RESEARCH LABORATORY DESIGN: DOS AND DON'TS

Farhad Memarzadeh, Ph.D., P.E.

National Institutes of Health, Bethesda, MD, USA

ABSTRACT

This article briefly describes a research program undertaken by the National Institutes of Health (NIH), Office of Research Services to investigate ventilation performance of different laboratory configurations, and their affect on the hood. It focuses on some specific recommendations identified by the work which should help designers optimize performance. The intent is to provide a basis for guidelines to maximize lab hood containment performance, while minimizing the impact of the lab layout and ventilation system. Found here are only a small fraction of the recommendations contained in the 520 page NIH publication titled "Methodology for Optimization of Laboratory Hood Containment" which is now available on the Internet (http://des.od.nih.gov/farhad/cover.htm).

PROJECT SUMMARY

The research involved analyzing more than 250 laboratory configurations using Computational Fluid Dynamics (CFD). CFD is an advanced 3-D mathematical model, which computes the motion of air, water, or any other gas or liquid through, or around objects. The CFD program used was "FLO*VENT*®" by Flomerics for ventilation analysis.

There are an infinite number of combinations of lab and ventilation configurations; reaching a conclusion is almost impossible without a considerable focus on the interactive effects of the many parameters. NIH research team spent much of this project developing analysis techniques for identifying the relative performance of the hood for each configuration modeled, and determining links between the containment or loss and the design configuration.

Configuration parameters varied included are lab size, hood position, nominal hood face velocity, supply diffuser type, supply diffuser layout, room ventilation rate, makeup air, supply air temperature, and presence or absence of a scientist in front of the hood.

In order to analyze the hood performance in each particular room configuration three parameters were selected as the main indicators as follows. The leakage through the sash opening was quantified as the fraction of contamination released inside the hood leaking back against the flow into the laboratory and is termed the 'sash leakage factor', and was found to correlate well with the level of turbulence in the air. Leakage further out into the body of the laboratory was found to be a result not only of the turbulence causing leakage through the sash opening, but also air currents immediately outside the sash opening sweeping air from an imaginary 12 inch box out into the laboratory. This leakage was quantified in a similar way and termed the 'box leakage factor'. This performance of the box independent of the sash performance can also be defined as the proportion of the contaminant reaching the box to leak out through the box. This leakage was termed the 'box / sash leakage proportion'.

Note that these criteria cannot represent a pass fail criteria since they account for neither the source generation rate, nor the toxicity of the substance.

By measuring parameters characterizing the turbulence and the flow just outside the sash opening through the surfaces of the imaginary 305 mm box an experimental validation of existing installations can also be made.

COMPUTER SIMULATION

As noted above, the configurations were considered using CFD. In the mechanical ventilation design fraternity, this technique is commonly known as 'airflow modeling'. As a starting point in the description of airflow modeling, the actual physical processes should first be considered. Air flow and heat transfer within a fluid are governed by the principles of conservation of mass, momentum and thermal energy. The equation that governs this physics is commonly know as the Navier-Stokes equation. A Finite Volume approach was used here, requiring the region being modeled, in this case the plenum, to be sub-divided into a number of small volumes or grid cells.

During the program solution, the CFD software integrates the relevant differential conservation equations over each computational grid cell, assembling a set of algebraic equations. Items from the physical situation, for example, the supplies into the laboratory, the workbenches, hoods, etc, provide so called boundary conditions to these equations. The equations relate the value of the variable in a cell to the value in adjacent cells. Since the equations display strong coupling (variables are dependent upon surrounding values and other variables) the solution is carried out iteratively.

To verify the accuracy of CFD software, predictions obtained from the software are often compared with appropriate experimental data. These comparisons show that, for well defined conditions, the environment can be well predicted in most cases, and in some instances it is difficult to determine whether discrepancies occur as a result of numerical or experimental inaccuracy. However, even where the boundary conditions are less well defined, the prediction provides more qualitative than quantitative results allowing for parametric design study.

RESULTS

The research project presents the results of the CFD simulations in a number of ways. These can be either based upon visualization of the CFD results file, or, upon an automated analysis of the data in terms of containment performance. The former, in the form of flow diagrams, provides a qualitative approach, which is helpful in understanding. The flow diagrams show the flow from an imaginary particle source where the particles follow the air streamlines and change color according to air speed. After a given time the particles disappear thus preventing the room filling with particles. The latter provides the quantitative measures of the leakage as described above. The containment performance of different configurations are shown using scatter diagrams of sash leakage factor v. box / sash leakage proportion.

CONCLUSIONS

The following recommendations can be made based on the results, which can be seen complete in the published report.

Hood Position: Protect the hood by placing it in a corner avoiding jets impinging on the working zone outside the sash opening.

Bulkhead: A bulkhead can be used to improve the containment performance by either:

- using a diffuser layout that will gently feed low velocity air into the hood;
- avoiding use of a diffuser layout which generates thin jets across the face of the hood from above;
- avoid using down-flow diffusers that cause a circulation in front of the hood so that the jet does not impinge.

Diffuser Blanking: Avoid diffuser blanking where the increased velocity jets have a path back to the hood.

Diffuser/ Hood Position: Avoid placing a square diffuser asymmetrically in front of the hood, since this increase exposure to the scientist by increasing sash leakage.

Diffuser/ Hood Separation: Where there is insufficient distance to move the diffuser well away from the hood in line with current guidance, position the diffuser in line with the center of the hood, close to the bulkhead to prevent the square diffuser jet blowing towards the diffuser.

Hood Separation/ Same Wall: Place hoods at least 102 mm apart, preferably selecting corner positions if available.

Hood Separation/ Opposite Walls: For hoods on opposite walls, avoid opposite or 51 mm separations of the hood.

Hood Separations/ Perpendicular Walls: Separate hoods by more that 102 mm. Placing two hoods on perpendicular walls is likely to produce a better performance than on opposite walls. In turn, either of these configurations can be expected to achieve lower leakage than hoods on the same wall.

REFRENCES

Fletcher, C.A. 1988. *Computational Techniques for Fluid Dynamics, Voll II.* Berlin.

Memarzadeh, F. 1996. *Methodology for optimization of laboratory hood Containment.* National Institutes of Health , Bethesda, Maryland.