4.0 What is CFD?

This section briefly outlines the fundamental aspects of CFD, the equations utilised through this process.

Computational Fluid Dynamics can be summarised by the following definitions:

> The computational part of CFD means using computers to solve the equations which predict the Fluid Dynamics. This can be compared with the other methods, such as full or part scale mock-ups.

Computational

> Fluid means any substance which cannot remain at rest under a sliding, or shearing stress.

Fluid

> Dynamics is the study of objects in motion and the forces involved. The field of fluid mechanics is similar to fluid dynamics, but usually is considered to be the motion through a fluid of constant density.

Dynamics

4.1 Governing Equations of Fluid Dynamics

The governing equations of fluid dynamics represent the conservation of mass, momentum, and energy for a fluid continuum. The Governing Equations have actually been known for over 150 years. In the 19th century two scientists, Navier and Stokes, described the equations for a viscous, compressible fluid, which are now known as the Navier-Stokes Equations. These equations form a set of Differential Equations. The generic form of these relationships follow the advection diffusion equation:

$$\frac{\partial}{\partial t}(\rho\varphi) + div\left(\rho\nabla\varphi - \Gamma_{\varphi} grad\varphi\right) = S_{\varphi} \quad (2.1)$$

transient + advection - diffusion = source

The variable PHI (φ) represents any of the predicted quantities such as Air Velocity, Temperature or Concentration at any point in the 3-dimensional model. This equation is derived by considering a small, or finite, volume of fluid. The change in time of a variable within this volume added to that advected into it, minus the amount diffused out is equal to the amount either created or destroyed. Though deceptively simple, only the emergence of ever faster computers over the past two decades has made it possible to solve the real world problems governed by this equation.

Despite their relatively old age, the Navier-Stokes (N-S) Equations have never been solved analytically. It is the numerical techniques used to solve these non-linear and coupled mathematical equations which are commonly known as Computational Fluid Dynamics, hence CFD for short. CFD is the only means for generating complete solutions at the present time.

For a particular physical configuration, boundary conditions are defined which represent the friction, heat transfer, air flow etc. These are included as part of the source term

Over the past 25 years CFD techniques have been extensively and successfully applied in advanced technology, such as the nuclear and the aerospace industries. In its raw and general form, CFD has always been the forte of fluids experts. However, recently the concept of tailoring CFD software, combined with the appropriate expertise in the market segment being addressed, specifically building heating and ventilation, has made it possible to apply these powerful methods to provide fast and accurate results to designers under severe time and budgetary constraints. In fact this project would not have been practical without these new elements in place.

5.2 How does it work?

In order to generate a CFD solution, two processes must be accomplished,

Geometry Definition and Grid Generation
Numerical Simulation

In broad terms, Grid Generation is the act of specifying the physical configuration to be simulated and dividing it up into a three dimensional grid containing a sufficient number of small regions known as control volume cells so that the Navier-Stokes partial differential equations can be solved iteratively. Numerical Simulation is the process of applying a mathematical model to that configuration and then computing a solution. In a typical 3-dimensional calculation the following variables would be present :

- pressure
- velocities in three directions
- temperature
- concentration

and

- turbulence quantities

The solution for each variable will depend upon the solution for each and every variable in the neighbouring cells and vice-versa. Turbulence is modelled using the well established and robust two parameter method known as the κ-ε model where κ represents the kinetic energy and ε the rate of dissipation.

The solution method is iterative with each iteration resulting in a set of errors. At the end of each iteration the errors for each variable are summed. A solution is reached when the sums of the errors, from all the cells, for each and all the variables, reaches a pre-determined and acceptable level.

4.0 Summary of Conclusions

2D or 3D Geometry

2D or 3D?

- For an engineering solution to bulk airflow into a room, a good place to start is to take a slice of that particular room and post CFD, multiply the result by the 2D to 3D window ratio.
- Unless the ventilation openings, geometry and heat gains are almost two dimensional in nature, typically a 2D model would be unsuitable for predicting penetration rates and internal flow patterns.

Representation of opening

Opening?

- Modelling explicitly is recommended wherever possible. If not, care must be taken in the assumptions that are made about the opening. i.e flow patterns and penetration will differ dramatically if using openings for particular window geometries and can not be assumed correct.
- When modelling openings explicitly, to obtain an engineering solution, use a generic form. This will reduce grid and processing time while still achieving a good solution.

Representation of Building

Building?

- For bulk flow within a room, it is possible to model just one of the floors rather than all of them (assuming all floors are geometrically the same). For approximations, this should be adequate, even for temperatures as increase in window detail (possible as you are only modelling one floor) reduces the total variation over more than one floor.
- To see the effects of re-entrainment for no wind worse case scenarios and the variations in floor temperatures, a full simulation is required.

Exclusion of ambient spaces

Ambient Spaces?

- To gain a reasonable value quickly for flows, omitting the outside ambient air will produce slightly over estimated flow rates. All flow rates fall within a 5% to 10% variation of the true value. This increase could however be taken off post CFD to obtain a more realsitic value.

Representation of heat gains

Internal Features?

- Excluding all internal details, (often unavoidable at early stages in design) gave errors of the order of 15% in inflow and up to 50% in penetration. If penetration is an issue then internal details must be modelled.

Surface heat transfer coefficients

Heat Transfer?

- It is important to incorporate surface heat transfer into a model for an accurate solution. Disregarding it will lead to inaccuracy especially where there are large temperature differences external to the model.
- Fixed heat flux rates should only be employed if the surface heat output is known.
- Typically, a fixed surface heat transfer coefficient is employed to avoid the major increase in grid required for a computed coefficient.

Portion of the building modelled

Portion of Building?

- If your design has any planes of symmetry within it, they can be exploited. This will reduce the grid required and will also speed up the calculation process. The risk is that it may impose false stability on the flow.
- If a case does not exhibit two dimensional geometry, it is not advisable to simulate by just taking a slice through the building. Three dimensional modelling is essential.

Abstract

Computational fluid dynamics (CFD) modelling is being promoted as a tool for predicting ventilation rates and air flow patterns as part of the building design process, and increasingly it is being used as such. The potential benefits of this form of modelling are that designs can be optimised to make the most efficient use of ventilation, and so to increase air quality and decrease energy use. Although CFD has shown itself to be a powerful tool in the nuclear, aeronautical and electronic industries for over two decades, its reputation has been built on extensive work specific to those fields. It is recognised that there is a need to establish guidelines for and validate the application of CFD to building design, and in particular to schemes involving natural ventilation.

Target Audience

This guide begins to address these matters and is aimed at any CFD practitioner applying CFD to naturally ventilated buildings.

Funding

This document has been produced by Flomerics and Halcrow Gilbert Associates with financial support from the Department of the Environment. Its aim through research has been to develop guidelines for engineers and designers of naturally ventilated buildings which will enable them to apply CFD appropriately to their buildings.

Issues

The work addresses the key unresolved issues which face practitioners when they use CFD as a design tool for naturally ventilated buildings including the sensitivity of the results to the following parameters:

- how much of a building and its surroundings are modelled;
- the treatment of openings eg representation of windows;
- the treatment of surface heat transfer;
- the level of detail in representing the internal geometry and heat sources.

Applicability of this document

The factors considered in this guide are those of representation of the features of a building in terms of how they can be simplified. The detail of how this is effected in a particular CFD program will differ from one piece of software to another. However, the guidelines in this report do not depend on these details, but rather on the basic level of physics employed in the representation. The guidance in this report can therefore be expected to apply to the use of any CFD program when modelling naturally ventilated buildings.

Contact

For more information about this project please contact:

Mark Seymour
Flomerics Ltd.
81 Bridge Road
Hampton Court
Surrey
KT8 9HH

Tel: 0181 941 8810
Fax: 0181 941 8730

Dr Robert Cohen
HGA Ltd.
Burderop Park
Swindon
Wiltshire
SN4 0QD

Tel: 01793 814756
Fax: 01793 815020

FLOMERICS