advances in computer processing power, combined with modern software, allow the application of CFD analysis to buildings. This paper describes how CFD works, how it can be applied to building sciences, and what its current limitations are. The examples used are derived from an analysis for a proposed National Museum of the Marine Corps in Quantico, Virginia. The simulation software was Fluent Airpak. This paper is intended to provide a user's perspective of CFD challenges and not a scientific report on cutting edge simulation techniques and algorithms.

HOW CFD WORKS

System Overview

CFD analysis can be used to predict comfort and ventilation performance for a building during the design phase.

The input for CFD analysis is derived from available CAD drawings and information from the mechanical engineers. The architectural drawings contain many details that need to be removed for CFD analysis. Figure 1 and Figure 2 show the dramatic level of simplification that was done to solve the example problem. This comparison shows that CFD analysis of large, complex buildings requires far more than importing a CAD drawing into a software package and pressing the start button.

Figure 1 - Architect's Rendering of Example Building (Courtesy of Fentress Bradburn Architects)

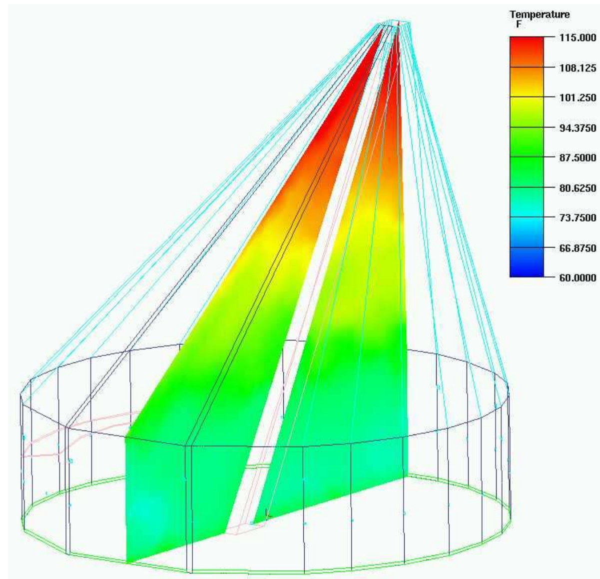

Figure 2 - Thermal Profile (Vertical Cut)

Computational Fluid Dynamics uses the concept of a computational domain, which encloses the analysis space. In our example analysis we were interested in the interior space of the building. All fluid dynamic equations need to be satisfied for the computational domain (e.g. balance of energy into and out of the space). Appropriate boundary conditions must be defined for the exterior surfaces. This includes pressure onto openings such as windows and exterior air temperature.

The CFD analysis is done by splitting the space of interest into a finite number of volume elements. A simple example would be splitting a rectangular room into a number of small cubes. This process of splitting the building space into discrete volume elements is called meshing. During the analysis the software solves a set of differential equations for continuity, energy, and x, y, and z velocity repeatedly until the change per iteration for all mesh cells is below a certain threshold. This process is called solving in steady-state mode. Some problems cannot be solved in steady-state mode because airflow patterns are unsteady. These dynamic cases require a transient analysis, which requires solving the model for a number of time increments.

Building Geometry and Meshing

Some architectural CFD systems can only solve models with a rectangular geometry efficiently because they offer only a hexagonal mesher. For many situations, like data centers, this is an acceptable environment. However, the simulation of complex landmark buildings requires a more sophisticated meshing approach, which Airpak provides through a tetrahedral mesher. A tetrahedral mesh can adjoin surfaces at any inclination and orientation angle. However, it is much more difficult to generate a tetrahedral mesh. Tetra solutions require up to an order of magnitude more computational time. This is caused in part by additional equation terms due to the non orthogonal nature of a tetra mesh. If solving a problem with 100,000 hexagonal grid cells takes 30 minutes, then solving a problem with the same number of tetrahedral mesh cells will take many hours. It is important to limit the number of grid cells to an absolute minimum to control solver times. This requirement for problem simplification mandates manual work. Simply importing a three-dimensional architectural drawing into the CFD system would result in a very complex model. A better approach is to use the imported architectural drawing as a blueprint that can be traced for creating a suitable CFD representation of the building.

The example building shown in Figure 1 represents an architect's rendering of the proposed National Museum of the Marine Corps. The round public space of this building is underground. At its highest point the conical shaped glass roof rises 150 feet above its base. The purpose of the CFD simulation was to predict temperatures in the occupied space and in the roof area. CFD was also used to confirm that no condensation would occur at the glass roof during evening banquets on winter nights.

Supply air for the building is only provided through nozzle diffusers on the perimeter of the occupied space. The large glazed area of the roof produces a high solar gain that needs to be offset by the HVAC system.

Comparing Figure 1 and Figure 2 illustrates the below described simplifications, which were done to eliminate all nonessential details. The simplified model required a tetra mesh of about 800,000 grid cells, which is a small fraction of the grid cell number for a similar model with all architectural details.

The list of simplifications includes:

- The base of the building was approximated through planar walls:
  - → This allows for easy placement of large return vents (rather than curved ones).
    Due to the large diameter of the building this simplification does not significantly impact airflow modeling.
  - → The sloped glass surfaces of the roof can be aligned with the vertical walls without requiring odd shaped interface pieces.
    Since convection is the only airflow in this region (slow velocity) this simplification won't cause different airflow than a straight piece of glass that slightly overlaps a circular wall.
  - → The supply vents can be perfectly aligned with the flat wall sections without requiring curved interface pieces.
    The CFD model has to be tight, without even minimal gaps between adjacent surfaces. Over the vent diameter the curvature of the wall is minimal.
- The round supply air nozzles were approximated through square nozzles with identical physical properties like throw, discharge velocity, and opening area. These square nozzles were much simpler to align to the walls than round nozzles. The primary reason for this simplification are limitations of our CFD software, Airpak. These limitations of Airpak are not present in Fluent's general purpose CFD suite. However, Airpak's ease of use for architectural CFD outweighs this limitation.
  By the nature of its implementation (identical vent performance) this simplification doesn't affect the airflow pattern.
- The glass roof was simulated through planar inclined surfaces.
  Flow velocities close to the glass were expected to be very low, which was confirmed during the simulation. Stepped windows would cause mixing and other effects only when combined with moderate or fast airflow. The larger surface area of the stepped design was accounted for by adjusting the surface temperature of the glass. Differences in solar transmittance due to the different inclination and surface area were also compensated.
- All architectural detail around the glass was removed.
  As described above, differences in solar gain and shading were accounted for through manual calculations. The exterior architectural details of the glazed roof were not part of the simulation so they were not needed in the model.
- The spire doesn't extend past the rooftop.
  The simulation only includes the building interior. Air is mechanically exhausted through the spire. The lack of a moderate stack effect through the removal of the extended spire is insignificant due to the dominance of the mechanical fans.

All of the geometry simplifications serve the primary goal of simplifying the solution mesh and thereby reducing solution time. The complex geometry of the example building mandates the use of a tetrahedral mesh. This mesh needs to offer sufficient resolution near areas of high flow velocity. In wide open areas, however, the mesh can be coarse to minimize the total number of grid cells for computational efficiency. Figure 3 shows a vertical cut of the mesh with its coarse and fine areas.

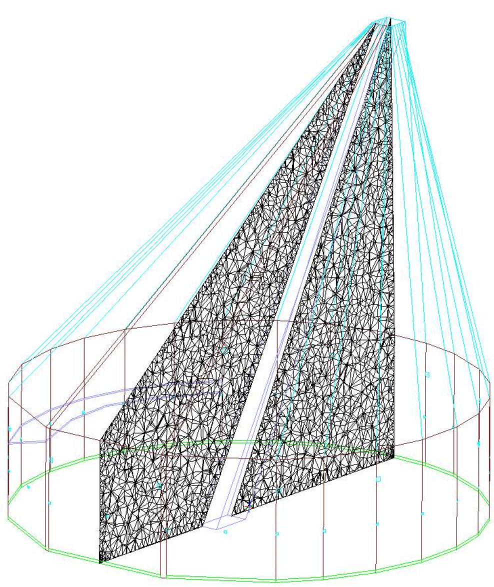

Figure 3 - Tetra Mesh

The quality of a mesh can be verified during the solution phase by comparing results from a simulation with a given mesh to results from a simulation with a finer mesh. Mesh independent results confirm that the coarser version of the mesh is sufficient. Future simulations like parametric studies with various boundary conditions can then be run with the less detailed mesh version.

Boundary Conditions

As with the geometry, determining boundary conditions for a CFD model means finding the simplest accurate description of the objects surrounding the simulation space.

It is important to understand that a steady-state CFD simulation will only finish after all of the surface and fluid properties change less than the convergence criteria defines. For example an interior wall surface will change its temperature until it reaches a point where the thermal conduction to the outside is equivalent to the thermal exchange (convection, conduction, radiation) on the inside. The thermal mass of the wall cannot be considered in a steady-state analysis. With this limitation in mind it is often advisable to simulate well insulated walls as adiabatic. The same applies to interior walls that separate spaces with similar air temperatures. This reduces calculation effort without any significant impact on the simulation results.

The boundary conditions for our example building were defined as follows:

- Adiabatic walls and floor for the occupied space. Materials for these objects were assigned as appropriate to ensure correct emissivity and other physical properties.
- The windows were simulated as glass with a given U value and transmittance (not adiabatic).
- The ambient conditions outside the building were defined through a combined solair temperature, which provides combined radiation and convective heat transfer data to the CFD software.
- No wind effects, such as pressure on building openings, were considered.
- The spire inside the building was simulated as a thermally conducting material (metal).

Solar Loading

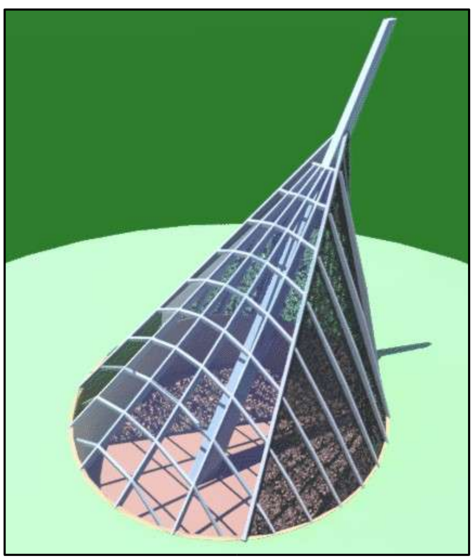

Figure 4 - Simulation of Sunlight

Solar gain is often part of architectural CFD simulations and presents a simulation challenge. Proper calculation of the projected sunlight requires optical calculations for reflection, refraction (as the light passes through windows), and shading from opaque objects. Most of these calculations can be done by modern CFD software. However, sun movement and thermal mass calculations are not part of the CFD software calculations. Solar gain for complex buildings with large window areas should be calculated outside the CFD software. For the example problem we chose Radiance (http://radsite.lbl.gov/radiance/HOME.html) to perform a lighting analysis. We then defined an aggregate heat flux to appropriate surfaces inside the model space to simulate solar gain. Figure 4 shows a simulated Radiance picture of the example building.

Exterior Boundary Conditions

Defining the exterior boundary conditions often requires a lot of work. Many important parameters are weather dependent and difficult to obtain.

A thorough CFD analysis often requires analyses for a variety of outdoor conditions. Below is a list of recommended simulation scenarios:

- Simulation of the hottest expected day and time is necessary to confirm proper cooling and comfort during peak summer heat. Certain types of spaces, e.g. higher zones of atria, don't need to fully satisfy ASHRAE comfort conditions during the hottest hours of summer.
- A typical summer day simulation is required to verify that a proposed cooling design can provide comfort.
- Simulation of the coldest possible winter condition is used to verify that the heating system is capable of maintaining adequate temperature and humidity. In some cases, condensation on exterior surfaces is also a concern that needs to be analyzed.

Some of the above scenarios need to be solved for a variety of ventilation scenarios. The CFD analysis may be used to reduce HVAC equipment costs through the addition of natural ventilation, shading devices, or other energy conservation technologies.

Wind is another exterior boundary condition challenge. It generates positive and negative pressure on operable windows and vents. Accurate calculation of wind pressure for openings often mandates an exterior CFD simulation to properly consider building architecture and the impact from surrounding buildings and trees. A conservative CFD analysis requires several solutions with various wind speeds from all common wind directions.

As mentioned before the thermal exchange of a building with its environment can often be simplified through the use of solair temperatures. Solair temperatures combine radiative, convective, and conductive heat exchange. For well insulated buildings this simplified approach offers significant time savings without sacrificing accuracy.

Solving

The next step after defining geometry, mesh, and boundary conditions is solving the CFD model. This step requires entering a few more parameters, such as selection of a turbulence model (we chose the zero equation indoor model), single or double precision solver, under-relaxation factors, and other solver specific information. Default values are a good starting point but if a solution doesn't converge, changing solver parameters can often solve the problem.

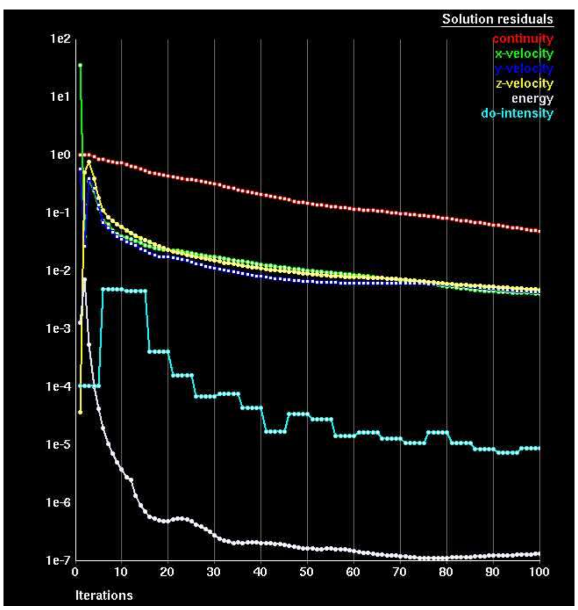

Figure 5 - Residual Plots from Airpak Solver

Depending on the size of the CFD model, it can be desirable to use multiple CPUs during the solution process. When using multiple CPUs the CFD software splits the model into sections and each section can then be solved independently by one CPU. Experience shows that there is little benefit from splitting a CFD model into smaller pieces than 200,000 mesh cells per CPU. We used three dual CPU machines running SuSE Linux (http://www.suse.com) for our simulations. We found that running three concurrent simulations on two CPUs each provided the best efficiency for the example problem.

Airpak's solver allows the use of SMP (Shared Memory Processing) systems or a cluster of computers with independent memory. Supported operating systems include Linux and Microsoft Windows.

Fluent also offers a Remote Solving Facility (RSF), which allows users to solve their problem on a large Unix system with significantly more capability than any workstation can offer. This can be a cost-effective solution for solving many scenarios concurrently and fast. Their fees are based on CPU hours (computation effort).

It is important to watch the residual plots during the solution phase for efficient solving. Once the residual plots diverge there is no point in continuing the solution phase. Figure 5 shows the residual plots for the first 100 iterations during the solution phase of one of the example runs. The convergence criteria is set to 0.001, which means that none of the values may change more than 0.1% between two iterations. The graph shows that a complete solution for this simulation requires many hundred iterations. An increase in residual values indicates a diverging solution and requires intervention. Sometimes the residuals simply flatten out to a horizontal line, which also indicates a problem. Problem solving at this point requires a good understanding of physics and experience. It may be necessary to modify some of the above mentioned solver parameters. Other potential input problems include a not perfectly sealed simulation volume or, as is frequently the case, mesh problems. However, it is also possible that the solution cannot converge because there isn't a stable flow regime for the modeled space. An example would be a large space with low ceiling and no air supply or return (sealed box). If the floor represents a uniform heat source, convection will start in one spot. However, air from the ceiling gets displaced and needs to flow back down. Since there is no favorable space for the location of rising air and dropping air, the rising air plume will migrate over time. This problem cannot be solved in steady-state mode. Problems of this type need to be solved in transient mode. The CFD software will then attempt to solve the model for a finite number of time iterations. Transient solutions multiply the solver time but they are sometimes the only option for solving a CFD problem.

A converged solution does not prove correct results. The results from a CFD simulation require rigorous analysis during post processing to confirm their accuracy. This includes checking the energy balance of the simulation, checking for temperature, pressure, and flow velocity extremes; and verification that the calculated flow regime is realistic.

Post Processing

Once the CFD simulation has finished it needs to be post processed. During this phase of the analysis the calculated physical properties for all mesh cells are read from disk and converted to visual form. The results can be presented in the form of plane cuts with color coded areas, vector plane cuts, or through animations with flying virtual particles that allow displaying the path and speed of airflow.

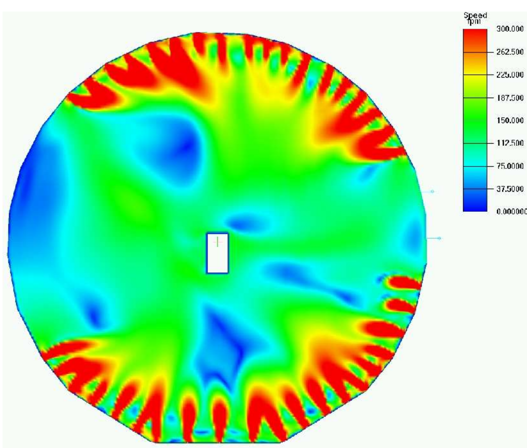

Figure 6 - Velocity Plot at Vent Height

Figure 6 shows a horizontal plane cut for flow velocity at the height of the supply air vents. Supply vents are only placed around the perimeter of the occupied space and CFD was used to verify that fresh air would reach the center of the circular space. The analysis helped determine optimal vent placement, supply air volume, and vent type.

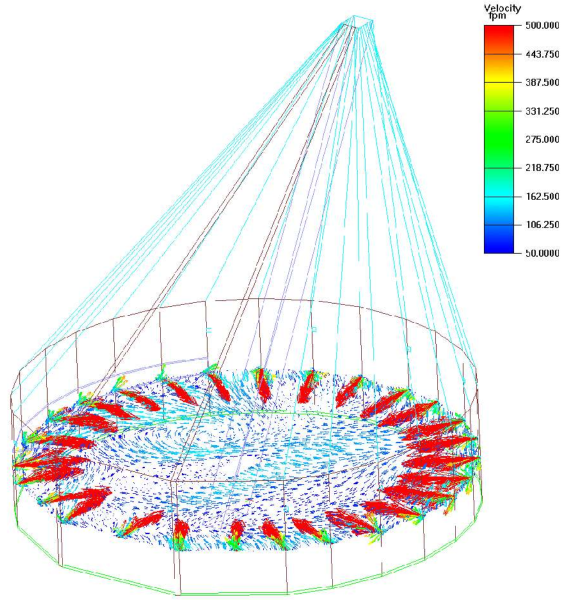

Figure 7 - Vector Plot of Flow at Supply Vent Level

Figure 7 shows the flow at vent level as a vector plot. This macroscopic overview illustrates the general flow pattern inside the space at vent level.

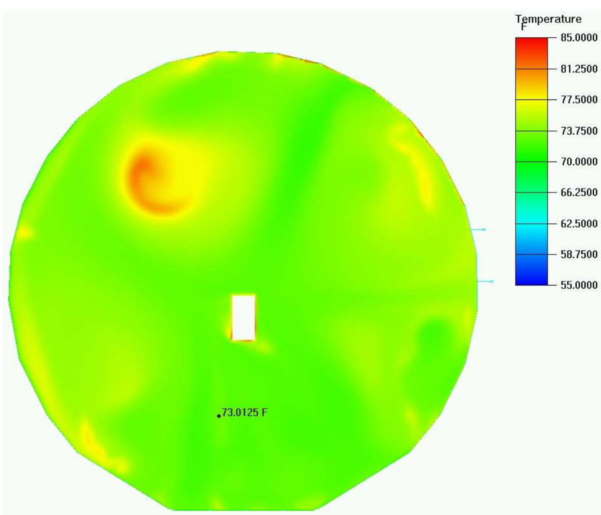

Figure 8 - Horizontal Cut Temperature Plot at six Feet Above Ground

Figure 8 shows temperature in a horizontal plane cut six feet above ground during peak summer conditions. A temperature probe in the picture shows the temperature at its location. This software feature lets users explore physical property details in arbitrary locations. The picture shows that there are warm spots within the conditioned space, as is the case in real buildings. Without a CFD analysis it would be impossible to predict such details and the magnitude of the temperature differences throughout the space. As stated before most conventional simulation software assumes perfectly mixed air.

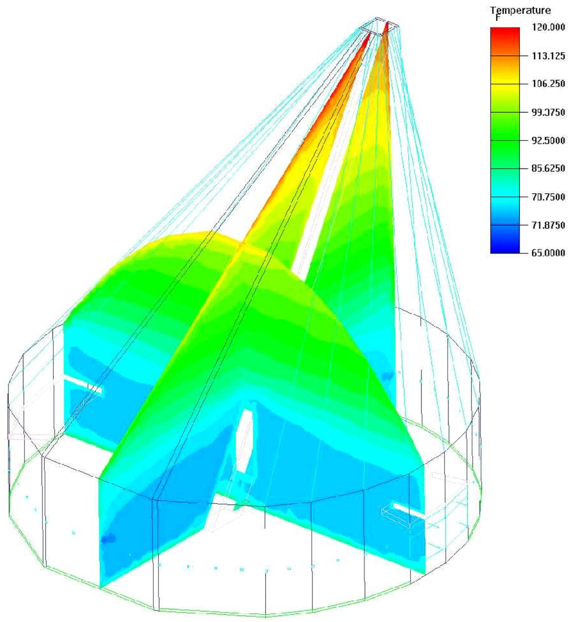

Figure 9- Vertical Plane Temperature Profile

Vertical plane cut temperature graphs as shown in Figures 1 and 9 complement horizontal views. They show that temperature stratification occurs inside the building and that the temperature in the occupied space is within its allowable range. They also confirm that the peak temperature at the top of the roof is within acceptable limits, thanks to the exhaust system, which is placed at the top of the spire.

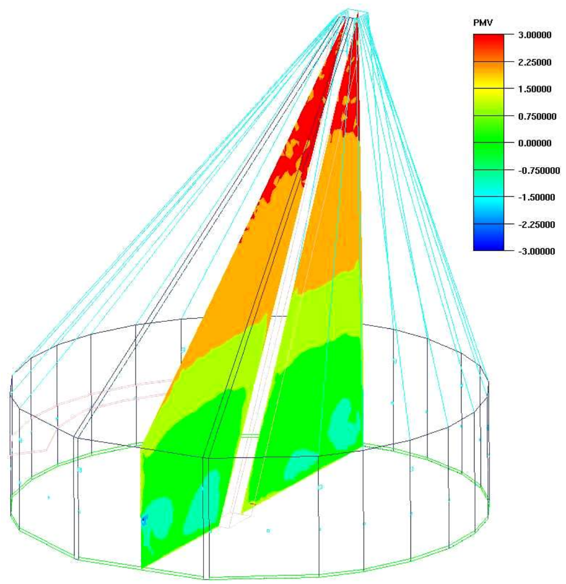

Figure 10 - Vertical Plane Cut For Predicted Mean Vote (PMV)

The most important performance goal for HVAC systems is comfort, which depends on humidity, air movement, and radiation in addition to temperature. Airpak is capable of calculating a comfort metric called Predicted Mean Vote. Positive PMV indicates the percentage of the population (people) who are too hot, negative numbers indicate the percentage of the population that are too cold. Of course clothing matters for this assessment. Figure 10 shows a vertical plane cut PMV graph for building occupants wearing short sleeve shirts and light trousers, doing moderate physical activity (standing and walking).

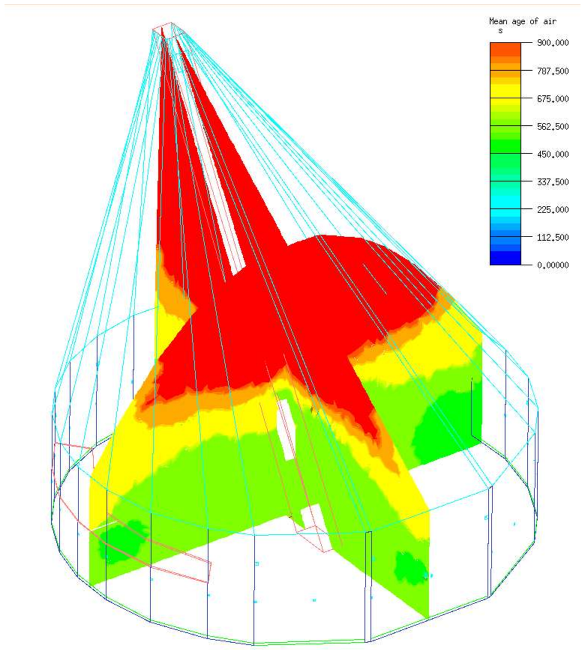

Figure 11- Mean Age Of Air Plane Cuts

Another valuable air quality metric is the mean age of air. CFD calculates all the required data to analyze the mean age of air. This metric is especially important in spaces with high occupancy and in rooms with polluting equipment, such as photocopiers. Figure 11 presents the mean age of air for one of the ventilation scenarios in our example building. The mean age of air in the occupied zone is between five and seven minutes, which is acceptable.

Conclusion

Computational Fluid Dynamic analysis for complex buildings requires significantly more effort than conventional building simulation, such as DOE2 models. The simulated building needs to be simplified through careful analysis to create a representative model without any unnecessary detail. This simplification may require hand calculations and even computer simulations. Radiance is a good tool to analyze solar gain, DOE2 is suitable for predicting the peak cooling and heating day for the building, and external CFD simulations can be used to accurately predict wind pressure on openings.

The computational effort for solving complex CFD models is significant. Simulating a complex building within the allowable time frame between architectural reviews may require access to at least a small cluster of computers. Combined hardware, labor, and software costs far exceed conventional building simulation costs.

The information from a CFD analysis is very valuable. CFD can help avoid millions of dollars in cost through an accurate prediction of heating and cooling loads. It helps design better buildings with improved comfort. The operational costs of CFD optimized buildings is typically much lower due to more efficient HVAC designs, shading systems, and other technologies; and the risk to architect and building owner when planning and building new and innovative structures is significantly reduced.

Of course, not all CFD models are as complicated as our example building. There can be very good reasons to do a CFD analysis in simple buildings. If smoke or pollutant management, quality of air at a detail level, or other detail ventilation aspects are important, then CFD is the answer. Examples could be airport terminals, factories, server rooms, and a long list of other applications. The required skill and computing resources are much lower for structures with a simple geometry, which makes CFD even more attractive.

ACKNOWLEDGMENTS

- We would like to thank Fentress Bradburn Architects (http://www.fentressbradburn.com) for allowing us to use their rendering of the proposed National Museum of the Marine Corps. We also thank them for permitting us to use CFD images that resulted from the above project in this paper.
- We would like to thank Mr. Walter Schwarz and Mr. Viralkumar Gandhi for their outstanding expert support for Fluent Airpak.