A Forensic Study of a Ventilated Environment in a Manufacturing Facility

Goodfellow H D, Phillips D A
Stantec Global Technologies Ltd, Mississauga, Ontario, Canada

Abstract

Computational fluid dynamics (CFD) is a powerful means to investigate the flows within a ventilated space. There are a number of advantages to using CFD in indoor environmental design. Among these are:

- the ability to investigate flows where it is not possible to make direct observations (e.g. in an area too dangerous to enter, or within a component that has not yet been built);
- the option to investigate the resulting flow that would arise from changes to conditions; and,
- the flexibility to optimize a design change prior to implementation.

Many authors (e.g. 1, 2, 3) are demonstrating the capability of CFD to predict flow conditions prior to actual construction of, or modification to, the ventilated environment. CFD also has the ability to look into the past.

The purpose of this paper is to demonstrate the use of CFD in a forensic study of flow conditions and contaminant migration within an industrial facility. The environment of interest is a medium sized manufacturing facility that had a heavy gas spill.

There are a number of complications associated with this sort of problem. Foremost among them is that data collection for validation is not possible since the flow environment had been changed sufficiently since the event to preclude fieldwork. Additionally, the environment had an air-conditioning system in place that recirculated diluted air from within the facility. This contributed heavily to the mixing of the contaminant.

This paper will address the challenge associated with a forensic CFD study and HVAC return boundary conditions. Examples will be shown of the need for relaxation in the code that calculates the dilution factor and the implications for a parametric study of the flows.

Description of Manufacturing Facility

Figure 1 is an isometric sketch of the manufacturing facility. The data shows location of all processing equipment, building openings, and air handling units. In a forensic study, it is necessary to establish from previous drawings and reports a complete description of the facility at the time of the incident. A computational grid is established in order to provide sufficient detail on the pathways for a release of a contaminant. All air-handling units are located and parameters defined for the different operating modes.

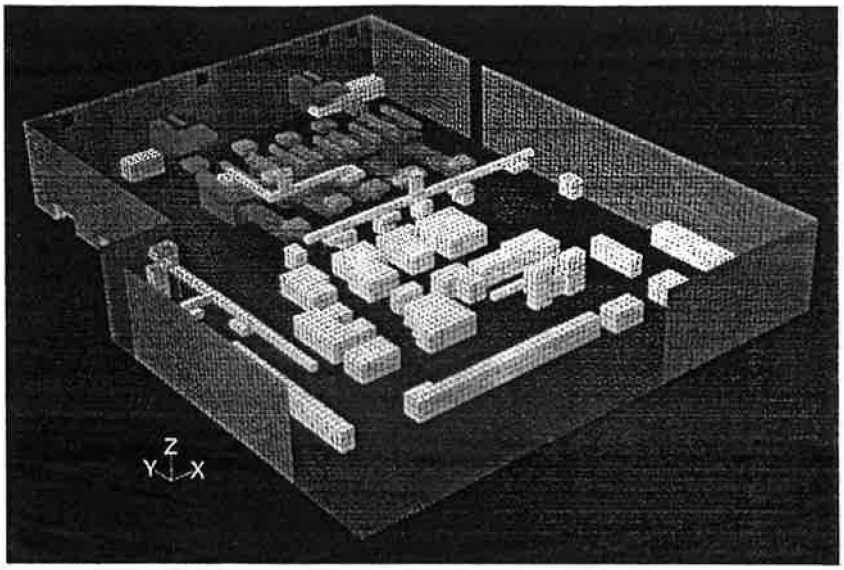

Figure 1. Isometric Sketch of Manufacturing Facility

Scope of CFD work

The information required is, as follows:

- drawings of plants, general arrangements, elevations
- locate all building openings, roof ventilators or exhausters
- reports on ventilation surveys
- changes to ventilation system
- details of contaminant release
- technical information on contaminant (physical and chemical properties)
- info on exposure levels for personnel

After a review of above information, the following steps are required:

- develop timeline and exposure (areas and personnel) for contaminant release
- develop CFD model to include all the ventilation units
- run model to establish different outflow patterns (Figure 2 shows different ventilation conditions)
- run model for different scenarios of contaminant release

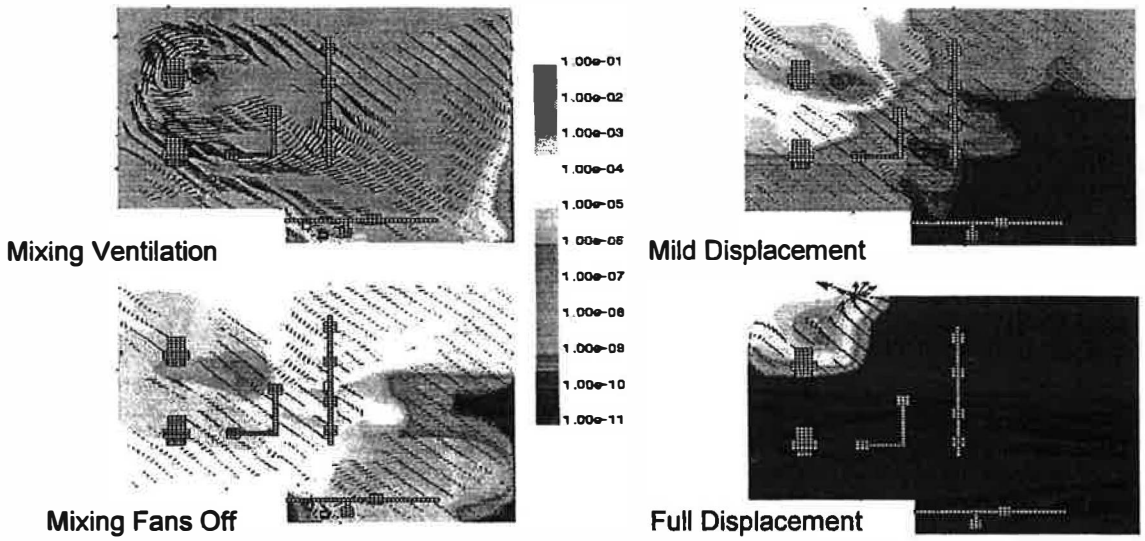

Figure 2. Forensic Study into an Industrial Facility

The different cases for CFD analysis are:

i. Prior to incident. Figure 3 shows the contours of mass fraction of contaminant prior to the release of the contaminant (i.e. contaminant spill)
ii. Prior to incident (reduced exhaust) Figure 4 shows typical concentrations at different locations in the facility
iii. At time of incident
iv. After incident

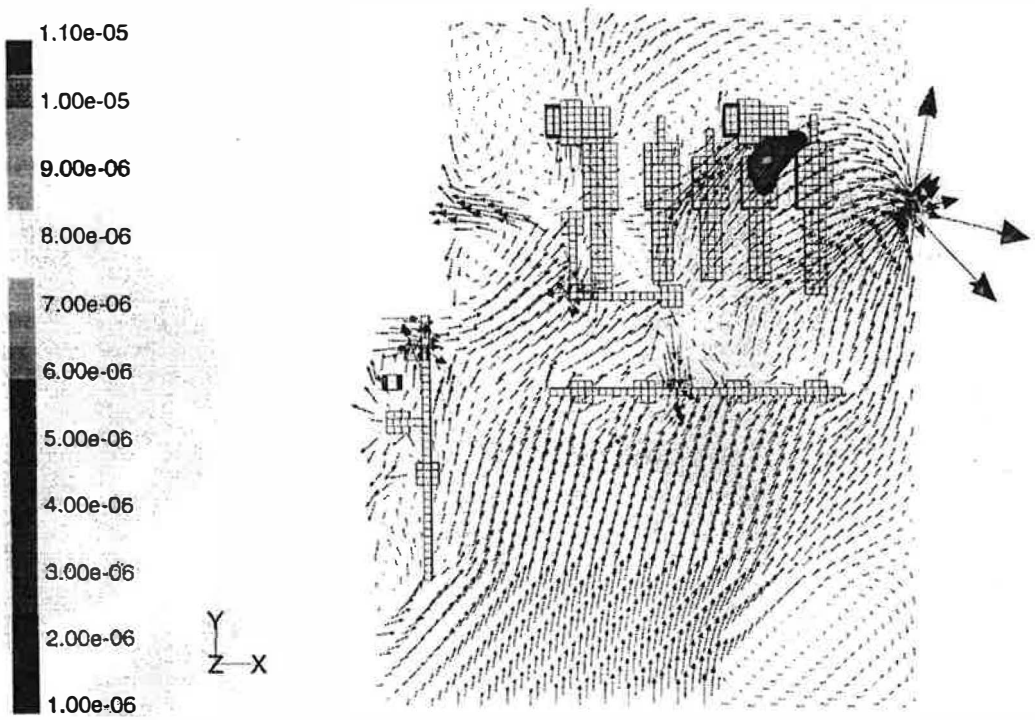

Figure 3. Contours of Mass Fraction of Contaminant

CFD runs can be carried out for these different operating conditions. It is possible to examine worst case scenarios and to establish possible exposure results.

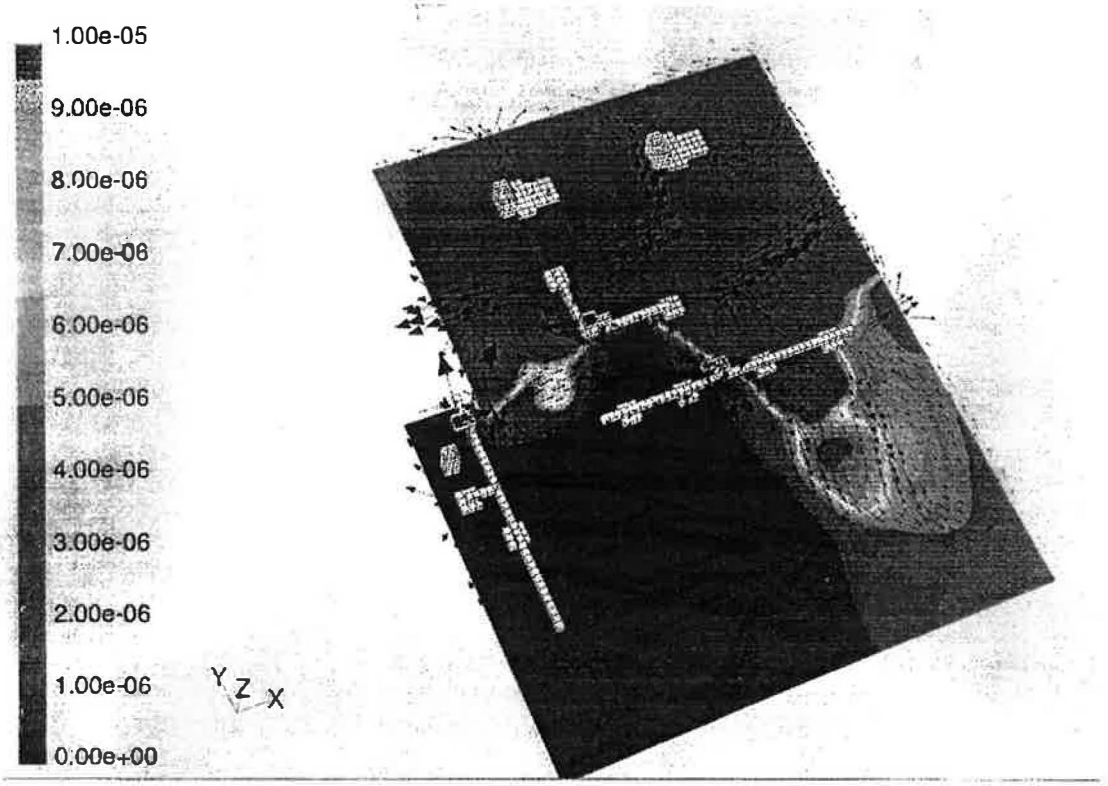

Figure 4. Contours of Mass Fraction of Contaminant

Conclusions

Detailed analyses of results from the CFD study can be a powerful tool for a forensic study of contaminant release and exposure.

References

1. Kurabuchi, T., Kobayashi, N., Arashiguchi, A. & Anai, T., (1998), Application of Wind Tunnel Experiment and CFD Simulation on Estimation of Wind Environment Inside and Outside a Large-Scale Building Complex with an Atrium Space, *ROOMVENT '98: Proceedings of the 6th International Conference on Air Distribution in Rooms*, Stockholm: June 14th - 17th, 1998, Vol. 2, paper 14_13.
2. Thomson, Murray & Goodfellow, Howard, (1997), Computational Fluid Dynamics as a Design Tool for Industrial Ventilation, *Ventilation '97 – Proceedings of the 5th International Symposium on Ventilation for Contaminant Control*, Ottawa: September 14th – 17th, 1997, pages 165 – 174.
3. Tsuji, Katsuhiko, Fukuhara, Isamu, Nakamura, Yasuhiro, & Mizuno, Minora (1997), Numerical Simulation of Push-Pull Ventilation for Controlling Buoyant Plume Metal Fumes in a Casting Plant, *Ventilation '97 – Proceedings of the 5th International Symposium on Ventilation for Contaminant Control*, Ottawa: September 14th – 17th, 1997, pages 309 – 318.