THE APPLICATION OF CFD TO LARGE SCALE INDUSTRIAL PREMISES

Duncan A. Phillips, Howard D. Goodfellow

Goodfellow Consultants Inc.
Mississauga, Ontario, Canada

ABSTRACT
This paper introduces a research programme investigating the application of CFD to large scale industrial premises. A number of modelling issues and two case studies are discussed. The research programme will lead to an increased degree of confidence of CFD simulation results in complicated environments.

KEYWORDS
CFD, industrial ventilation, large enclosure

INTRODUCTION

The application of computational fluid dynamics (CFD) to ventilated environments is becoming increasingly common. CFD is recognized as a powerful design and diagnostic tool capable of providing an analysis of flow patterns within enclosures. As part of this recognition, there has been a concerted international effort to investigate the role of modelling parameters on a CFD solution. International Energy Agency (IEA) Annexes 20 "Air Flow Patterns Within Buildings" (Moser, 1991) and 26 "Energy Efficient Ventilation of Large Enclosures" (Moser, 1993) have generated valuable contributions from researchers in the application of CFD to ventilated spaces.

The focus for the present work is in the application of CFD to large scale industrial premises. Moser (1993) defines a large enclosure to be an enclosure occupied, at least in part, containing various heat and contaminant sources: it distinguishes itself from other ventilated spaces because of the strong buoyancy effect creating vertical streams of air. Typically, most of the previous applications of CFD to large ventilated environments described have boundary conditions that impose temperature differences (e.g. atria). A large scale industrial environment here will be defined as an environment that has all the characteristics defined by Moser (1993), but also encloses dominant **internal** sources of buoyancy and/or momentum.

Emmerich (1997) provides a comprehensive review of CFD applied to occupied settings with a discussion of CFD in large spaces which are predominantly atria. In fact, the vast majority of the applications of CFD to large ventilated spaces have been of atria.

It is here that the foundations for the present research programme is based. Many of the publications on atria using CFD (e.g. Kato *et al.*, 1995, Schild *et al.*, 1995), and articles addressing specific issues such as wall turbulence models (e.g. Renz & Vogl, 1996), provide guidance for the treatment of modelling atria. The purpose of this paper is to identify modelling issues and review recommendations from previous work for application to flows within large scale industrial enclosures. The paper does this through the discussion of two case studies of industrial environments and describes the complications in each.

This paper does not discuss CFD theory specifically. There are a number of references in the literature that provide good reviews of CFD theory e.g. Rhie & Chow (1983), Nielsen (1994), and Chen (1997). Furthermore, it does not delve into the merits of either Large Eddy Simulation (LES) or Direct Numerical Simulation (DNS) over

CFD.

CASE STUDIES

The purpose of the current research programme is to develop a high degree of confidence in the results from CFD simulations of the flows within large scale industrial environments. To do this, it is necessary to have a clear understanding of the role the numerous variables play on the parameters of interest. The variables identified for study in this research programme include: geometry specification, boundary conditions, turbulence models, modelling momentum sources (e.g. supply jets), heat transfer, the relative roles of buoyancy and momentum driving forces, modelling partial flow obstructions, solution stability for under-specified problems, data for validation of solutions and modelling transient releases of high temperature species.

Figure 1 shows the layout of an industrial facility of interest to the authors. It is a melt shop within a larger foundry. This facility measures 37.5 m in length (x), 27.0 m wide (y) and 11.0 m high (z) at the tallest point. There is a door at the South end of the trench that opens into the adjacent part of the foundry, and there is a large doorway in the North end. The volume is approximately 11,400 m^3. There are four melting furnaces, and three holding furnaces. The origin for the grid is located at the North West corner of the facility.

The facility has three roof level general ventilation extraction points each designed to draw 14.2 m^3/s. It also has local exhaust capture rated at 5.7 m^3/s at each of the four furnaces along the East side of the trench. The three furnaces along the West side of the trench also have local exhaust capture, although only two of them are operating at any given time. They are individually rated at 4.7 m^3/s. In addition there are three local exhaust points towards the North end of the facility on the East side of the trench which total an additional 14.2 m^3/s extraction. Table 1 summarises the exhaust flow rates.

Table 1 - Extraction Flow Rates from Facility Presented in Figure 1.

Location	Flow Rate [m^3/s]	Flow Rate [cfm]
3 General Exhaust Fans in Ceiling	42.5	90,000
Local Exhaust at East Furnaces	22.7	48,000
Local Exhaust at West Furnaces	9.4	20,000
3 Local Exhaust Points in NE Corner	14.2	30,000
Total:	88.8	188,000

The air balance in this facility is provided through make-up air entering ducts located along the East and West walls. These are used to provide cooling to the workers performing tasks near the furnaces. In addition, there are three roof level sources of low momentum air that are designed to clear the fume from the upper corners of the building. These are long plenums with many small holes serving as vents. Finally, the volume below the decks on both the East and West sides of the trench are ventilated. The air introduced into these volumes enters the main facility along the vertical walls around the furnaces. Table 2 summarises the total flow rates entering into the facility.

This problem presents many design challenges. There are four competing requirements for the ventilation system: 1) maintaining indoor air quality; 2) reducing the ambient temperature near the operators; 3) capture of fugitive emissions; and, 4) minimizing costs. Furthermore, there are several operating conditions two of which provide contradictory design requirements, Summer and Winter.

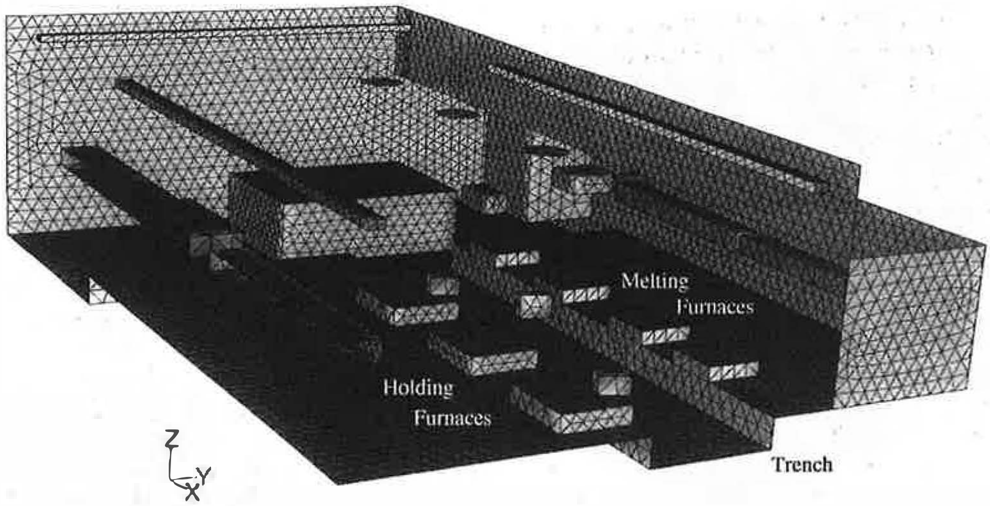

Figure 1 - CFD Industrial Case Study With Momentum Sources

Table 2 - Supply Air Flow Rates from Facility Presented in Figure 1.

Source Location	Flow Rate [m^3/s]	Flow Rate [cfm]
NE duct with 6 grills	13.2	28,000
SE duct with 4 grills	10.5	22,300
NW duct with 5 grills	7.9	16,750
SW duct with 4 grills	11.8	25,000
3 Roof level cleaning vents	1.7	3,600
East deck wall	13.2	28,000
West deck wall	14.2	30,000
Total:	72.5	153,650

The second case study presented in this paper is of a facility containing a Basic Oxygen Furnace. This is shown in Figure 2. The facility measures 28 m high, 16 m wide, and 30 m long. The furnace, represented by the cylindrical and rectangular body in the centre of the building measures 18 m high, and is 6 m in diameter at the base, which is the widest point. The facility is characterized by large openings in the exterior boundary. Approximately 75% of the total boundary is open, with the lower sections completely open, and the upper sections slatted to outdoors. With the exception of pressure driven forces from the wind, the sole dominant driving mechanism is natural convection generated by heat transfer from the furnace. The combination of horizontal platforms and natural convection flow provide a challenge for CFD modelling.

The requirements for this facility are indoor air quality and capture of fugitive emissions.

Other industrial scenarios relevant to this work are discussed in Thompson & Goodfellow (1997). This paper discusses a number of cases, including a transient fume capture problem.

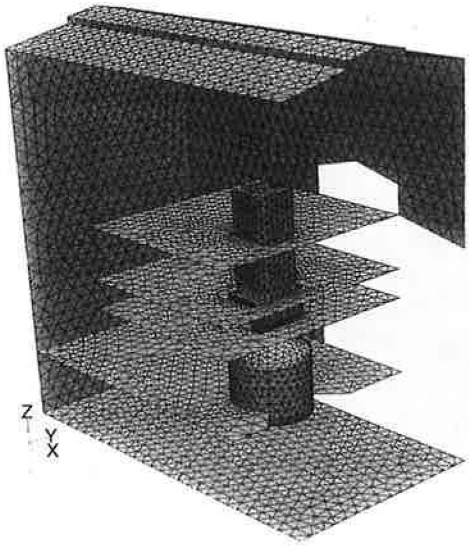

Figure 2 - CFD Industrial Case Study With a Single Buoyancy Source

LITERATURE

As stated above, the starting point for this research programme is based in the knowledge generated through work primarily on atria. The conclusions of Annex 26 include: knowledge of the boundary conditions is essential; radiative heat transfer is important in large spaces with glazing; care must be taken to model convective heat transfer properly; and the k-ϵ model is adequate for ventilation flows.

Turbulence

There is a vast amount of literature on the various turbulence models available. Many of these discuss the degree of accuracy of the numerical predictions to velocity measurements. For instance, Murakami and Kato (1989) describe the validation of the k-ϵ model for a number of iso-thermal room airflows and conclude that the model provides reasonably good results for ventilation flows. Within a large industrial space, one expects to find heat sources, and thus it is important that the buoyancy production and/or suppression of turbulence is understood. Chen *et al.* (1990) conclude that including the buoyancy terms in a low-Reynolds-number formulation of the k-ϵ model provides good agreement with the velocity and temperature profiles, along with the convective heat transfer rates.

One investigation of turbulence models within industrial-like environments specifically has been reported in Dupuis *et al.* (1993). These authors investigated the performance of the k-ϵ model versus the differential Reynolds flux model and compared the simulation results to data from a 1:24 model for both natural and mixed convection cases. The conclusion of interest from this work is that the Reynolds flux model predicted thermal plumes rising from hot equipment better than the k-ϵ, however it gave very poor predictions of temperature distribution near the floor.

It is well known (e.g. Chen, 1997) that the k-ϵ model is not capable of adequately predicting recirculating flows. In many cases within a ventilated space, especially those with high velocity jets, the Reynolds Stress Model (RSM) is more appropriate. One paper of particular interest here is that by Chen (1996). This paper compares the results of three Reynolds stress formulations to the k-ϵ model in three cases: natural convection, isothermal forced convection and mixed convection. In general, it is reported that the Reynolds stress models perform better than the k-ϵ model for the three cases reported.

The disadvantage of using the RSM is the computational effort required to model the six additional "stress" terms. Peng *et al.* (1996) report the application of a two equation model similar in form to the k-ϵ but instead of modelling the turbulence dissipation rate (ϵ), it models the specific turbulence dissipation rate (ω), and includes a source term to suppress the turbulence specific dissipation rate near walls. This has the potential advantage, in the limit as $k \rightarrow 0$, of providing a low Reynolds number prediction.

Heat Transfer

Heat transfer within an industrial setting can occur in through radiation and both forced and natural convection. In cases where heat transfer plays a dominant role, accurate prediction of heat transfer is essential in order to model the driving forces appropriately. Here too there has been knowledge generated that is of use in an industrial setting. Chen *et al.* (1995) provide a time dependent thermal simulation of a room including radiation and conjugate heat transfer (CHT). One of the important points in this paper is that modelling CHT and radiation is expensive. As an alternative, if one specifies a wall temperature, then the CHT modelling is not required, nor is the long wavelength radiation heat transfer.

For many simulations of flow in atria (e.g. Kato *et al.*, 1995 and Schild, 1996) the radiation heat transfer is split into two components - long wave and shortwave. The solar (short wave) radiation is modelled independently of the long wave (emission from room temperature surfaces) as the room temperature has no effect on the radiation received from the sun. However, in industrial settings, this breakdown is not possible, as one of the important considerations involved in the separation of the long- and short-wave radiation in atria is the presence of window glazing. These do not permit long-wave radiation to pass through (Kato *et al.*, 1995). However, in industrial settings, the high temperature radiating source(s) is (are) inside the facility: the surface temperature of molten steel is on the order of 2000 K. Thus the net radiation between a surface at **near** room temperature, and the molten steel might be approximated by assuming a net heat transfer to the walls from the furnace via a source term at the wall.

Of equal importance is the knowledge regarding the convective heat transfer from hot industrial equipment to the surrounding air. There are three options for this boundary condition: 1) specify the wall temperature; 2) specify the heat transfer rate; 3) and specify the heat transfer coefficient (with or without the wall temperature being specified). Schild *et al.* (1995) describe the various issues associated with the different modelling options. An important point that arises from Schild *et al.* (1996) is that wall functions are lacking. However, Renz & Vogl (1996) provide some encouraging results that suggest one does not require a high degree of grid resolution near walls in order to accurately predict heat transfer from the surface, and Moser in Schild *et al.* (1995) states that new models exist for natural convection which is grid independent. While natural convection and forced convection can be treated individually, wall functions for mixed convection cases are still lacking.

Jet Interaction With Room Air

A very common boundary condition for a room airflow is that of the air supply diffuser. While the interaction of a jet within a room is reasonably well understood, modelling the jet is a difficult task, especially if there are thermal gradients.

It is well known that jet behaviour is described by the Archimedes number (Ar). As Randall & Battams (1979) show, the stability of flow in a room is also described by Ar. In the investigation of RSM turbulence models to room flows, Chen (1996) tested the ability of the various models to predict jet penetration depth. Both the k-ϵ and RSM models predicted the penetration reasonably well at low Ar (RSM predicted high, k-ϵ low), but for high Ar both gave unacceptable results.

The Chen (1996) study was investigating the jet itself, and therefore a reasonably high degree of grid resolution was possible. In an industrial space, the size of an inlet grill might be on the order of 1.25 m × 0.5 m, while the length scale of the entire facility is on the order of 37 m. Providing a fine enough resolution near the grids in order to resolve fine flow details is restricted by computer memory and CPU resources. As a potential solution, Huo *et al.* (1996) suggest a means to handle diffuser boundary conditions that permits a coarse grid, but provides better flow simulation than a simple velocity source.

Validation with Field Data

One of the difficulties in any CFD industrial case study is the acquisition of accurate, representative data to be used to validate the CFD model. Only then can predictions be made of the influence of changes to the environment. In an ideal situation, the data would be taken over the course of a very short period of time, with respect to the time scales within the environment, and under steady state conditions. Unfortunately, this is not always possible. In addition, errors are likely to be present in the measurements. It is imperative that the roles that field measurement errors play in CFD boundary conditions and solution validation be well understood. This is to be addressed in Phillips & Roth (1998).

A second issue associated with field data is the difficulty in matching simulation data calculated at a point with measurements made in the field. For instance, a CFD solution would provide for velocities at all points within the flow domain, but the presence of an individual taking measurements in the field immediately alters the flow pattern. The degree of influence that this person has on the flow field is not always known. Additionally, there is some question as to the value of a point measurement in a ventilated space. Instead of matching very localized flow features in the simulation and real environment, it might be possible to devise a set of parameters that are more global, along the lines of Brouns & Waters (1994). In this study the authors use similarities in the ventilation effectiveness of regions in the flow domain as a criteria by which to choose tracer/air sampling locations.

SAMPLE RESULTS

The facility shown in Figure 1 has a fume haze that gathers in the North end. Figure 3 presents the velocity vectors across a plane 6.1 m from the North wall. It highlights a dead zone above and to the right of the scrap bay. The low velocities in this area are a partial cause of the haze problem. This particular simulation was performed without heat transfer, using the RNG k-ϵ turbulence formulation. While the results identify the cause of the problem, and a remedy can be investigated, completion of the investigation identified in this paper will improve the understanding that such results present.

CONCLUSIONS

The purpose of this paper has been to identify CFD modelling issues and potential

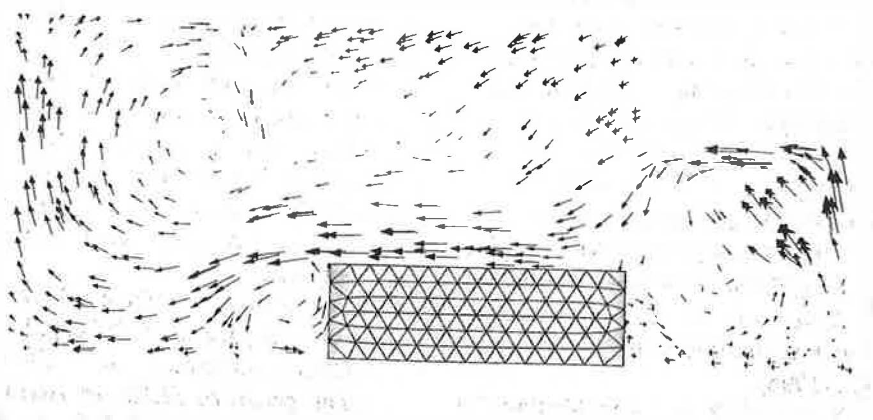

Figure 3 - Velocity Vectors at a Plane 6.1 m from the North Wall of the Facility in Figure 1

solutions for applications of CFD within large industrial premises. A number of items have been highlighted requiring study as part of this effort to increase reliability of CFD simulations in large industrial premises. These are:

- identifying a turbulence model that provides a sufficient degree of reliability in the prediction of low Reynolds number regimes, near wall effects, and recirculating flows;
- understanding the relative importance of long- and short- wave radiation in an industrial setting, to determine if an long-wave radiation can be omitted from simulations;
- identifying heat transfer correlations for natural, forced and mixed convection that accurately predict the heat transfer off a surface;
- testing methods to provide for jets from diffusers in a ventilated room in order to accurately model the penetration length without resorting to fine grids;
- identifying the role or magnitude of the effect that potential errors in boundary condition specification might play on the final solution.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the work of Michael Roth of Goodfellow Consultants Inc. who prepared the CFD grid to one of the case studies, and the Natural Sciences and Engineering Research Council (NSERC) of Canada which is contributing to this work through the Industrial Post-Doctoral Fellowship Programme.

REFERENCES

Brouns, C.E. & Waters, J.R., (1994), **The Positioning of Tracer Gas Sampling Points in Large Enclosures**, *IEA Annex 26: 4th Expert Meeting - Forum Presentation*, Gjøvik, May, 1994.

Chen, Q., (1996), **Prediction of Room Air Motion by Reynolds-Stress Models**, *Building and Environment*, Vol. 31, No. 3, pages 233 - 244.

Chen, Qingyan, (1997), **Computational Fluid Dynamics for HVAC: Successes and Failures**, *ASHRAE Transactions*, Vol. 103, Part I, pages 178 - 187.

Chen, Q., Moser, A. & Huber, A., (1990), **Prediction of Buoyant, Turbulent Flow by a Low-Reynolds-Number k-ϵ Model**, *ASHRAE Transactions*, Vol. 96, Part I, pages 564 - 573.

Chen, Qingyan, Peng, Xiudong, & van Paassen, A.C.H., (1995), **Prediction of Room Thermal Response by CFD Technique with Conjugate Heat Transfer and Radiation Models**, *ASHRAE Transactions*, Vol. 101, Part II, pages 50 - 60.

Dupuis, Marc, Derneede, Edgar & Clarke, Diane S., (1993), **Turbulence Modeling of Air Circulation in an Enclosure With Multiple openings and Local Heat Sources**, *Proceedings of the 32nd Annual Conference of the Canadian Institute of Mining and Metallurgy-CIM*, Bale, C.W. & Irons, G.A., Editors, Québec, QC: Aug 29th - Sept 2nd, 1993, pages 229 - 236.

Emmerich, Steven J., (1997), *Use of Computational Fluid Dynamics to Analyze Indoor Air Quality Issues - NISTIR Report # 5997*, Building and Fire Research Laboratory (BFRL), Gaithersburg, MD: April 1997.

Huo, Yan, Zhang, Jianshun, Shaw, Chiayu & Haghighat, Fariborz, (1996), **A New Method to Describe the Diffuser Boundary Conditions in CFD Simulation**, *ROOMVENT '96, Proceedings of the 5th International Conference on Air Distribution in Rooms*, Tokyo: July 17th - 19th, 1996, Vol. 2, pages 233 - 240.

Kato, Shinsuke, Murakami, Shuzo, Shoya, Satoshi, Hanyu, Futoshi & Zeng, Jie, (1995), **CFD Analysis of Flow and Temperature Fields in Atrium With Ceiling Height of 130 m**, *ASHRAE Transactions*, Vol. 101, Part II, pages

1144 - 1157.

Moser, Alfred, (1991),
The Message of Annex 20: Air Flow Patterns within Buildings, *Air Movement and Ventilation Control Within Buildings: Proceedings of the 12th Air Infiltration and Ventilation Centre Conference*, Ottawa: September 24th - 27th, 1991, Vol. 1, pages 1 - 26.

Moser, Alfred, (1993),
Energy Efficient Ventilation of Large Enclosures: Annex 26, *IEA Energy Conservation in Buildings and Community Systems Programme News*, Issue 18, October, 1993, pages 5 - 7.

Murakami, Shuzo & Kato, Shinsuke, (1989),
Numerical and Experimental Study on Room Airflow - 3-D Predictions Using the k-ϵ Turbulence Model, *Building and Environment*, Vol. 24, No. 1, pages 85 - 97.

Nielsen, Peter V., (1994),
Prospects for Computational Fluid Dynamics in Room Air Contaminant Control, *Ventilation '94: Proceedings of the 4th International Symposium on Ventilation for Contaminant Control*, Stockholm: September 5th - 9th, 1994, Part 1, pages 26 - 46.

Peng, Shia-hui, Davidson, Lars & Holmberg, Sture, (1996),
Performance of Two-Equation Turbulence Models for Numerical Simulation of Ventilation Flows, *ROOMVENT '96, Proceedings of the 5th International Conference on Air Distribution in Rooms*, Tokyo: July 17th - 19th, 1996, Vol. 2, pages 153 - 160.

Phillips, Duncan A. & Roth, Michael J. (1998),
The Sensitivity of Industrial Ventilation CFD Results to Errors in Field Measurements, to be presented at *CFD 98: The Sixth Annual Conference of the Computational Fluid Dynamics Society of Canada*, Quebec, QC: June 7th - 9th, 1998.

Randall, J.M. & Battams, V.A., (1979),
Stability Criteria for Airflow Patterns in Livestock Buildings, *Journal of Agricultural Engineering Research*, Vol. 24, pages 361 - 374.

Renz, Ulrich & Vogl, Norbert, (1996),
Numerical Predictions of Air Flow Patterns in Large Enclosures, *ROOMVENT '96, Proceedings of the 5th International Conference on Air Distribution in Rooms*, Tokyo: July 17th - 19th, 1996, Vol. 3, pages 163 - 170.

Rhie, C.M. & Chow, W.L., (1983),
Numerical Study of the Turbulent Flow Past and Airfoil with Trailing Edge Separation, *AIAA Journal*, Vol. 21, no. 11, pages 1525 - 1532.

Schild, Peter G., (1996),
CFD Analysis of an Atrium, Using a Conjugate Heat Transfer Model Incorporating Long-Wave and Solar Radiation, *ROOMVENT '96, Proceedings of the 5th International Conference on Air Distribution in Rooms*, Tokyo: July 17th - 19th, 1996, Vol. 2, pages 185 - 194.

Schild, Peter G., Tjeflaat, Per O. & Aiulfi, Dario, (1995),
Guidelines for Modelling Atria, *ASHRAE Transactions*, Vol 101, Part II, pages 1311 - 1332.

Thompson, Murray & Goodfellow, Howard, (1997),
Computational Fluid Dynamics as a Design Tool for Industrial Ventilation, *Ventilation '97: Global Developments in Industrial Ventilation - Proceedings of the 5th International Symposium on Ventilation for Contaminant Control*, Ottawa: September 14th - 17th, 1997, Vol. 1, pages 165 - 174.