CFD INTRODUCTION

CFD is an acronym that refers to “Computational Fluid Dynamics”. CFD uses numerical methods to solve the fundamental nonlinear differential equations that describe fluid flow (the Navier-Stokes and allied equations), for predefined geometries and boundary conditions. The result is a wealth of predictions for flow velocity, temperature, density, and chemical concentrations for any region where flow occurs.

A key advantage of CFD is that CFD is a very potent, non-intrusive, virtual modeling technique with powerful visualization capabilities and engineers can evaluate the performance of a wide range of IAQ/HVAC system configurations on the computer without the time, expense, and disruption required to make actual changes onsite.

CFD is one of the areas that are being dramatically developed for the last several decades. This technique has widely been applied to various engineering applications like automobile and aircraft design, weather science, civil engineering, and oceanography. Today, HVAC/IAQ industry is one of the fields that start to utilize CFD techniques widely and rigorously in its design.

PRACTICAL ADVANTAGES OF EMPLOYING CFD

The followings are among the many reasons why CFD is being widely used today:

CFD predicts performance before modifying or installing the systems:
Without modifying and/or installing actual systems or prototype, CFD can predict what design change is most crucial to enhance performance.

CFD provides exact and detailed information about HVAC design parameters:
The advances in HVAC/IAQ technology require broader and more detailed information about the flow within an occupied zone, and the CFD technique meets this goal better than any other method, i.e., theoretical or experimental methods.

CFD saves cost and time:
CFD costs much less than experiments because physical modifications are not necessary. (Note that the cost and time for physical changes/modifications increase almost exponentially as the size of the system increases).

CFD is reliable:
Most importantly, numerical schemes and methods that CFD is based on are improving rapidly so that reliability on the results produced by CFD is getting very high. Increased reliability makes CFD a dependable tool in any design and analysis purpose.

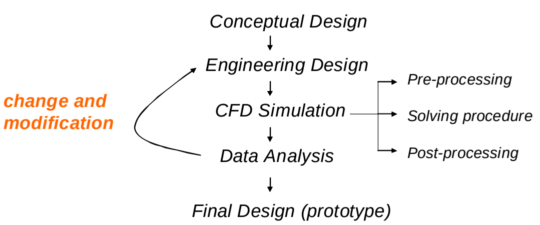
WHERE CAN CFD BE UTILIZED?

In validation/Optimization of HVAC design parameters:
CFD data can be utilized to validate various design parameters such as the location and number of diffusers and exhausts, temperature and flow rate (CFM) of supplied air to meet design criteria.

In modification/Improvement of malfunctioning HVAC systems:
The system with suggested modifications can be simulated computationally without actual physical modifications to the existing systems. The information from CFD reveals what modification satisfies the design criteria.

In comparisons between alternative systems:
Under some circumstances, there may be several different options for designing HVAC systems for a space (for example, mixing ventilation or displacement ventilation). Computer simulation data can provide crucial information to find the best possible system.

CFD AS AN ENGINEER'S TOOL
THE CONCEPT OF VIRTUAL PROTOTYPING



In an engineering investigation:
CFD analysis of temperature, velocity and chemical concentration distributions can help engineers understand the problem correctly and provide idea for best resolution.

CONSULTING PROCDURE

- We will discuss the projects for which you like to use computational analysis. The projects can involve modifications, improvements, design validations or any other engineering issues.
- We will team up to find a way to resolve the situation with computational techniques.
- We will submit a proposal describing our approach, time schedule, and fee for the project.
- Upon your approval of the proposal, we will start the computational analysis.

MORE DETAIL INFORMATION ABOUT CFD

This process starts with converting unsolvable governing equations (Navier-Stokes equations) to a solvable set of algebraic equations for a finite number of points (usually around a million of cells) within the space under consideration. By visiting and solving the equations cell by cell as well as an iteration technique, all detailed information for velocity, pressure, temperature, and chemical species within that space are acquired as a whole.

Theoretically, to analyze the fluid flow, the basic conservation equations have to be solved. The equations that govern the flow include those for the conservation of momentum (Navier-Stokes equations), the conservation of mass (continuity equation) and the conservation of energy (energy equation). All of these equations are in a form of a partial differential, non-linear equation that rarely issues exact solutions for most of the cases.

The principal approach of CFD is to represent those equations as well as flow domain in discretized form by using one of “finite differencing”, “finite element”, or “finite volume methods”. Each discretization scheme differs in the assumption of profile within a small volume considered and the way space is discretized. Once discretized, it leaves meshes that cover the whole domain and a set of algebraic equations for that small volume (control volume). Whenever the linearization procedure is necessary, a iterative calculation procedure must be adopted, whereby the equations are successively re-linearized and solved until the solution to the original numerical form of the equations is attained.

There are two ways to solve the set of algebraic equations obtained by discretization, direct method (i.e., those requiring no iteration) and iterative method. One of the direct methods is called the Tri-diagonal Matrix Algorithm (TDMA), which is very efficient but applicable only in 1-D applications because the direct method is usually involved with matrix inversion that may cause very expensive calculations in 2-D or 3-D problems. Alternatively, the iterative method is widely accepted since it is stable and applies to 2-D and 3-D situations. It starts with guessed variables and uses the algebraic equations to get improved variables. It goes on until the difference between new values and the previous values is minor. Then the converged solution is acquired.

EXAMPOLES OF CFD APPLICATIONS FOR HVAC SYSTEMS

- General office/room simulations
- Contaminant/species simulations
- Fume hood design
- Copy machine room (VOC)
- Contamination control chemical lab design
- Industrial ventilation design
- Smoking lounge
- External building flows
- Problem solving simulations
- AHU mixing enhancement investigation
- Fire and smoke management
- Building atria fire simulation
- Warehouse fire simulation
- Educational facilities
- Library
- Classroom
- Swimming Pool ventilation
- Medical facilities (operating room)
- Clean room simulation
- Animal and plant environments
- Enclosed vehicular facilities
- Hall, stadiums, arenas, and places of assembly
- Computer cluster room

FLONOMIX, INC

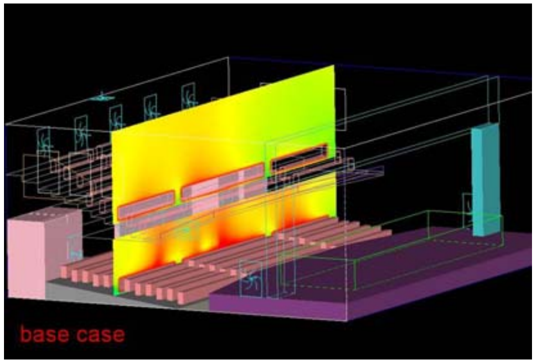
Flonomix, Inc. has been offering computational fluid dynamics (cfd) analysis for various engineering applications, especially for HVAC/IAQ industry to achieve better indoor air quality through optimization of designs and enhancement of ventilation effectiveness. Among projects that Flonomix has been 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:
6375 NE Alder St. #D
Hillsboro, OR 97124
Tel) 503.648.0775; 612.235.6656
Fax) 503.648.0777
Heejin Park at hjpark@flonomix.com
www.flonomix.com

COMPUTATIONAL SOLUTIONS FOR ENGINEERING EXCELLENCE

FLONOMIX ARTICLE SERIES 101

INTRODUCTION TO AIR FLOW SIMULATION FOR INDOOR AIR QUALITY/ HVAC APPLICATIONS



Temperature gradients in a public auditorium

FLONOMIX, INC

More.info @
www.flonomix.com